



QGIS Desktop 3.16 User Guide

QGIS Project

2022 年 03 月 24 日

1	导言	1
1.1	QGIS 3.16 更新内容	2
2	序言	3
3	约定	5
3.1	图形用户界面约定	5
3.2	文本或键盘约定	6
3.3	特定于平台的说明	6
4	特性	7
4.1	视图数据	7
4.2	浏览数据与制图	8
4.3	创建、编辑、管理与输出数据	8
4.4	分析数据	9
4.5	在互联网上发布地图	9
4.6	通过插件扩展 QGIS 功能	9
4.6.1	核心插件	9
4.6.2	外部 Python 插件	9
4.7	Python 终端	10
4.8	已知问题	10
4.8.1	打开文件数量限制	10
5	入门指南	11
5.1	安装 QGIS	11
5.1.1	二进制文件安装	11
5.1.2	从源代码安装	11
5.1.3	安装到外媒介	12
5.1.4	下载示例数据	12
5.2	启动和关闭 QGIS	12
5.3	案例：加载栅格图层和矢量图层	13
6	使用工程文件	19
6.1	QGIS 工程简介	19
6.2	处理损坏的文件路径	21
6.3	生成输出	21
7	QGIS 图形用户界面	23
7.1	菜单栏	24
7.1.1	工程	24
7.1.2	编辑	25

7.1.3	视图	29
7.1.4	图层	31
7.1.5	设置	33
7.1.6	插件	33
7.1.7	矢量	34
7.1.8	栅格	35
7.1.9	数据库	36
7.1.10	Web	36
7.1.11	网孔	36
7.1.12	空间运算	36
7.1.13	帮助	37
7.1.14	QGIS	37
7.2	面板和工具栏	37
7.2.1	工具栏	37
7.2.2	面板	39
7.3	地图视图	40
7.3.1	探索地图视图	40
7.3.2	设置其他地图视图	41
7.3.3	导出地图视图	42
7.4	3D 地图视图	44
7.4.1	导航选项	45
7.4.2	创建动画	46
7.4.3	场景配置	47
7.4.4	3D 矢量图层	49
7.5	状态栏	49
7.5.1	搜索栏 - 定位器	49
7.5.2	汇报行为	50
7.5.3	控制地图画布	50
7.5.4	实时信息	51
8	浏览器面板	53
8.1	从浏览器打开或运行资源	55
8.2	浏览器面板顶级条目	56
8.2.1	收藏夹	56
8.2.2	空间书签	56
8.2.3	主目录	56
8.2.4	/	56
8.2.5	Geopackage	56
8.2.6	Spatialite	57
8.2.7	PostGIS	57
8.2.8	MSSQL	57
8.2.9	DB2	58
8.2.10	WMS/WMTS	58
8.2.11	矢量瓦片	58
8.2.12	XYZ 瓦片	58
8.2.13	WCS	59
8.2.14	WFS / OGC API - 要素	59
8.2.15	OWS	59
8.2.16	ArcGIS 地图服务器	59
8.2.17	ArcGIS 要素服务器	59
8.2.18	GeoNode	59
8.3	资源	60
9	QGIS 配置	61
9.1	选项	61
9.1.1	通用设置	62
9.1.2	系统设置	63
9.1.3	CRS 设置	65

9.1.4	变换设置	66
9.1.5	数据源设置	67
9.1.6	渲染设置	69
9.1.7	画布和图例设置	71
9.1.8	地图工具设置	72
9.1.9	颜色设置	73
9.1.10	数字化设置	74
9.1.11	布局设置	76
9.1.12	GDAL 设置	76
9.1.13	变量设置	79
9.1.14	认证设置	80
9.1.15	网络设置	81
9.1.16	搜索设置	82
9.1.17	高级设置	83
9.1.18	加速设置	84
9.1.19	数据处理设置	85
9.1.20	Python 控制台设置	85
9.1.21	代码编辑器设置	87
9.2	使用用户配置文件	88
9.3	工程属性	88
9.3.1	通用属性	89
9.3.2	元数据属性	90
9.3.3	CRS 属性	91
9.3.4	变换属性	91
9.3.5	默认样式属性	91
9.3.6	数据源属性	92
9.3.7	Relations Properties	93
9.3.8	Variables Properties	94
9.3.9	Macros Properties	94
9.3.10	QGIS Server Properties	95
9.3.11	Temporal Properties	95
9.4	Customization	96
9.5	Keyboard shortcuts	98
9.6	Running QGIS with advanced settings	100
9.6.1	Command line and environment variables	100
9.6.2	Deploying QGIS within an organization	104
10	Working with Projections	107
10.1	Overview of Projection Support	107
10.2	Layer Coordinate Reference Systems	107
10.3	Project Coordinate Reference Systems	109
10.4	Coordinate Reference System Selector	110
10.5	Custom Coordinate Reference System	111
10.5.1	Integrate an NTV2-transformation in QGIS	112
10.6	Datum Transformations	113
11	常用工具	115
11.1	上下文帮助	115
11.2	面板	115
11.2.1	Layers Panel	115
11.2.2	Layer Styling Panel	120
11.2.3	Layer Order Panel	123
11.2.4	Overview Panel	123
11.2.5	Log Messages Panel	124
11.2.6	Undo/Redo Panel	124
11.2.7	Statistical Summary Panel	124
11.3	Nesting Projects	125
11.4	Working with the map canvas	126

11.4.1	Rendering	126
11.4.2	Zooming and Panning	128
11.4.3	Spatial Bookmarks	129
11.4.4	Decorations	131
11.4.5	Annotation Tools	138
11.4.6	Measuring	141
11.5	Interacting with features	143
11.5.1	Selecting features	143
11.5.2	Identifying Features	146
11.6	Save and Share Layer Properties	150
11.6.1	Managing Custom Styles	150
11.6.2	Storing Styles in a File or a Database	151
11.6.3	Layer definition file	153
11.7	Storing values in Variables	153
11.8	Authentication	154
11.9	Common widgets	155
11.9.1	Color Selector	155
11.9.2	Symbol Widget	159
11.9.3	Font Selector	159
11.9.4	Unit Selector	160
11.9.5	Number Formatting	160
11.9.6	Blending Modes	161
11.9.7	Data defined override setup	162
12	样式库	165
12.1	The Style Manager	165
12.1.1	The Style Manager dialog	165
12.1.2	Setting a Color Ramp	170
12.2	The Symbol Selector	172
12.2.1	The symbol layer tree	173
12.2.2	Configuring a symbol	174
12.3	Setting a label	182
12.3.1	Formatting the label text	184
12.3.2	Configuring interaction with labels	191
12.4	Creating 3D Symbols	198
12.4.1	Point Layers	199
12.4.2	Line layers	200
12.4.3	Polygon Layers	201
12.4.4	Application example	202
13	管理数据源	203
13.1	Opening Data	203
13.1.1	The Browser Panel	204
13.1.2	The DB Manager	208
13.1.3	Provider-based loading tools	209
13.1.4	QGIS Custom formats	225
13.1.5	QLR - QGIS Layer Definition File	225
13.1.6	Connecting to web services	225
13.2	Creating Layers	228
13.2.1	Creating new vector layers	228
13.2.2	Creating new layers from an existing layer	234
13.2.3	Creating new DXF files	238
13.2.4	Creating new layers from the clipboard	239
13.2.5	Creating virtual layers	240
13.3	Exploring Data Formats and Fields	242
13.3.1	Raster data	242
13.3.2	Vector Data	243

14 使用矢量数据	251
14.1 The Vector Properties Dialog	251
14.1.1 Information Properties	252
14.1.2 Source Properties	252
14.1.3 Symbology Properties	255
14.1.4 Labels Properties	277
14.1.5 Diagrams Properties	287
14.1.6 Masks Properties	292
14.1.7 3D View Properties	292
14.1.8 Fields Properties	293
14.1.9 Attributes Form Properties	294
14.1.10 Joins Properties	302
14.1.11 Auxiliary Storage Properties	304
14.1.12 Actions Properties	313
14.1.13 Display Properties	318
14.1.14 Rendering Properties	320
14.1.15 Variables Properties	321
14.1.16 Metadata Properties	321
14.1.17 Dependencies Properties	321
14.1.18 Legend Properties	322
14.1.19 QGIS Server Properties	322
14.1.20 Digitizing Properties	323
14.2 Expressions	326
14.2.1 The Expression string builder	326
14.2.2 Function Editor	331
14.3 功能列表	333
14.3.1 Aggregates Functions	333
14.3.2 Array Functions	343
14.3.3 Color Functions	351
14.3.4 Conditional Functions	357
14.3.5 Conversions Functions	360
14.3.6 Custom Functions	365
14.3.7 Date and Time Functions	366
14.3.8 Fields and Values	376
14.3.9 Files and Paths Functions	376
14.3.10 Form Functions	379
14.3.11 Fuzzy Matching Functions	379
14.3.12 General Functions	381
14.3.13 Geometry Functions	384
14.3.14 Layout Functions	433
14.3.15 Map Layers	433
14.3.16 Maps Functions	434
14.3.17 Mathematical Functions	438
14.3.18 Operators	447
14.3.19 Processing Functions	448
14.3.20 Rasters Functions	448
14.3.21 Record and Attributes Functions	449
14.3.22 Relations	456
14.3.23 String Functions	456
14.3.24 User Expressions	465
14.3.25 Variables	465
14.3.26 Recent Functions	469
14.4 属性表操作	469
14.4.1 前言：空间与非空间属性表	469
14.4.2 属性表界面介绍	469
14.4.3 Interacting with features in an attribute table	474
14.4.4 Using action on features	476
14.4.5 Editing attribute values	478

14.4.6	Creating one or many to many relations	481
14.5	Editing	490
14.5.1	Setting the Snapping Tolerance and Search Radius	490
14.5.2	Topological editing	492
14.5.3	Digitizing an existing layer	494
14.5.4	Advanced digitizing	502
14.5.5	Shape digitizing	509
14.5.6	The Advanced Digitizing panel	512
14.5.7	The Processing in-place layer modifier	518
15	使用栅格数据	521
15.1	Raster Properties Dialog	521
15.1.1	Information Properties	522
15.1.2	Source Properties	522
15.1.3	Symbology Properties	523
15.1.4	Transparency Properties	530
15.1.5	Histogram Properties	532
15.1.6	Rendering Properties	533
15.1.7	Pyramids Properties	533
15.1.8	Metadata Properties	534
15.1.9	Legend Properties	535
15.1.10	QGIS Server Properties	536
15.2	Raster Analysis	537
15.2.1	Raster Calculator	537
15.2.2	Raster Alignment	539
15.3	Georeferencer	541
15.3.1	Usual procedure	542
16	Working with Mesh Data	547
16.1	What's a mesh?	547
16.2	Supported formats	549
16.3	Mesh Dataset Properties	550
16.3.1	Information Properties	550
16.3.2	Source Properties	550
16.3.3	Symbology Properties	550
17	Working with Vector Tiles	557
17.1	What are Vector Tiles?	557
17.2	Supported Formats	558
18	Laying out the maps	559
18.1	Overview of the Print Layout	559
18.1.1	Sample Session for beginners	559
18.1.2	The Layout Manager	560
18.1.3	Menus, tools and panels of the print layout	561
18.2	布局项	574
18.2.1	Layout Items Common Options	574
18.2.2	The Map Item	579
18.2.3	The 3D Map Item	588
18.2.4	The Label Item	589
18.2.5	The Legend Item	592
18.2.6	The Scale Bar Item	599
18.2.7	The Table Items	602
18.2.8	The Picture and the North Arrow Items	611
18.2.9	The HTML Frame Item	613
18.2.10	图形项	616
18.3	Creating an Output	619
18.3.1	Export settings	619
18.3.2	Export as Image	620

18.3.3	Export as SVG	621
18.3.4	Export as PDF	622
18.3.5	Generate an Atlas	623
18.4	Creating a Report	629
18.4.1	What is it?	629
18.4.2	Get started	629
18.4.3	Layout Report Workspace	630
18.4.4	Export settings	645
19	Working with OGC / ISO protocols	647
19.1	WMS/WMTS Client	648
19.1.1	Overview of WMS Support	648
19.1.2	Overview of WMTS Support	648
19.1.3	Selecting WMS/WMTS Servers	649
19.1.4	Loading WMS/WMTS Layers	651
19.1.5	图块集	653
19.1.6	Using the Identify Tool	653
19.1.7	Show WMS legend graphic in table of contents and layout	655
19.1.8	WMS Client Limitations	655
19.2	WCS Client	655
19.3	WFS and WFS-T Client	656
20	使用 GPS 数据	659
20.1	GPS Plugin	659
20.1.1	What is GPS?	659
20.1.2	Loading GPS data from a file	659
20.1.3	GPSTools	660
20.1.4	Importing GPS data	660
20.1.5	Downloading GPS data from a device	661
20.1.6	Uploading GPS data to a device	661
20.1.7	Defining new device types	661
20.1.8	Download of points/tracks from GPS units	662
20.2	Live GPS tracking	663
20.2.1	Position and additional attributes	664
20.2.2	GPS signal strength	665
20.2.3	GPS options	665
20.2.4	Connect to a Bluetooth GPS for live tracking	667
20.2.5	Using GPSTools 60cs	667
20.2.6	Using BTGP-38KM datalogger (only Bluetooth)	668
20.2.7	Using BlueMax GPS-4044 datalogger (both BT and USB)	668
21	认证系统	669
21.1	Authentication System Overview	669
21.1.1	Authentication database	669
21.1.2	Master password	670
21.1.3	Authentication Configurations	671
21.1.4	Authentication Methods	673
21.1.5	Master Password and Auth Config Utilities	677
21.1.6	Using authentication configurations	678
21.1.7	Python bindings	678
21.2	User Authentication Workflows	679
21.2.1	HTTP(S) authentication	679
21.2.2	Database authentication	680
21.2.3	PKI authentication	681
21.2.4	Handling bad layers	688
21.2.5	Changing authentication config ID	689
21.2.6	QGIS Server support	690
21.2.7	SSL server exceptions	690
21.3	Security Considerations	693

21.3.1	Restrictions	694
22	GRASS GIS Integration	695
22.1	Demo dataset	695
22.2	Loading GRASS raster and vector layers	695
22.3	Importing data into a GRASS LOCATION via drag and drop	696
22.4	Managing GRASS data in QGIS Browser	696
22.5	GRASS Options	696
22.6	Starting the GRASS plugin	696
22.7	Opening GRASS mapset	697
22.8	GRASS LOCATION and MAPSET	697
22.9	Importing data into a GRASS LOCATION	697
22.9.1	Creating a new GRASS LOCATION	698
22.9.2	Adding a new MAPSET	699
22.10	The GRASS vector data model	700
22.11	Creating a new GRASS vector layer	700
22.12	Digitizing and editing a GRASS vector layer	701
22.13	The GRASS region tool	703
22.14	The GRASS Toolbox	703
22.14.1	Working with GRASS modules	704
22.14.2	GRASS module examples	707
22.14.3	Customizing the GRASS Toolbox	710
23	QGIS 处理框架	713
23.1	简介	713
23.2	Configuring the Processing Framework	715
23.3	The Toolbox	717
23.3.1	The algorithm dialog	718
23.3.2	Data objects generated by algorithms	723
23.4	The history manager	724
23.4.1	The processing history	724
23.4.2	The processing log	725
23.5	The graphical modeler	725
23.5.1	Definition of inputs	726
23.5.2	Definition of the workflow	729
23.5.3	Interacting with the canvas and elements	732
23.5.4	Saving and loading models	734
23.5.5	Editing a model	735
23.5.6	Editing model help files and meta-information	736
23.5.7	Exporting a model as a Python script	738
23.5.8	About available algorithms	738
23.6	The batch processing interface	738
23.6.1	简介	738
23.6.2	The parameters table	739
23.6.3	Filling the parameters table	740
23.6.4	Executing the batch process	741
23.7	Using processing algorithms from the console	741
23.7.1	Calling algorithms from the Python console	741
23.7.2	Creating scripts and running them from the toolbox	746
23.7.3	Pre- and post-execution script hooks	749
23.8	Using processing from the command line	749
23.9	Writing new Processing algorithms as Python scripts	750
23.9.1	Extending QgsProcessingAlgorithm	751
23.9.2	The @alg decorator	755
23.9.3	Input and output types for Processing Algorithms	756
23.9.4	Handing algorithm output	758
23.9.5	Communicating with the user	758
23.9.6	Documenting your scripts	759

23.9.7	Flags	759
23.9.8	Best practices for writing script algorithms	759
23.10	Configuring external applications	759
23.10.1	A note for Windows users	760
23.10.2	A note on file formats	760
23.10.3	A note on vector layer selections	760
23.10.4	SAGA	760
23.10.5	R scripts	761
23.10.6	R libraries	769
23.10.7	GRASS	769
23.10.8	激光雷达处理工具 (LAStools)	770
23.10.9	OTB Applications	770
24	数据处理提供程序和算法	773
24.1	QGIS algorithm provider	773
24.1.1	Cartography	773
24.1.2	Database	788
24.1.3	File tools	794
24.1.4	Interpolation	795
24.1.5	Layer tools	805
24.1.6	Modeler tools	806
24.1.7	Network analysis	811
24.1.8	Plots	822
24.1.9	栅格分析	828
24.1.10	Raster Creation	866
24.1.11	Raster terrain analysis	879
24.1.12	Raster tools	890
24.1.13	Vector analysis	895
24.1.14	Vector creation	913
24.1.15	Vector general	936
24.1.16	Vector geometry	966
24.1.17	Vector overlay	1078
24.1.18	Vector selection	1091
24.1.19	Vector table	1104
24.1.20	Vector Tiles	1117
24.2	GDAL algorithm provider	1120
24.2.1	Raster analysis	1120
24.2.2	Raster conversion	1143
24.2.3	Raster extraction	1149
24.2.4	Raster miscellaneous	1155
24.2.5	Raster projections	1172
24.2.6	Vector conversion	1176
24.2.7	Vector geoprocessing	1181
24.2.8	Vector miscellaneous	1187
24.3	LAStools algorithm provider	1196
24.3.1	blast2dem	1196
24.3.2	blast2iso	1198
24.3.3	las2dem	1199
24.3.4	las2iso	1201
24.3.5	las2las_filter	1203
24.3.6	las2las_project	1207
24.3.7	las2las_transform	1213
24.3.8	las2txt	1216
24.3.9	lasindex	1217
24.3.10	lasgrid	1218
24.3.11	lasinfo	1220
24.3.12	lasmerge	1223
24.3.13	lasprecision	1224

24.3.14	lasquery	1225
24.3.15	激光雷达数据校验 (lasvalidate)	1226
24.3.16	laszip	1226
24.3.17	txt2las	1227
24.4	TauDEM algorithm provider	1231
24.4.1	Basic Grid Analysis	1232
24.4.2	Specialized Grid Analysis	1244
24.4.3	Stream Network Analysis	1268
24.5	OTB 应用程序提供者	1286
25	插件	1287
25.1	QGIS 插件	1287
25.1.1	Core and External plugins	1287
25.1.2	The Plugins Dialog	1288
25.2	使用 QGIS 核心插件	1293
25.2.1	DB Manager Plugin	1293
25.2.2	Geometry Checker Plugin	1296
25.2.3	MetaSearch Catalog Client	1300
25.2.4	Offline Editing Plugin	1307
25.2.5	Topology Checker Plugin	1309
25.3	QGIS Python console	1311
25.3.1	The Interactive Console	1311
25.3.2	The Code Editor	1312
26	帮助和支持	1315
26.1	Mailing lists	1315
26.1.1	QGIS Users	1315
26.1.2	QGIS Developers	1315
26.1.3	QGIS Community Team	1315
26.1.4	QGIS Translations	1316
26.1.5	QGIS Project Steering Committee (PSC)	1316
26.1.6	QGIS User groups	1316
26.2	IRC	1316
26.3	Commercial support	1316
26.4	BugTracker	1316
26.5	Blog	1317
26.6	Plugins	1317
26.7	Wiki	1317
27	贡献者	1319
27.1	作者	1319
27.2	翻译者	1320
27.3	翻译统计	1320
28	附录	1323
28.1	附录 A: GNU 通用公共许可证 (GUN GPL)	1323
28.2	Appendix B: GNU Free Documentation License	1326
28.3	Appendix C: QGIS File Formats	1331
28.3.1	QGS/QGZ - QGIS 工程文件格式	1331
28.3.2	QLR - QGIS 图层定义文件定义	1333
28.3.3	QML - QGIS 样式文件格式	1334
28.4	Appendix D: QGIS R script syntax	1335
28.4.1	Inputs	1336
28.4.2	Outputs	1336
28.4.3	Syntax Summary for QGIS R scripts	1336
28.4.4	Examples	1338
29	Literature and Web References	1341

这是地理信息系统 (GIS) 软件 QGIS 的用户指南，QGIS 受 GNU 通用公共许可证的约束。更多信息请访问 QGIS 主页 <https://www.qgis.org>。

本文档的内容是作者和编辑对 QGIS 相关知识进行系统的编写和详细认真的验证，即使如此，错误还是不可避免的存在。

因此，作者、编辑和发布者不对文档的错误以及可能导致的后果承担任何责任。我们鼓励您报告可能存在的错误。

本文档已使用 reStructuredText 排版。可以在 [github](https://github.com) 网站下载 reST 格式代码，也可以通过网址 <https://www.qgis.org/en/docs/> 在线查看获得 HTML 或者 PDF 格式文档。该文档的翻译版本也可以通过 QGIS 项目的文档区浏览和下载。

有关为本文档做出贡献和翻译的更多信息，请访问 <https://qgis.org/en/site/getinvolved/index.html>。

本文档中的链接

该文档包含内部和外部链接。单击内部链接将在文档内移动，而单击外部链接将打开互联网地址。

文档作者与编辑者

对以下文档的撰写、审查和翻译作出贡献的人员名单可在[贡献者](#) 获得。

版权所有 (c) 2004 - 2020 QGIS 开发组

互联网： <https://www.qgis.org>

文档许可

允许您根据 GNU 自由文档许可证 1.3 版或自由软件基金会发布的任何后续版本的条款，进行复制、分发和 (或) 修改本文档，这没有固定的章节，封面和封底文本。该许可的副本包含在附录 [Appendix B: GNU Free Documentation License](#) 中。

1.1 QGIS 3.16 更新内容

与 QGIS 3.10 相比，该版本的 QGIS 包含了数百个 Bug 修复，增加许多新特性和增强功能，我们建议您使用此版本而不是以前的版本。有关新功能列表，请访问 <https://qgis.org/en/site/forusers/visualchangelogs.html> 查看更新日志。

欢迎来到地理信息系统 (GIS) 的奇妙世界!

QGIS 是一款开源地理信息系统。该项目于 2002 年 5 月诞生，同年 6 月作为 SourceForge 上的一个项目建立。我们一直在努力使任何有权使用个人电脑的人都能使用地理信息系统软件（传统上是昂贵的专有软件）。QGIS 目前运行在大多数 Unix 平台、Windows 和 macOS 上。QGIS 是使用 Qt 工具包 (<https://www.qt.io>) 和 C++ 开发的。这意味着 QGIS 感觉很快，并且有一个令人愉快、易于使用的图形用户界面 (GUI)。

QGIS 旨在成为一个用户友好的地理信息系统，提供常用的功能和特性。该项目的最初目标是提供一个 GIS 数据查看器。QGIS 在其发展过程中已经达到了这样一个阶段：它被用于日常 GIS 数据查看需求、数据捕获、高级 GIS 分析以及以复杂的地图、地图册和报告的形式呈现。QGIS 支持大量的栅格和矢量数据格式，使用插件架构可以轻松添加新的格式支持。

QGIS 是根据 GNU 通用公共许可证 (GPL) 发布的。根据此许可证开发 QGIS 意味着您可以检查和修改源代码，并保证您，我们的快乐用户，将始终能够访问一个免费和可以自由修改的 GIS 程序。您应该已经收到一份完整的许可证副本和您的 QGIS 副本，您也可以在附录附录 A: *GNU 通用公共许可证 (GNU GPL)*。

小技巧: 更新文档

您能在 QGIS 网站的文档区域找到该文档的最新版本，QGIS 网站地址为: <https://www.qgis.org/en/docs/>

该节介绍本手册将使用的统一样式。

3.1 图形用户界面约定

图形用户界面约定样式旨在模拟 GUI 的外观。通常，样式将反映非悬停外观，因此用户可以直观地浏览 GUI，以找到与手册中的说明类似的内容。

- 菜单选项: 图层 ▶ 添加栅格图层或 设置 ▶ 工具栏 ▶ 数字化

- 工具:  添加栅格图层

- 按钮: 另存为默认

- 对话框标题: 图层属性

- 选项卡: 常规

- 复选框: 渲染

- 单选按钮: *Postgis SRID* *EPSG ID*

- 选择数字:

- 选择字符串:

- 浏览文件: ...

- 选择颜色:

- 滑块:

- 输入文本: Display name

阴影表示可单击的 GUI 组件。

3.2 文本或键盘约定



本手册还包括与文本、键盘命令和编码相关的样式，以提示不同的对象，例如类或方法。这些样式与 QGIS 中任何文本或编码的实际外观不一致。

- 超链接: <https://qgis.org>
- 按键组合: 按 Ctrl+B, 意思是先按住 Ctrl 键, 再按 B 键。
- 文件名: lakes.shp
- 类名: **NewLayer**
- 方法: *classFactory*
- 服务器: *myhost.de*
- 用户文本: `qgis --help`



代码行由固定宽度的字体表示:

```
PROJCS["NAD_1927_Albers",
  GEOGCS["GCS_North_American_1927",
```


3.3 特定于平台的说明


图形用户界面顺序和少量文本可以内联格式化: 单击   文件 **X** *QGIS* ► 退出关闭 *QGIS*. 这表明在 Linux、Unix 和 Windows 平台上, 应先单击文件菜单, 然后退出; 而在 macOS 平台上, 应先单击 QGIS 菜单, 然后退出。

大量文本可以格式化为列表:

-  Do this
-  Do that
- **X** Or do that

或作为段落:

 **X** 示例文本示例文本示例文本示例文本示例文本示例文本示例文本

 示例文本示例文本示例文本示例文本示例文本示例文本示例文本示例文本示例文本示例文本示例文本示例文本示例文本示例文本示例文本。

出现在使用手册中的屏幕截图是在不同平台上创建的。

QGIS 通过核心特性和插件提供了丰富的 GIS 功能。定位器条使得搜索各功能模块，数据集和其他信息变得更加方便。

下面简要概述了六大类的特性和插件，然后是对集成的 Python 控制台的初步了解。

4.1 视图数据

您可以将不同格式、不同投影方式的矢量与栅格数据（以 2D 或 3D 形式）的组合显示，且无需转换为内部或通用格式。受支持的格式包括：

- 使用 PostGIS、Spatialite, MS SQL Spatial 和 Oracle Spatial 的，支持空间属性的表格或者视图；由安装的 OGR 库支持的矢量格式，包括 GeoPackage, ESRI Shapefile, MapInfo, SDTS, GML 等等。参见[使用矢量数据](#)。
- 由安装的 GDAL 库（地理空间数据抽象库）支持的栅格和图片格式，例如 GeoTIFF, ERDAS IMG, ArcInfo ASCII GRID, JPEG, PNG 等。参见[:ref:working_with_raster](#)。
- 网格数据（支持 TINs 等常规网格格式）。参见章节[:ref:label_meshdata](#)。
- 矢量图块
- GRASS 数据库（本地/地图集）中的 GRASS 栅格与矢量数据。参见[:ref:sec_grass](#)。
- 以 OGC Web 服务形式提供的在线空间数据，包括 WMS, WMTS, WCS, WFS, 与 WFS-T。参见章节[:ref:working_with_ogc](#)。

QGIS 的身份认证架构可以帮助你管理用于 Web 服务和其他资源的用户名/密码、证书以及密钥。

- 表格文件（ODS/XLSX）

支持时态数据。

4.2 浏览数据与制图

您可以使用友好的用户界面进行制图与交互式浏览空间数据。用户界面上提供了许多非常有用的工具，包括：

- QGIS 浏览器
- 动态二次投影
- 数据库管理器
- 打印布局
- 报告
- 全览面板
- 地理书签
- 注记工具
- 识别/选择要素
- 编辑/查看/搜索属性
- 基于数据定义的要素标注
- 基于数据定义的矢量与栅格符号工具
- 带网格层的地图集制图
- 地图的指北针、比例尺与版权标注
- 支持项目的保存与恢复

4.3 创建、编辑、管理与输出数据

你可以创建、编辑、管理与输出多种格式的矢量和栅格图层。QGIS 提供了以下功能：

- 矢量数字化工具
- 能够创建并编辑多种格式的文件和 GRASS 矢量图层
- 用于图片地理编码的几何校正插件
- 用于导入、导出 GPX 格式，将其他 GPS 格式转换成 GPX 格式，或者直接从一台 GPS 设备（在 Linux 上通过 USB 添加到 GPS 设备列表中）下载/上传数据的 GPS 工具
- 支持可视化与编辑 OpenStreetMap 数据
- 能够通过数据库管理器插件从文件创建空间数据库中的数据表
- 改进的空间数据表处理能力
- 管理矢量属性表的工具
- 将屏幕截图保存为带空间坐标参考图片的选项
- 具有增强功能的 dxf 导出工具，可以导出样式，且可以通过插件来执行类似 cad 的功能

4.4 分析数据

您可以对空间数据库和其他 OGR 支持的格式执行空间数据分析。QGIS 当前提供矢量分析、栅格分析、采样、地理处理、几何与数据库管理工具。你也可以使用集成的 GRASS 工具，其中包括 400 多个模块的完整 GRASS 功能（参见:ref:sec_grass 章节）。或者，您可以使用 Processing 插件，它提供了一个强大的地理空间分析框架，可以从 QGIS 调用本地和第三方算法，如 GDAL、SAGA、GRASS、R 等（参见:ref:sec_processing_intro 章节）。所有分析功能都在后台运行，使您可以在处理完成之前继续进行工作。

图形建模器允许您在直观的图形环境中将功能组合/链接成一个完整的工作流。

4.5 在互联网上发布地图

QGIS 可以用作 WMS、WMTS、WMS-C 或者 WFS 与 WFS-T 客户端（参见 *Working with OGC / ISO protocols* 章节），同时 QGIS Server（参见 QGIS-Server-manual 章节）允许您使用 Web 服务器在互联网上通过 WMS、WCS 与 WFS 协议将数据发布。

4.6 通过插件扩展 QGIS 功能

QGIS 可以通过可扩展的插件系统和可用于创建插件的库来满足你的特殊需求。你可以使用 C++ 或者 Python 来创建新应用。

4.6.1 核心插件

核心插件包括：

1. 数据库管理器（从/到数据库中转换、编辑与查看图层及属性表；执行 SQL 查询）
2. 几何检查器（检查几何错误）
3. 几何校正工具 GDAL（利用 GDAL 给栅格数据添加投影信息）
4. GPS 工具（加载或导入 GPS 数据）
5. GRASS 7（集成 GRASS GIS）
6. MetaSearch 目录客户端（与支持 Web 的 OGC 目录服务（CSW）标准的元数据目录服务进行交互）
7. 离线编辑（允许离线编辑与数据库同步）
8. 处理（QGIS 空间数据处理框架）
9. 拓扑检查器（查找矢量图层中的拓扑错误）

4.6.2 外部 Python 插件

QGIS 提供了越来越多的由社区提供的外部 Python 插件。这些插件位于官方插件库中，可以使用 Python 插件安装程序轻松安装。参见章节:ref:managing_plugins。

4.7 Python 终端

可以利用集成的 Python 终端执行脚本，该控制台可以通过：插件 ► *Python* 终端打开。该终端作为非模态实用程序窗口打开。可以通过：`data:qgis.utils iface` 变量与 QGIS 环境进行交互操作，改变了是：`class:QgisInterface <qgis.gui.QgisInterface>` 类的一个实例。该接口提供了对 QGIS 应用程序中地图画布、菜单、工具栏和其他部分的访问。您可以创建一个脚本，然后将其拖放到 QGIS 窗口中，它将自动执行。

为了进一步了解有关 Python 终端、QGIS 插件与应用编程相关的信息，请跳转到：`ref:console` 与 `ref:PyQGIS-Developer-Cookbook`。

4.8 已知问题

4.8.1 打开文件数量限制

如果您正在打开一个大型 QGIS 项目，并且您确定所有图层都是有效的，但是有些图层被标记为坏的，那么您可能会面临这个问题。Linux（以及其他类似的操作系统）有按进程打开文件的限制。每个进程的资源是受限的，且可以被继承。`ulimit` 命令是一个内置的 shell 命令，它仅为当前 shell 进程更改限制，新的限制将被每一个子进程继承。

输入以下命令，你可以看到所有当前的 `ulimit` 信息：

```
$ ulimit -aS
```

在终端输入以下命令，你可以看到当前每个进程所允许打开的文件数量：

```
$ ulimit -Sn
```

要更改 ** 现有会话 ** 的限制，您可以使用以下命令：

```
$ ulimit -Sn #number_of_allowed_open_files
$ ulimit -Sn
$ qgis
```

永久修改

在大多数 Linux 系统中，资源限制是在登录时通过“`pam_limits`”模块设定的，该模块依据的设置保存在文件：`file:/etc/security/limits.conf` 或者：`file:/etc/security/limits.d/*conf` 中。如果你拥有 root 权限（也可以通过 `sudo`）则可以编辑这些文件，但是你需要注销并重新登录使得变更生效。

更多信息：




<https://www.cyberciti.biz/faq/linux-increase-the-maximum-number-of-open-files/> <https://linuxaria.com/article/open-files-in-linux>

本章简要介绍了 QGIS 的安装、QGIS 示例数据的下载和运行第一个简单的栅格数据和矢量数据可视化的案例。

5.1 安装 QGIS

QGIS 项目根据您的平台提供了不同的安装 QGIS 的方法。

5.1.1 二进制文件安装

标准安装适用于  Windows 操作系统和  macOS 操作系统。二进制包（rpm 和 deb 格式）或者软件库方式适用于多种版本的 GUN/Linux  操作系统。

有关您的操作系统的更多信息和说明请查看 <https://download.qgis.org>。

5.1.2 从源代码安装

如果你需要从源代码编译 QGIS，请参照安装说明。该说明随源代码一起发布在:file:INSTALL'文件中。你也可以在网站上找到他们 <https://github.com/qgis/QGIS/blob/master/INSTALL.md>。

如果你想要编译一个特定版本，而不是开发中的版本，你需要将上面提到的连接中的 'master' 替换为发行分支（通常为 "release-X_Y"形式）。（安装说明可能会有不同）

5.1.3 安装到外媒介

可以将 QGIS（包括所有的插件和设置）安装到一个闪存驱动器上。这是通过定义一个 `--profiles-path` 选项来实现的，该选项覆盖默认的用户配置文件（*user profile*）路径，并强制设置（**QSettings**）也使用这个目录。更多信息请查阅系统设置（系统设置）章节。

5.1.4 下载示例数据

This user guide contains examples based on the QGIS sample dataset (also called the Alaska dataset). Download the sample data from <https://github.com/qgis/QGIS-Sample-Data/archive/master.zip> and unzip the archive on any convenient location on your system.




阿拉斯加数据集包含了用于示例和用户手册中截图的所有 GIS 数据。它还包含了一个较小的 GRASS 数据库。QGIS 样本数据集的投影坐标系是单位为英尺的阿拉斯加阿尔伯斯等面积坐标系（Alaska Albers Equal Area）。该投影坐标系的 EPSG 号为 2964。

```
PROJCS["Albers Equal Area",
GEOGCS["NAD27",
DATUM["North_American_Datum_1927",
SPHEROID["Clarke 1866",6378206.4,294.978698213898,
AUTHORITY["EPSG","7008"]],
TOWGS84[-3,142,183,0,0,0,0],
AUTHORITY["EPSG","6267"]],
PRIMEM["Greenwich",0,
AUTHORITY["EPSG","8901"]],
UNIT["degree",0.0174532925199433,
AUTHORITY["EPSG","9108"]],
AUTHORITY["EPSG","4267"]],
PROJECTION["Albers_Conic_Equal_Area"],
PARAMETER["standard_parallel_1",55],
PARAMETER["standard_parallel_2",65],
PARAMETER["latitude_of_center",50],
PARAMETER["longitude_of_center",-154],
PARAMETER["false_easting",0],
PARAMETER["false_northing",0],
UNIT["us_survey_feet",0.3048006096012192]]
```




如果您打算使用 QGIS 作为 GRASS 的图形化前端，您可以在 GRASS GIS 官方网站上找到一些示例位置（例如，斯皮尔菲什市 Spearfish 或南达科他 South Dakota），网址为 <https://grass.osgeo.org/download/sample-data/>

5.2 启动和关闭 QGIS

QGIS 可以像您电脑上的其他应用一样打开。这意味着你可以通过以下方式打开：

- 使用  应用菜单， 开始菜单，或者  DOCK
- 在应用文件夹或者桌面快捷方式上双击
- 双击一个已经存在的 QGIS 项目文件（扩展名为 `.qgz` 或者 `.qgs`）。注意，这也将打开项目。
- 在命令提示符里输入 `qgis`（假设 QGIS 已经被添加到默认路径 `PATH` 里面，或者当前目录为安装目录）

关闭 QGIS，使用：

-   菜单选项 项目 ► 退出 QGIS，或者用快捷键 `Ctrl+Q`
-  QGIS ► 退出 QGIS，或者使用快捷键 `Cmd+Q`
- 或者使用应用程序主界面右上角的红十字。

5.3 案例：加载栅格图层和矢量图层

现在你已经安装好了 QGIS (*QGIS installed*)，也有了一个示例数据集 (*sample dataset*)，接下来我们将演示第一个样本范例。在这个例子中，我们将一个栅格图层和一个矢量图层进行可视化。我们将使用：

- 土地覆盖 (landcover) 栅格图层 (qgis_sample_data/raster/landcover.img)
- 和湖泊 (lakes) 矢量图层 (qgis_sample_data/gml/lakes.gml)

其中，qgis_sample_data 代表数据集解压目录。

1. 启动 QGIS，参见启动和关闭 QGIS (*启动和关闭 QGIS*)。

2. 在 QGIS 中加载文件：

1. 单击 |dataSourceManager| :sup: 打开数据源管理器 ‘图标。数据源管理器应该在浏览器模式下打开。
2. 浏览到文件夹 qgis_sample_data/raster/
3. 选择 ERDAS IMG 文件 landcover.img，然后双击。土地覆盖图层在后台被添加，同时数据源管理器窗口依旧保持打开状态。

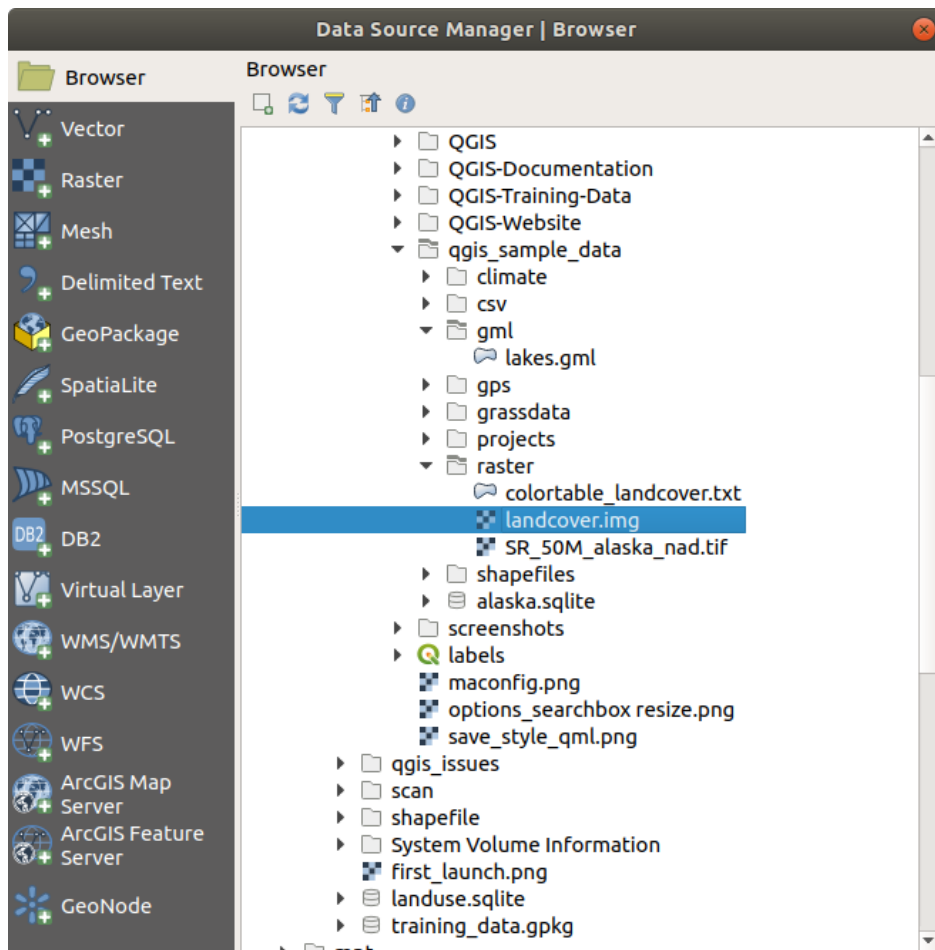


图 5.1: 在 QGIS 中添加数据到新项目

4. 要加载湖泊数据，请浏览到 qgis_sample_data/gml/ 文件夹，然后双击 :file:lakes.gml 文件打开。
5. 打开 坐标参考系选择器 ‘对话框。在 :guilabel: 过滤器菜单中输入 “2964”，过滤下面的坐标参考系列表。

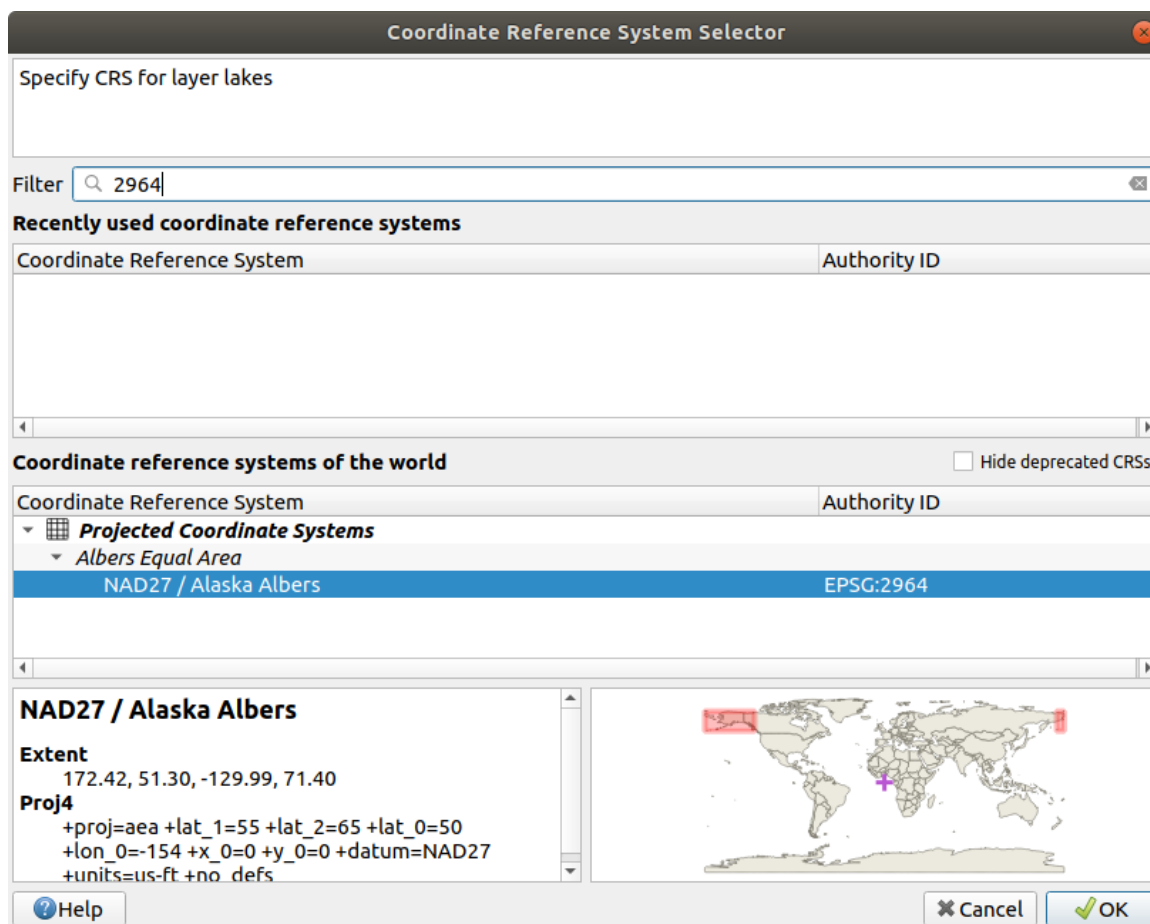



图 5.2: 选择数据的坐标参考系

6. Select the *NAD27 / Alaska Albers* entry
7. 单击 确定
8. 关闭数据源管理窗口

现在您的项目中有了一些随机颜色渲染的两个可用图层。让我们在湖泊图层做一些定制吧。

1. 在:guilabel:Navigation 工具栏上选择  Zoom In 工具
2. 缩放到一些湖泊的区域
3. 在图例中双击 lakes 图层，打开 属性 对话框
4. 改变湖泊颜色
 1. 单击 |symbology| 符号选项卡
 2. 选择蓝色作为填充颜色

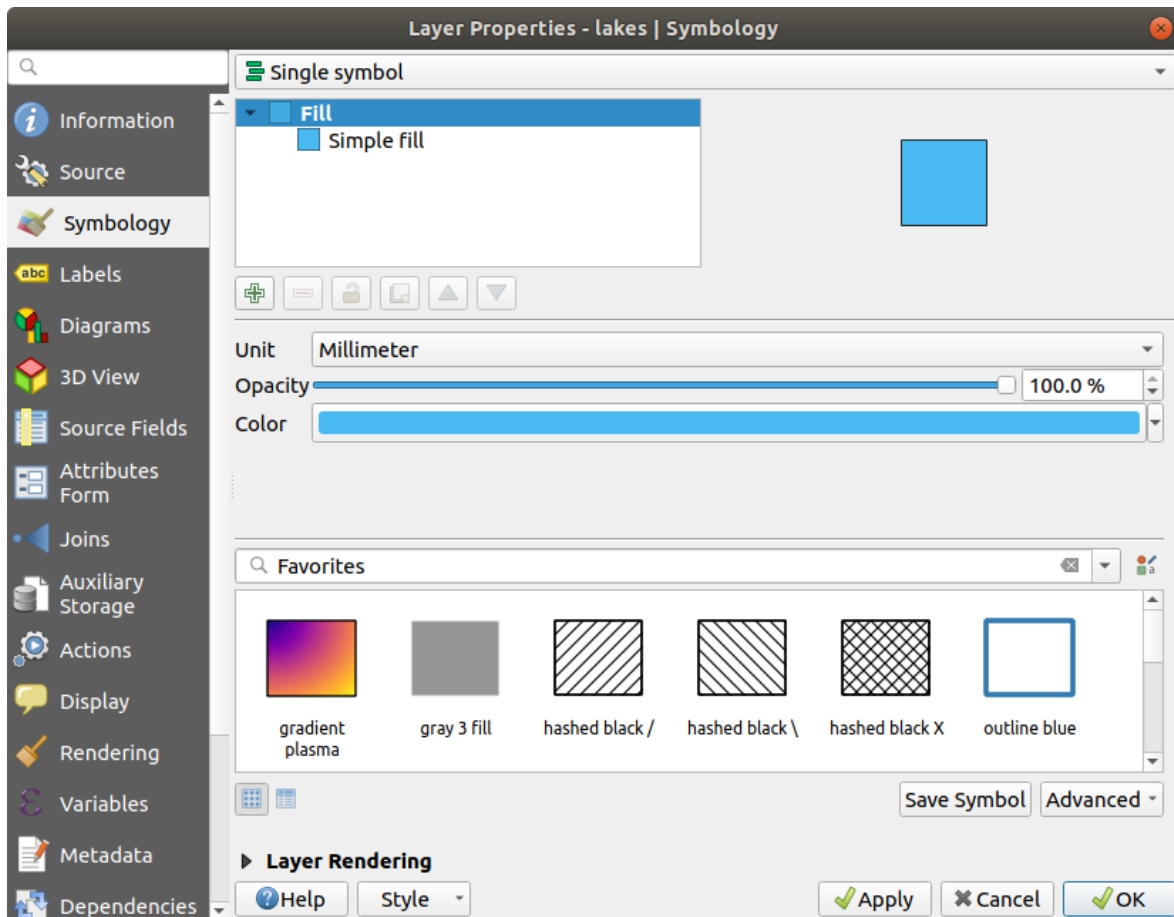


图 5.3: 选择湖泊颜色

3. 按下 确定。现在地图画布中湖泊显示为蓝色了。
5. 显示湖泊名称:
 1. 重新打开 lakes layer 属性对话框
 2. 单击 **abc** :guilabel:‘标注’选项卡
 3. 在下拉菜单中选择:guilabel:‘单一标注’来启用标注。
 4. 从:guilabel:‘Label with’列表中选择“NAMES”字段

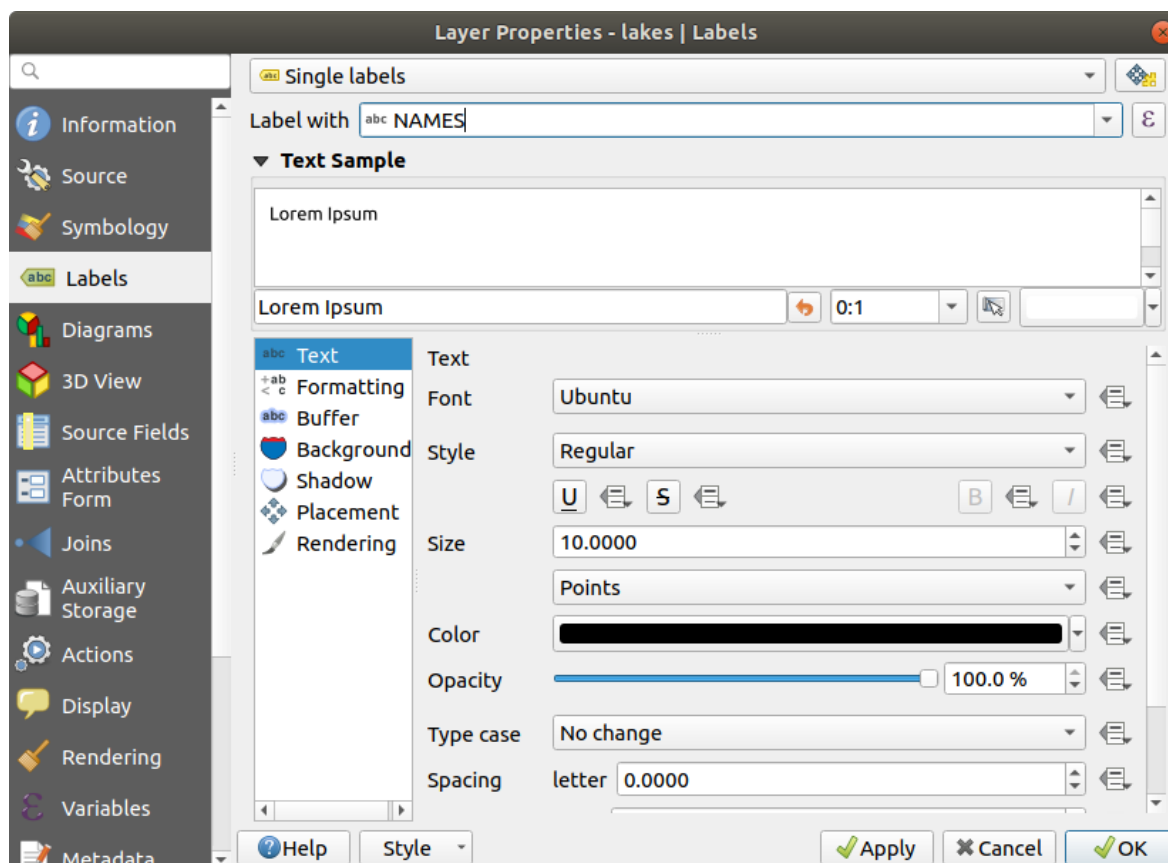




图 5.4: 显示湖泊名称

5. 按下 应用。现在名称加载在边界上。
6. 你可以在标注周围添加白色轮廓已增加标注的辨识度：
 1. 单击左边列表中的:guiabel:‘Buffer’选项卡
 2. 勾选 绘制文本轮廓
 3. 轮廓大小选择“3”
 4. 单击:guiabel: 应用
 5. 检查结果是否良好，如果需要可以调整轮廓大小值。
 6. 最终单击 确定来关闭:guiabel:‘图层属性’对话框并应用变更。

现在，让我们添加一些装饰以塑造地图并将其导出 QGIS：



1. 选择 视图 ► 装饰 ► 比例尺菜单
2. 在打开的对话框中，选中 启用比例尺
3. 根据你的需要自定义对话框中的选项
4. 按下:guiabel: 应用
5. 同样，可以从装饰菜单中添加更多项（指北针、版权…）到自定义属性的地图画布中。
6. 单击 工程 ► 导入/导出 ►  导出为图片...
7. 在打开的对话框中按下 保存
8. 选择文件位置与格式，并再次按下:guiabel:‘保存’确认。

9. 按下 工程 ►  保存... ，将你的修改保存到:file:'.qgz'工程文件中。


就这样！您可以看到在 QGIS 中可视化栅格和矢量图层、配置它们并以可以在其他软件中使用的图像格式生成地图是多么容易。让我们继续了解有关可用功能、特性和设置的更多信息，以及如何使用它们。

注解： 跟随 Training manual，通过一步步练习来继续学习 QGIS。

6.1 QGIS 工程简介

QGIS 会话的状态称为工程，QGIS 一次只能处理一个工程。设置可以是特定于工程的，也可以是新工程的应用程序范围的默认设置（请参见选项）。QGIS 可以通过使用菜单选项 工程 ►  保存或者 工程 ►  另存为... 将工作区的状态保存到 QGIS 工程文件中。

注解：如果工程已被修改过，该 * 符号将出现在标题栏中，默认情况下，QGIS 将询问您是否要保存更改。此行为由 设置 ► 选项 ► 常规下的 需要时提示保存工程和数据源更改情况设置而进行控制的。

可以从浏览器面板或通过 工程 ►  打开...、工程 ► 由模板新建或 工程 ► 最近打开的文件 ► 将现有工程加载到 QGIS 里。

启动时，会显示 工程模板和 最近工程列表，包括屏幕截图、名称和文件路径（最多 10 个工程）。最近工程列表便于访问最近使用的工程，双击条目打开工程或工程模板。还可以添加图层并自动创建新工程。然后列表将消失，取而代之的是地图画布。

如果您想清理会话并重新开始，请转到 工程 ►  新建。如果在现有工程打开或上次保存后进行了更改，将提示您保存该工程。

打开新工程时，标题栏将显示 未命名的工程，直到您将其保存。

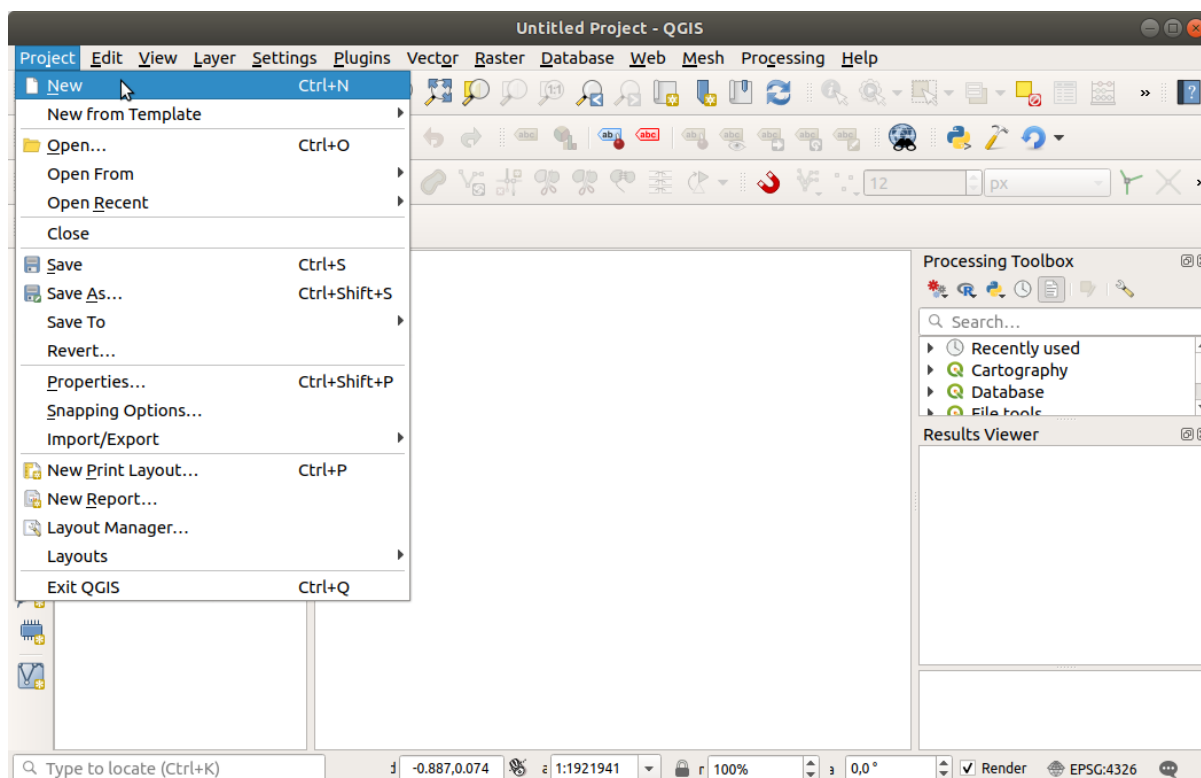



图 6.1: 在 QGIS 中启动新工程

保存在工程文件中的信息包括：

- 添加图层
- 可以查询哪些图层
- 图层属性，包括符号化和样式
- 地图视图的投影
- 上次查看范围
- 打印布局
- 使用设置打印布局元素
- 打印布局地图集设置
- 数字化设置
- 表关系
- 工程宏命令
- 工程默认样式
- 插件设置
- 工程属性中 OWS 设置选项卡的 QGIS 服务器设置
- 数据库管理器中存储的查询

工程文件以 XML 格式保存 (参见 [QGS/QGZ - QGIS 工程文件格式](#))。这意味着，如果您知道自己在做什么，就可以在 QGIS 之外编辑文件。工程文件格式已更新多次，来自较旧 QGIS 版本的工程文件可能不再正常工作。

注解：默认情况下，QGIS 将警告您版本差异。此行为在 设置 ► 选项的 通用选项卡中进行控制 ( 当打开旧版 QGIS 格式的工程文件时提出警告)。

每当您在 QGIS 中保存 .qgs 工程文件时，都会在与工程文件相同的目录中创建备份文件，扩展名为 .qgs~。

QGIS 工程的扩展名是 .qgs，但当从 QGIS 保存时，默认情况下使用带有 .qgz 扩展名的压缩格式保存。.qgs 与用于辅助数据相关的 SQLite 数据库 (.qgd) 一起嵌入到 .qgz 文件中 (zip 归档文件)，您可以通过解压 .qgz 文件来获取上述文件。

注解： *Auxiliary Storage Properties* 机制使压缩工程特别有用，因为它嵌入了辅助数据。

还可以使用以下工程菜单项将工程保存 / 加载到 PostgreSQL 数据库或从中加载工程：


- 工程 ► 打开从
- 工程 ► 保存到

这两个菜单项都有一个子菜单，其中列出了另外的工程存储实现方式 (PostgreSQL 和 GeoPackage)。单击将打开一个对话框，选择 GeoPackage (连接和工程两个选项) 或 PostgreSQL (连接、架构和工程三个选项)。


存储在 Geopackage 或 PostgreSQL 中的工程也可以通过 QGIS 浏览器面板加载，方法是双击或将它们拖到地图画布。

6.2 处理损坏的文件路径

在打开工程时，由于服务器/数据库不可用、重命名或移动了文件等原因，QGIS 可能无法访问某些数据源。随即 QGIS 打开 处理不可用图层对话框，检测并列出了未找到的图层，您可以：


- 双击 数据源字段，校正每个图层的路径，然后单击 应用更改；
- 选择某一行，单击 浏览按钮找到相关图层所在数据源的正确位置，然后单击 应用更改；
- 单击 自动查找按钮浏览文件夹，并尝试自动修复所有或选中的断开路径。请注意，浏览可能需要一些时间。
- 忽略该信息，单击 保留不可用图层按钮打开带有损坏路径的工程。然后，图层将显示在 图层面板，在 图层面板相关图层名称旁边显示  无效图层! 图标，单击该图标打开修复数据源对话框进行修复，在修复路径之前不会显示任何数据。或在图层上下文菜单中 修复数据源...。

使用 修复数据源... 工具，一旦图层路径被修复，QGIS 会扫描所有其他损坏的路径，并尝试自动修复那些具有相同损坏文件路径的路径。

- 从工程  移除不可用图层。

6.3 生成输出

有几种方法可以从 QGIS 会话中生成输出。我们已经在 *QGIS 工程简介* 讨论了保存为工程文件，生成输出文件的其他方法包括：

- 创建图像: 工程 ► 导入/导出 ►  导出地图为图像... 将地图画布渲染输出为自定义比例、分辨率、大小等的图像格式 (PNG, JPG, TIFF...), 可以对图像进行地理配准, 更多详细信息请参见 *导出地图视图*。
- 导出为 PDF 文件: 工程 ► 导入/导出 ► 导出地图为 PDF 文件... 以自定义比例、分辨率和一些高级设置 (简化、地理配准等) 将地图画布渲染输出为 PDF。更多详细信息请参见 *导出地图视图*。

- 导出为 DXF 文件: 工程 ► 导入/导出 ► 导出工程为 *DXF* 文件... 打开一个对话框, 可以在其中定义要导出到 DXF 的'符号模式','符号比例'和矢量图层。通过'符号模式', 可以高保真地导出原始 QGIS 符号系统中的符号 (请参见 *Creating new DXF files*)。
- 设计地图: 工程 ►  新建打印布局... 打开一个对话框, 可以在其中布局和打印当前地图画布 (请参见 *Laying out the maps* 一节)。

QGIS 图形用户界面

QGIS 图形用户界面 (GUI) 如下图所示 (黄色圆圈中的数字 1 到 5 表示 QGIS 图形用户界面的重要元素, 将在下面讨论)。

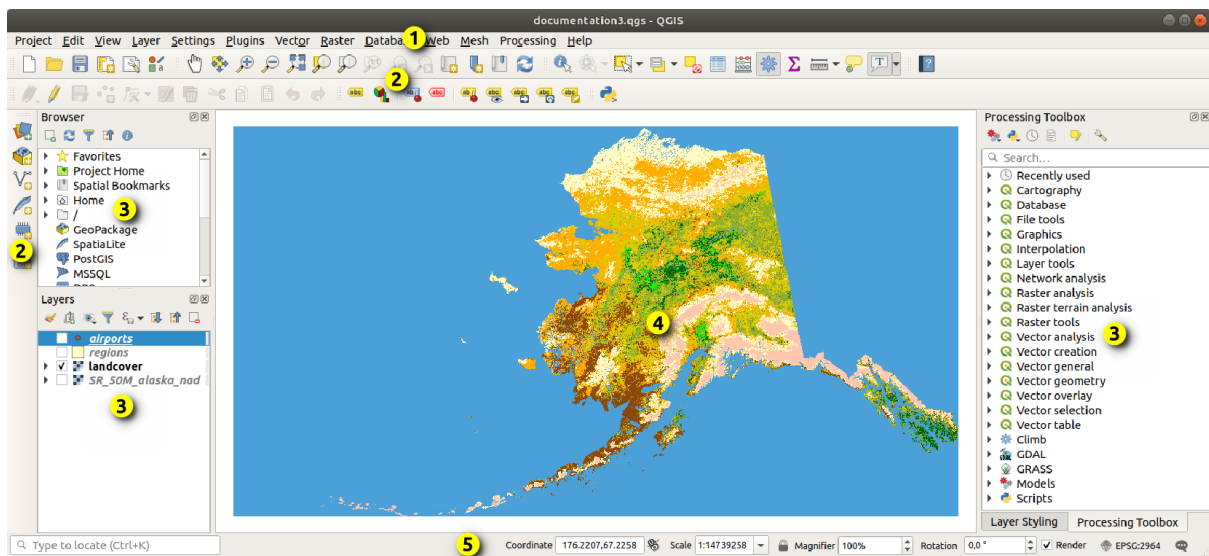


图 7.1: 带阿拉斯加样本数据的 QGIS 图形用户界面

注解: 根据操作系统和窗口管理器的不同, 窗口装饰 (标题栏等) 可能会有所不同。

QGIS 主界面 (图 7.1) 由五个组件类型构成:

1. 菜单栏
2. 工具栏
3. 面板
4. 地图视图
5. 状态栏

向下滚动，并查看这些内容的详细说明。

7.1 菜单栏

菜单栏提供使用标准分层菜单访问 QGIS 功能的功能。菜单及其选项、相关图标和键盘快捷键如下所述。可以重新配置键盘快捷键 ([设置](#) ► [键盘快捷键](#))。











大多数菜单选项都有相应的工具，反之亦然。但是，菜单的组织方式与工具栏不完全相同。下表显示了工具栏中菜单选项的位置。插件可以为菜单添加新的选项。有关工具和工具栏的详细信息，请参阅[工具栏](#)。

注解： QGIS 是一个跨平台的应用程序。工具通常在所有平台上都可以使用，但根据操作系统的不同，它们可能被放置在不同的菜单中。下面的列表显示了最常见的位置，包括已知的变量。

7.1.1 工程

工程菜单提供了[工程文件](#)的访问和退出等选项，提供如下工具：

- 新建工程文件或使用其他工程文件作为模板 (请参阅[工程文件选项](#)了解模板配置)
- 打开... 来自文件、GeoPackage 或 PostgreSQL 数据库的工程
- 关闭工程或将其恢复到上次保存的状态
- 保存工程为 .qgs 或 .qgz 文件格式，要么作为文件或保存在 GeoPackage，或 PostgreSQL 数据库
- 将地图画布导出为不同格式，或使用[打印布局](#)进行更复杂的输出
- 为几何图形编辑设置工程属性和捕捉选项。

菜单选项	快捷键	工具栏	参考
 新建	Ctrl+N	工程	QGIS 工程简介
由模板新建 ▶			QGIS 工程简介
 打开...	Ctrl+O	工程	QGIS 工程简介
打开 ▶			
▶ GeoPackage...			QGIS 工程简介
▶ PostgreSQL...			QGIS 工程简介
最近打开的文件 ▶	Alt+J+R		QGIS 工程简介
关闭			QGIS 工程简介
 保存	Ctrl+S	工程	QGIS 工程简介
 另存为...	Ctrl+Shift+S	工程	QGIS 工程简介
保存到 ▶			
▶ 模板...			QGIS 工程简介
▶ GeoPackage...			QGIS 工程简介
▶ PostgreSQL...			QGIS 工程简介
恢复...			
工程属性...	Ctrl+Shift+P		工程属性
捕捉选项...			Setting the Snapping Tolerance and Search Radius
导入/导出 ▶			
▶  导出地图为图像...			导出地图视图
▶  导出地图为 PDF 文件...			导出地图视图
▶ 导出工程为 DXF 文件...			Creating new DXF files
▶ 从 DWG/DXF 文件导入图层...			Importing a DXF or DWG file
 新建打印布局...	Ctrl+P	工程	Laying out the maps
 新建报告...			Creating a Report
 布局管理器...		工程	Laying out the maps
布局 ▶			Laying out the maps
 退出 QGIS	Ctrl+Q		

在 X macOS 系统中，退出 QGIS 命令对应于 QGIS ▶ 退出 QGIS (Cmd+Q)。

7.1.2 编辑

编辑菜单提供了编辑图层属性或几何图形所需的大部分本地工具 (有关详细信息，请参阅 [Editing](#))。

菜单选项	快捷键	工具栏	参考
 撤销	Ctrl+Z	数字化	Undo and Redo
 重做	Ctrl+Shift+Z	数字化	Undo and Redo
 剪切要素	Ctrl+X	数字化	Cutting, Copying and Pasting Features
 复制要素	Ctrl+C	数字化	Cutting, Copying and Pasting Features
 粘贴要素	Ctrl+V	数字化	Cutting, Copying and Pasting Features

下页继续

表 7.1 - 续上页

菜单选项	快捷键	工具栏	参考
粘贴要素为 ▶			属性表操作
▶ 新建矢量图层...			属性表操作
▶ 临时图层...	Ctrl+Alt+V		属性表操作
选择 ▶			<i>Selecting features</i>
▶  选择要素		选择	<i>Selecting features</i>
▶  按多边形选择要素		选择	<i>Selecting features</i>
▶  自由手绘选择要素		选择	<i>Selecting features</i>
▶  按半径选择要素		选择	<i>Selecting features</i>
▶  按值选择要素...	F3	选择	<i>Selecting features</i>
▶  按表达式选择要素...	Ctrl+F3	选择	<i>Selecting features</i>
▶  在所有图层中取消对要素的选择	Ctrl+Alt+A	选择	<i>Selecting features</i>
▶  从当前活动图层中取消选择要素	Ctrl+Shift+A	选择	<i>Selecting features</i>
▶ 重新选择要素			<i>Selecting features</i>
▶  选择全部要素	Ctrl+A	选择	<i>Selecting features</i>
▶  反选要素		选择	<i>Selecting features</i>
 添加记录	Ctrl+.	数字化	
 添加点要素	Ctrl+.	数字化	<i>Adding Features</i>
 添加线要素	Ctrl+.	数字化	<i>Adding Features</i>
 添加多边形要素	Ctrl+.	数字化	<i>Adding Features</i>
 添加圆弧		形状数字化	<i>Add Circular string</i>
 按半径添加圆弧		形状数字化	<i>Add Circular string</i>
添加圆 ▶		形状数字化	<i>Draw Circles</i>
▶  通过 2 点添加圆		形状数字化	<i>Draw Circles</i>
▶  通过 3 点添加圆		形状数字化	<i>Draw Circles</i>
▶  通过 3 条切线添加圆		形状数字化	<i>Draw Circles</i>
▶  通过 2 条切线和一个点添加圆		形状数字化	<i>Draw Circles</i>
▶  通过中心点和另一点添加圆		形状数字化	<i>Draw Circles</i>
添加矩形 ▶		形状数字化	<i>Draw Rectangles</i>
▶  由范围添加矩形		形状数字化	<i>Draw Rectangles</i>
▶  通过中心点和另一点添加矩形		形状数字化	<i>Draw Rectangles</i>












下页继续

表 7.1 - 续上页







菜单选项	快捷键	工具栏	参考
▶  由 3 个点创建矩形 (距离来自第 2 和第 3 个点)		形状数字化	<i>Draw Rectangles</i>
▶  由 3 个点创建矩形 (距离来自投影后的第一和第二个点)		形状数字化	<i>Draw Rectangles</i>
添加正多边形 ▶		形状数字化	<i>Draw Regular Polygons</i>
▶  从中心和另一点添加正多边形		形状数字化	<i>Draw Regular Polygons</i>
▶  通过中心点和拐角添加正多边形		形状数字化	<i>Draw Regular Polygons</i>
▶  通过 2 点添加正多边形		形状数字化	<i>Draw Regular Polygons</i>
添加椭圆 ▶		形状数字化	<i>Draw Ellipses</i>
▶  通过中心点和另 2 点添加椭圆		形状数字化	<i>Draw Ellipses</i>
▶  通过中心点和另一点添加椭圆		形状数字化	<i>Draw Ellipses</i>
▶  从范围添加椭圆		形状数字化	<i>Draw Ellipses</i>
▶  从焦点添加椭圆		形状数字化	<i>Draw Ellipses</i>
添加注记 ▶			<i>Annotation Tools</i>
▶  文本注记		属性	<i>Annotation Tools</i>
▶  表单注记		属性	<i>Annotation Tools</i>
▶  HTML 注记		属性	<i>Annotation Tools</i>
▶  SVG 注记		属性	<i>Annotation Tools</i>
 移动要素		高级数字化	<i>Move Feature(s)</i>
 复制并移动要素		高级数字化	<i>Move Feature(s)</i>
 删除所选要素		数字化	<i>Deleting Selected Features</i>
 修改所选要素的属性		数字化	<i>Editing attribute values</i>
 旋转要素		高级数字化	<i>Rotate Feature(s)</i>
 简化要素		高级数字化	<i>Simplify Feature</i>
 添加环		高级数字化	<i>Add Ring</i>
 添加部件		高级数字化	<i>Add Part</i>

下页继续

表 7.1 - 续上页

菜单选项	快捷键	工具栏	参考
 填充环		高级数字化	<i>Fill Ring</i>
 删除环		高级数字化	<i>Delete Ring</i>
 删除部件		高级数字化	<i>Delete Part</i>
 重塑要素		高级数字化	<i>Reshape Features</i>
 偏移曲线		高级数字化	<i>Offset Curves</i>
 分割要素		高级数字化	<i>Split Features</i>
 分割部件		高级数字化	<i>Split parts</i>
 合并所选要素		高级数字化	<i>Merge selected features</i>
 合并所选要素的属性		高级数字化	<i>Merge attributes of selected features</i>
 顶点工具 (所有图层)		数字化	<i>Vertex tool</i>
 顶点工具 (当前图层)		数字化	<i>Vertex tool</i>
 旋转点符号		高级数字化	<i>Rotate Point Symbols</i>
 偏移点符号		高级数字化	<i>Offset Point Symbols</i>
 反转线		高级数字化	<i>Reverse Line</i>
 裁剪/扩展要素		高级数字化	<i>Trim/Extend Feature</i>

依赖于所选图层几何图形类型 (即点、折线或多边形) 的工具会相应被激活:

菜单选项	点	折线	多边形
移动要素			
复制并移动要素			

7.1.3 视图

地图在地图视图中渲染，您可以使用视图工具与这些视图进行交互 (有关更多信息，请参阅 *Working with the map canvas*)。例如，您可以：

- 在主地图画布旁边新建 2D 或 3D 地图视图
- 缩放或平移 到任意位置
- 查询显示要素的属性或几何图形
- 使用预览模式、注记或装饰改善地图视图
- 访问所有面板或工具栏

该菜单还允许您使用以下操作重新组织 QGIS 界面：

- 切换全屏模式: 占用整个屏幕，同时隐藏标题栏
- 切换面板可见性: 显示或隐藏已启用:ref:“面板”- 在数字化要素 (最大化画布可见性)，以及使用 QGIS 的主画布进行 (投影/记录) 展示时很有用
- 只切换地图: 隐藏面板、工具栏、菜单和状态栏，只显示地图画布。结合全屏选项，让您的屏幕只显示地图

菜单选项	快捷键	工具栏	参考
 新建地图视图	Ctrl+M		地图视图
 新建 3D 地图视图	Ctrl+Alt+M		3D 地图视图
 平移地图		地图浏览	<i>Zooming and Panning</i>
 平移地图到所选内容		地图浏览	
 放大	Ctrl+Alt++	地图浏览	<i>Zooming and Panning</i>
 缩小	Ctrl+Alt+-	地图浏览	<i>Zooming and Panning</i>
 识别要素	Ctrl+Shift+I	属性	<i>Identifying Features</i>
测量 ▶		属性	<i>Measuring</i>
▶  测量线	Ctrl+Shift+M	属性	<i>Measuring</i>
▶  测量面积	Ctrl+Shift+J	属性	<i>Measuring</i>
▶  测量角度		属性	<i>Measuring</i>
 统计汇总		属性	<i>Statistical Summary Panel</i>
 全图显示	Ctrl+Shift+F	地图浏览	<i>Zooming and Panning</i>
 缩放到选中的区域	Ctrl+J	地图浏览	<i>Zooming and Panning</i>
 缩放到图层范围		地图浏览	<i>Zooming and Panning</i>
 缩放到原始分辨率 (100%)		地图浏览	<i>Zooming and Panning</i>
 回到上一视图		地图浏览	<i>Zooming and Panning</i>
 下一视图		地图浏览	<i>Zooming and Panning</i>
装饰 ▶	Alt+V+D		<i>Decorations</i>
▶  网格...			<i>Grid</i>
▶  比例尺...			<i>Scale Bar</i>
▶  图像...			<i>Image Decoration</i>

下页继续


表 7.2 - 续上页

菜单选项	快捷键	工具栏	参考
▶  指北针...			<i>North Arrow</i>
▶  标题标注...			<i>Title Label</i>
▶  版权标注...			<i>Copyright Label</i>
▶  布局范围...			<i>Layout Extents</i>
预览模式 ▶			
▶ 正常			
▶ 模拟全色盲 (灰度)			
▶ 模拟单色 (单色)			
▶ 模拟色盲 (红色盲)			
▶ 模拟色盲 (<i>Deuteranope</i>)			
 显示地图提示		属性	<i>Display Properties</i>
 新建空间书签...	Ctrl+B	地图浏览	<i>Spatial Bookmarks</i>
 显示空间书签	Ctrl+Shift+B	地图浏览	<i>Spatial Bookmarks</i>
 显示空间书签管理器			<i>Spatial Bookmarks</i>
 刷新	F5	地图浏览	
 显示所有图层	Ctrl+Shift+U		<i>Layers Panel</i>
 隐藏所有图层	Ctrl+Shift+H		<i>Layers Panel</i>
 显示选中的图层			<i>Layers Panel</i>
 隐藏选中的图层			<i>Layers Panel</i>
 切换选中图层			<i>Layers Panel</i>
独立切换选中图层			<i>Layers Panel</i>
 隐藏未选中的图层			<i>Layers Panel</i>
面板 ▶			面板和工具栏
▶ 高级数字化			<i>The Advanced Digitizing panel</i>
▶ 浏览器			<i>The Browser Panel</i>
▶ 浏览器 (2)			<i>The Browser Panel</i>
▶ GPS 信息			<i>Live GPS tracking</i>
▶ GRASS 工具			<i>GRASS GIS Integration</i>
▶ 图层顺序			<i>Layer Order Panel</i>
▶ 图层样式			<i>Layer Styling Panel</i>
▶ 图层			<i>Layers Panel</i>
▶ 日志信息			<i>Log Messages Panel</i>
▶ 鹰眼图			<i>Overview Panel</i>
▶ 工具箱			<i>The Toolbox</i>
▶ 结果查看器			<i>The Toolbox</i>
▶ 捕捉和数字化选项			<i>Setting the Snapping Tolerance and Search Radius</i>
▶ 空间书签管理器			<i>Spatial Bookmarks</i>
▶ 统计			<i>Statistical Summary Panel</i>
▶ 瓦片比例			图块集
▶ 撤销/重做			<i>Undo/Redo Panel</i>
工具栏 ▶			面板和工具栏

下页继续

表 7.2 - 续上页

菜单选项	快捷键	工具栏	参考
▶ 高级数字化工具栏			<i>Advanced digitizing</i>
▶ 属性工具栏			
▶ 数据源管理器工具栏			管理数据源
▶ 数据库工具栏			
▶ 数字化工具栏			<i>Digitizing an existing layer</i>
▶ 帮助工具栏			
▶ 标注工具栏			<i>The Label Toolbar</i>
▶ 图层管理工具栏			管理数据源
▶ 地图浏览工具栏			
▶ 插件工具栏			插件
▶ 工程工具栏			
▶ 栅格工具栏			
▶ 选择工具栏			<i>Selecting features</i>
▶ 形状数字化工具栏			<i>Shape digitizing</i>
▶ 捕捉工具栏			<i>Setting the Snapping Tolerance and Search Radius</i>
▶ 矢量工具栏			
▶ Web 工具栏			
▶ GRASS			<i>GRASS GIS Integration</i>
切换全屏模式	F11		
切换面板可见性	Ctrl+Tab		
只切换地图	Ctrl+Shift+Tab		

在  Linux KDE 系统中, 面板 ▶, 工具栏 ▶ 和 切换全屏模式在 设置菜单中。

7.1.4 图层

图层菜单提供了大量工具用于创建新数据源, 添加它们到工程中或保存修改。使用相同的数据源, 您还可以:

- 创建副本用于生成副本的图层, 可以在其中修改名称、样式(符号、标注等)、连接等, 副本使用与原始副本相同的数据源。
- 复制和 粘贴图层或组从一个工程到另一个工程作为一个新实例, 其属性可以独立修改。至于 * 副本 *, 图层仍然基于相同的数据源。
- 或 嵌入图层和组... 来自另一个工程, 作为只读副本您无法进行修改(请参阅 *Nesting Projects*)

The 图层菜单还包含用于配置、复制或粘贴图层属性的工具(样式、比例、CRS...)。

菜单选项	快捷键	工具栏	参考
 数据源管理器	Ctrl+L	数据源管理器	打开数据
创建图层 ▶			<i>Creating new vector layers</i>
▶  新建 GeoPackage 图层...	Ctrl+Shift+N	数据源管理器	<i>Creating a new GeoPackage layer</i>
▶  新建 Shapefile 图层...		数据源管理器	<i>Creating a new Shapefile layer</i>
▶  新建 SpatiaLite 图层...		数据源管理器	<i>Creating a new SpatiaLite layer</i>

下页继续

表 7.3 - 续上页

菜单选项	快捷键	工具栏	参考
 新建临时图层...		数据源管理器	<i>Creating a new Temporary Scratch Layer</i>
 新建虚拟图层...		数据源管理器	<i>Creating virtual layers</i>
添加图层 ▶			<i>Opening Data</i>
 添加矢量图层.....	Ctrl+Shift+V	图层管理	<i>Loading a layer from a file</i>
 添加栅格图层...	Ctrl+Shift+R	图层管理	<i>Loading a layer from a file</i>
 添加网孔图层...		图层管理	<i>Loading a mesh layer</i>
 添加分隔文本图层...	Ctrl+Shift+T	图层管理	<i>Importing a delimited text file</i>
 添加 PostGIS 图层...	Ctrl+Shift+D	图层管理	<i>Database related tools</i>
 添加 SpatiaLite 图层...	Ctrl+Shift+L	图层管理	<i>SpatiaLite Layers</i>
 添加 MSSQL 空间图层...		图层管理	<i>Database related tools</i>
 添加 Oracle 空间图层...		图层管理	<i>Database related tools</i>
 添加 DB2 空间图层...	Ctrl+Shift+2	图层管理	<i>Database related tools</i>
 添加/编辑虚拟图层...		图层管理	<i>Creating virtual layers</i>
 添加 WMS/WMTS 图层...	Ctrl+Shift+W	图层管理	<i>Loading WMS/WMTS Layers</i>
 添加 XYZ 图层...			<i>Using XYZ Tile services</i>
 添加 ArcGIS 地图服务器图层...		图层管理	
 添加 WCS 图层...		图层管理	<i>WCS Client</i>
 添加 WFS 图层...		图层管理	<i>WFS and WFS-T Client</i>
 添加 ArcGIS 要素服务器图层...		图层管理	
 添加矢量瓦片图层...			
嵌入图层和组...			<i>Nesting Projects</i>
从图层定义文件添加...			<i>Layer definition file</i>
 复制样式			<i>Save and Share Layer Properties</i>
 粘贴样式			<i>Save and Share Layer Properties</i>
 复制图层			
 粘贴图层/组			
 打开属性表	F6	属性	<i>属性表操作</i>
 切换编辑模式		数字化	<i>Digitizing an existing layer</i>
 保存图层编辑		数字化	<i>Saving Edited Layers</i>
 当前编辑 ▶		数字化	<i>Saving Edited Layers</i>
▶ 保存所选图层		数字化	<i>Saving Edited Layers</i>
▶ 恢复所选图层		数字化	<i>Saving Edited Layers</i>
▶ 取消所选图层		数字化	<i>Saving Edited Layers</i>
▶ 保存所有图层		数字化	<i>Saving Edited Layers</i>

下页继续

表 7.3 - 续上页

菜单选项	快捷键	工具栏	参考
▶ 恢复所有图层		数字化	<i>Saving Edited Layers</i>
▶ 取消所有图层		数字化	<i>Saving Edited Layers</i>
另存为...			<i>Creating new layers from an existing layer</i>
另存为图层定义文件...			<i>Layer definition file</i>
 移除图层/组	Ctrl+D		
 创建图层副本			
设置图层可见比例			
设置图层 CRS	Ctrl+Shift+C		<i>Layer Coordinate Reference Systems</i>
将图层的 CRS 设置为工程的 CRS			<i>Project Coordinate Reference Systems</i>
图层属性...			<i>The Vector Properties Dialog, Raster Properties Dialog, Mesh Dataset Properties</i>
过滤...	Ctrl+F		<i>Query Builder</i>
 标注			<i>Labels Properties</i>
 在鹰眼图中显示			<i>Overview Panel</i>
 在鹰眼图中显示所有数据			<i>Overview Panel</i>
 在鹰眼图中隐藏所有数据			<i>Overview Panel</i>

7.1.5 设置

菜单选项	参考
用户配置 ▶	使用用户配置文件
▶ 默认	使用用户配置文件
▶ 打开当前配置文件夹	使用用户配置文件
▶ 新建配置...	使用用户配置文件
 样式管理器...	<i>The Style Manager</i>
 自定义投影...	<i>Custom Coordinate Reference System</i>
 键盘快捷键...	<i>Keyboard shortcuts</i>
 界面自定义...	<i>Customization</i>
 选项...	选项

在  Linux KDE 系统中，您在 设置菜单中会找到更多工具，例如：面板 ▶，工具栏 ▶ 和 切换全屏模式。




7.1.6 插件

菜单选项	快捷键	工具栏	参考
 管理并安装插件...			<i>The Plugins Dialog</i>
 Python 控制台	Ctrl+Alt+P	插件	<i>QGIS Python console</i>

首次启动 QGIS 时，并非所有核心插件都加载。

7.1.7 矢量

如果启用了所有核心插件，矢量菜单看起来应该如下。

菜单选项	快捷键	工具栏	参考
 检查几何图形...			<i>Geometry Checker Plugin</i>
 GPS 工具	Alt+O+G	矢量	<i>GPS Plugin</i>
 拓扑结构检查器		矢量	<i>Topology Checker Plugin</i>
地理处理工具 ▶	Alt+O+G		
▶ 缓冲区...			<i>Buffer</i>
▶ 裁剪...			<i>Clip</i>
▶ 凸包...			<i>Convex hull</i>
▶ 差异...			<i>Difference</i>
▶ 融合...			融合 (<i>dissolve</i>)
▶ 相交...			<i>Intersection</i>
▶ 交集取反...			<i>Symmetrical difference</i>
▶ 联合...			<i>Union</i>
▶ 清除选中的多边形...			<i>Eliminate selected polygons</i>
几何图形工具 ▶	Alt+O+E		
▶ 质心...			<i>Centroids</i>
▶ 聚合几何图形...			<i>Collect geometries</i>
▶ 提取顶点...			<i>Extract vertices</i>
▶ 多部件转单部件...			<i>Multipart to singleparts</i>
▶ 多边形转线...			<i>Polygons to lines</i>
▶ 简化...			<i>Simplify</i>
▶ 检查有效性...			<i>Check validity</i>
▶ 德洛内三角网...			<i>Delaunay triangulation</i>
▶ 按计数增密...			<i>Densify by count</i>
▶ 添加几何图形属性...			<i>Add geometry attributes</i>
▶ 线转多边形...			<i>Lines to polygons</i>
▶ 泰森多边形...			<i>Voronoi polygons</i>
分析工具 ▶	Alt+O+A		
▶ 线相交...			<i>Line intersections</i>
▶ 平均坐标...			<i>Mean coordinate(s)</i>
▶ 字段基本统计...			<i>Basic statistics for fields</i>
▶ 统计点在多边形中的数量...			<i>Count points in polygon</i>
▶ 距离矩阵...			<i>Distance matrix</i>
▶ 列出唯一值...			<i>List unique values</i>
▶ 最近邻分析...			<i>Nearest neighbour analysis</i>
▶ 计算线总长...			<i>Sum line lengths</i>
数据管理工具 ▶	Alt+O+D		
▶ 合并矢量图层...			<i>Merge vector layers</i>
▶ 重投影图层...			<i>Reproject layer</i>
▶ 创建空间索引...			<i>Create spatial index</i>
▶ 按位置连接属性...			<i>Join attributes by location</i>
▶ 分割矢量图层...			<i>Split vector layer</i>
研究工具 ▶	Alt+O+R		
▶ 按位置选择...			<i>Select by location</i>
▶ 提取图层范围...			<i>Extract layer extent</i>
▶ 范围内随机生成点...			<i>Random points in extent</i>
▶ 图层边界内生成随机点...			<i>Random points in layer bounds</i>
▶ 多边形内部的随机点...			<i>Random points inside polygons</i>
▶ 随机选择...			<i>Random selection</i>

下页继续

表 7.4 - 续上页



菜单选项	快捷键	工具栏	参考
▶ 子集内随机选择...			<i>Random selection within subsets</i>
▶ 生成规则点...			<i>Regular points</i>

默认情况下，QGIS 会将数据处理算法添加到 矢量菜单中，并按子菜单分组。这为来自不同提供程序的许多常见基于矢量的 GIS 任务提供了快捷方式。如果不是所有这些子菜单都可用，请在数据处理插件中启用 插件 ▶ 管理并安装插件...

请注意，矢量菜单工具列表可以通过任何数据处理算法或一些外部插件 进行扩展。

7.1.8 栅格

如果启用了所有核心插件，栅格菜单看起来应该如下。

菜单选项	快捷键	工具栏	参考
 栅格计算器...			<i>Raster Calculator</i>
对齐栅格...			<i>Raster Alignment</i>
 配准工具	Alt+R + G	栅格	<i>Georeferencer</i>
分析 ▶			
▶ 坡向...			<i>Aspect</i>
▶ 填充无数据区域...			<i>Fill nodata</i>
▶ 网格化 (滑动平均法)...			<i>Grid (Moving average)</i>
▶ 网格化 (数据度规法)...			<i>Grid (Data metrics)</i>
▶ 网格化 (距离倒数乘方法)...			<i>Grid (Inverse distance to a power)</i>
▶ 网格化 (最近邻算法)...			<i>ref:gdalgridinversedistancenearestneighbor</i>
▶ 山体阴影...			<i>Hillshade</i>
▶ 邻近分析 (生成栅格距离图)...			<i>Proximity (raster distance)</i>
▶ 粗糙度...			<i>Roughness</i>
▶ 滤除碎斑...			<i>Sieve</i>
▶ 坡度...			<i>Slope</i>
▶ 地形位置指数 (TPI)...			<i>Topographic Position Index (TPI)</i>
▶ 地形耐用指数 (TRI)...			<i>Terrain Ruggedness Index (TRI)</i>
投影 ▶			
▶ 指定投影...			<i>Assign projection</i>
▶ 提取投影...			<i>Extract projection</i>
▶ 变形 (重投影)...			<i>Warp (reproject)</i>
杂项 ▶			
▶ 构建虚拟栅格...			<i>Build virtual raster</i>
▶ 栅格信息...			<i>Raster information</i>
▶ 合并...			<i>Merge</i>
▶ 构建鹰眼图 (金字塔)...			<i>Build overviews (pyramids)</i>
▶ 瓦片索引...			<i>Tile index</i>
提取 ▶			
▶ 按范围裁剪栅格...			<i>Clip raster by extent</i>
▶ 按掩膜图层裁剪栅格...			<i>Clip raster by mask layer</i>
▶ 等高线...			<i>Contour</i>
转换 ▶			
▶ PCT 转 RGB...			<i>PCT to RGB</i>
▶ 栅格矢量化 (栅格转矢量)...			<i>Polygonize (raster to vector)</i>
▶ 矢量栅格化 (矢量转栅格)...			<i>Rasterize (vector to raster)</i>
▶ RGB 转 PCT...			<i>RGB to PCT</i>
▶ 转换 (格式转换)...			<i>Translate (convert format)</i>

默认情况下，QGIS 会将数据处理算法添加到 栅格菜单中，并按子菜单分组。这为来自不同提供程序的许多常见基于栅格的 GIS 任务提供了快捷方式。如果不是所有这些子菜单都可用，请在数据处理插件中启用 插件 ► 管理并安装插件...

请注意，栅格菜单工具列表可以通过任何数据处理算法或一些外部插件 进行扩展。

7.1.9 数据库

如果启用了所有核心插件，以下就是 数据库菜单包含项。如果没有启用数据库插件，则不会有 数据库菜单。

菜单选项	快捷键	工具栏	参考
离线编辑...	Alt+D + O		<i>Offline Editing Plugin</i>
►  转换为离线工程...		数据库	<i>Offline Editing Plugin</i>
►  同步		数据库	<i>Offline Editing Plugin</i>
 数据库管理器...		数据库	<i>DB Manager Plugin</i>

首次启动 QGIS 时，并非所有核心插件都加载。

7.1.10 Web


如果启用了所有核心插件，以下就是 Web 菜单包含项。如果没有启用 Web 插件，则不会有 Web 菜单。

菜单选项	快捷键	工具栏	参考
元搜索 ►	Alt+W + M		<i>MetaSearch Catalog Client</i>
►  元搜索		Web	<i>MetaSearch Catalog Client</i>
► 帮助			<i>MetaSearch Catalog Client</i>

首次启动 QGIS 时，并非所有核心插件都加载。

7.1.11 网孔

网孔菜单提供了操作网孔图层 所需的工具。

菜单选项	快捷键	工具栏	参考
 网孔计算器...			

7.1.12 空间运算

菜单选项	快捷键	工具栏	参考
 工具箱	Ctrl+Alt+T		<i>The Toolbox</i>
 模型构建器...	Ctrl+Alt+G		<i>The graphical modeler</i>
 历史...	Ctrl+Alt+H		<i>The history manager</i>
 结果查看器	Ctrl+Alt+R		<i>Configuring external applications</i>
 原地编辑要素			<i>The Processing in-place layer modifier</i>

首次启动 QGIS 时，并非所有核心插件都加载。

7.1.13 帮助

菜单选项	快捷键	工具栏	参考
 帮助	F1	帮助	
API 文档			
插件 ►			
报告问题			
需要商业支持?			
 QGIS 主页	Ctrl+H		
 检查 QGIS 版本			
 关于			
 QGIS 维护人员			


7.1.14 QGIS

此菜单仅在 **X** macOS 下可用，并包含一些与操作系统相关的命令。

菜单选项	快捷键
首选项	
关于 QGIS	
隐藏 QGIS	
显示全部	
隐藏其他	
退出 QGIS	Cmd+Q

对于其他平台 首选项对应 设置 ► 选项, 关于 QGIS 对应 帮助 ► 关于和 退出 QGIS 对应 工程 ► 退出 QGIS。

7.2 面板和工具栏

从视图菜单(或  设置), 您可以打开和关闭 QGIS 控件(面板 ►)和工具栏(工具栏 ►)。要(取消)激活其中任何一项, 请右键单击菜单栏或工具栏, 选择所需项。面板和工具栏可以在 QGIS 界面中移动和放置在任何你喜欢的地方。该列表还可以通过激活核心或外部插件 进行扩展。

7.2.1 工具栏

工具栏提供了对菜单中大多数功能的访问, 以及与地图交互的其他工具。每个工具栏项都有可用的弹出帮助。将鼠标悬停在所选项上, 将显示工具用途的简短描述。

每个工具栏都可以根据需要移动。此外, 可以使用鼠标右键上下文菜单或将鼠标放在工具栏上关闭它们。

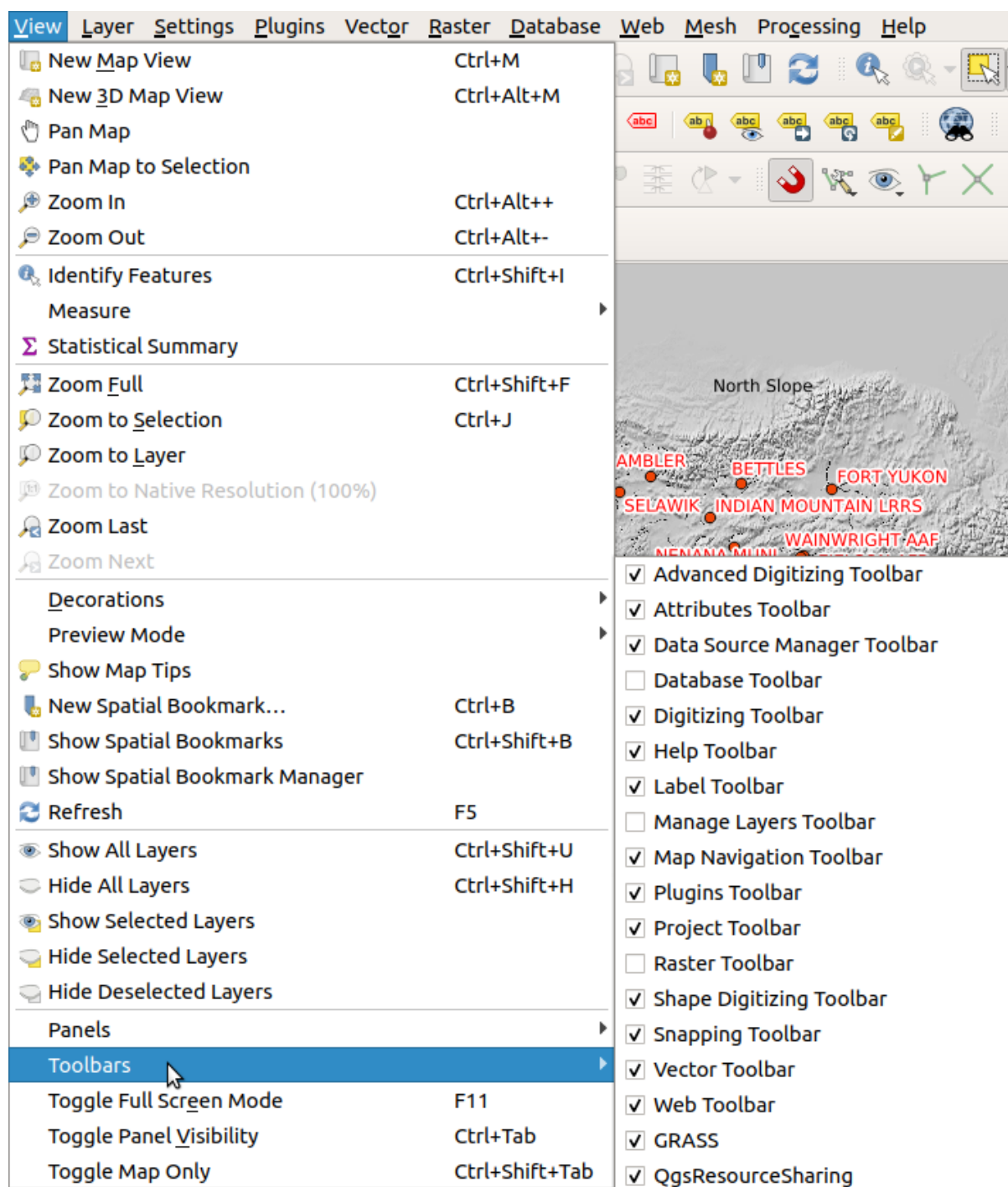



图 7.2: 工具栏菜单

小技巧: 恢复工具栏

如果不小心隐藏了工具栏，您可以使用 视图 ► 工具栏 ► (或  设置 ► 工具栏 ►) 将其恢复。如果由于某种原因，工具栏 (或任何其他控件) 完全从界面中消失，您可以在恢复初始 GUI 中找到恢复它的提示。

7.2.2 面板

QGIS 提供了许多面板。面板是特殊控件，可以与之交互的（选择选项、复选框、填充值…）执行更复杂的任务。

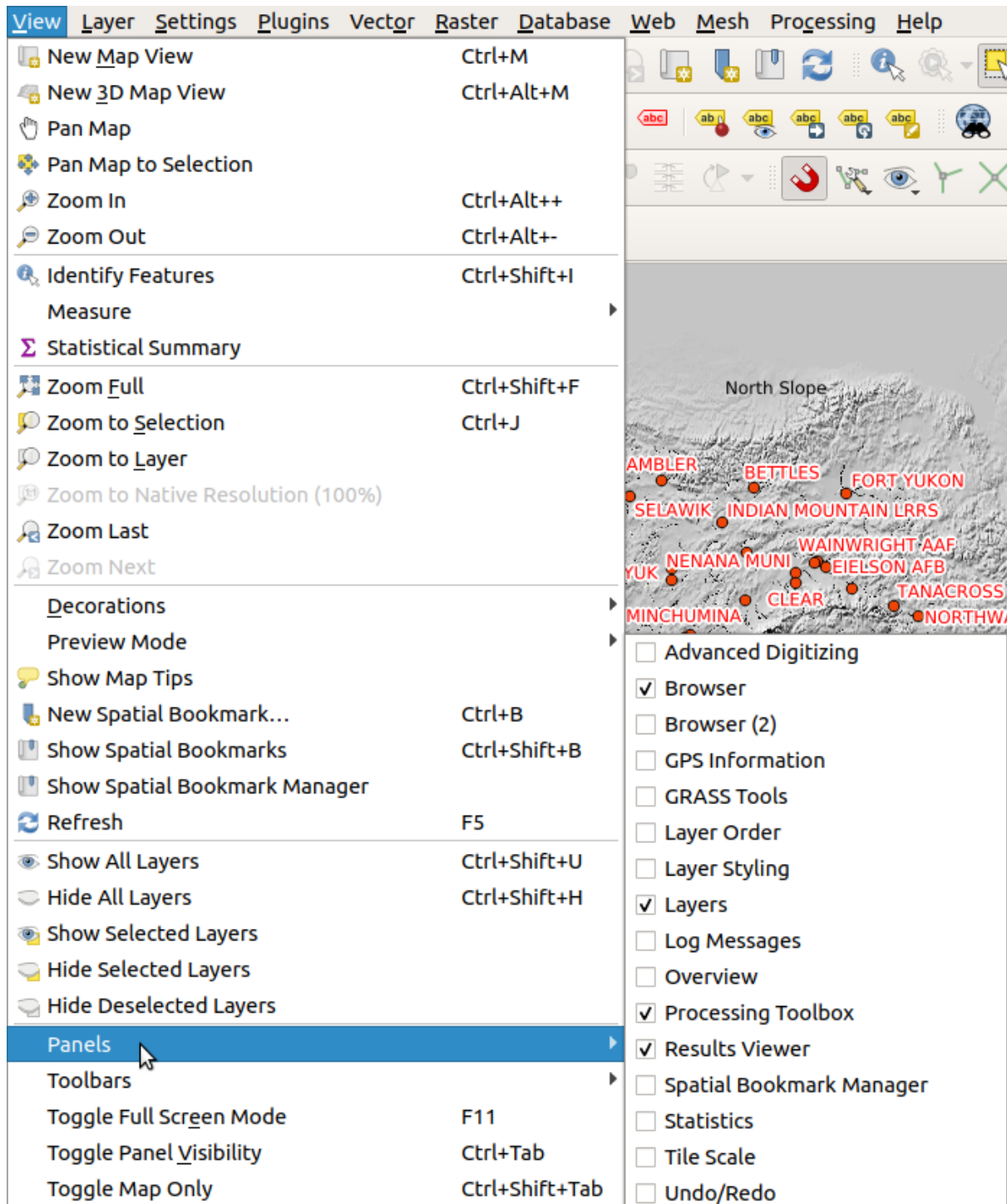


图 7.3: 面板菜单

以下是 QGIS 提供的默认面板列表：

- 高级数字化面板
- 浏览器面板

- GPS 信息面板
- 识别面板
- 图层顺序面板
- 图层样式面板
- 图层面板
- 日志信息面板
- 鹰眼图面板
- 工具箱
- the 结果查看器面板
- 空间书签管理器面板
- 统计面板
- 瓦片比例面板
- 撤销/重做面板









7.3 地图视图


7.3.1 探索地图视图

地图视图 (又称为 **地图画布**) 是 QGIS 的”业务端” --- 地图以 2D 的形式显示在该区域, 此窗口中显示的地图将反映您应用于已加载图层的渲染 (符号化、标注、可见性...), 它还取决于图层和工程的坐标参照系 (CRS)。

当您添加图层时 (例如, 请参阅 *Opening Data*), QGIS 自动查找其 CRS。如果默认情况下为工程设置了不同的 CRS (请参阅 *Project Coordinate Reference Systems*), 图层范围将”动态”转换为该 CRS, 如果从空 QGIS 工程开始, 地图视图将缩放到该范围。如果工程中已存在图层, 则不会执行地图画布调整, 因此只有位于当前地图画布范围内的要素可见。


单击地图视图, 您应该能够与之交互:

- 可以进行平移, 将显示移动到地图的另一个区域: 这可以使用  平移地图 工具、方向键来执行, 也可以按住 空格键、鼠标中键或鼠标滚轮的同时移动鼠标。
- 可以使用专门的  放大 和  缩小 工具进行放大和缩小操作。按住 Alt 键可从一个工具切换到另一个工具。缩放也可以通过向前滚动滚轮进行放大, 向后滚动滚轮进行缩小。缩放以鼠标光标位置为中心。
您可以在 设置 ► 选项 ► 地图工具菜单中自定义 缩放因子。
- 可以将其缩放到所有加载图层的整个范围 ( 全图显示)、图层范围 ( 缩放到图层范围) 或选中要素的范围 ( 缩放到选中的区域)
- 您可以使用  回到上一视图 和  下一视图 按钮或使用后退/前进鼠标按钮在画布视图历史中进行浏览。

右键单击地图, 您应该能够在地图的 CRS、WGS84 或自定义 CRS 中  复制坐标单击点的坐标, 然后将复制的信息粘贴到表达式、脚本、文本编辑器或电子表格中...

默认情况下, QGIS 打开单个地图视图 (称为”主地图”), 该视图与 图层面板紧密绑定; 主地图会 自动反映您在 图层面板区域中所做的更改。但也可以打开其他地图视图, 其内容可能与 图层面板的当前状态不同。它们可以是 2D 或 3D 类型, 显示不同的比例或范围, 或者由于 **地图主题** 而显示不同的加载图层集。

7.3.2 设置其他地图视图

要添加新的地图视图，请转到 视图 ►  新建地图视图。QGIS 中添加了新的浮动控件，模拟主地图视图的渲染。可以根据需要添加任意数量地图视图，它们可以保持浮动，并排放置或堆叠在一起。

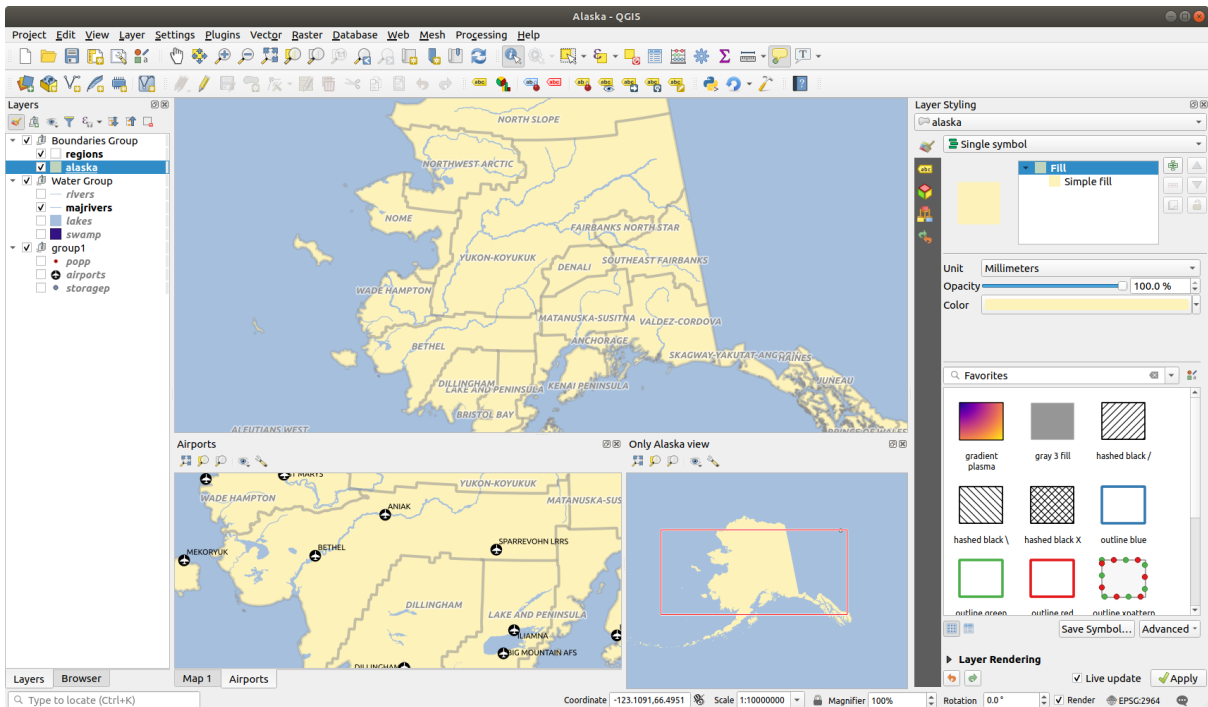














图 7.4: 具有不同设置的多个地图视图

在新建的地图画布顶部，有一个具有以下功能的工具栏：



-  全图显示、 缩放到选中的区域 和  缩放到图层范围 在视图中浏览
-  设置视图主题 以选择要在地图视图中显示的:ref: 地图主题 <map_themes>。如果设置为 (无)，视图将跟随 图层面板的变化而变化。
-  视图设置 用来配置地图视图：
 -  视图中心与主地图同步: 在不更改比例的情况下同步地图视图的中心，这允许您主画布中心有一个鹰眼图样式或放大的地图。
 -  将视图与所选内容同步: 与”缩放到选中的区域”作用相同
 - 比例
 - 旋转角度
 - 放大
 -  同步比例与主地图比例一致。然后可以应用 比例因子，使您可以拥有一个例如始终是主画布比例 2 倍的视图。
 -  显示标注
 -  显示指针位置
 -  显示主要画布范围
 -  显示标注: 允许隐藏标注，无论它们在显示图层的属性中如何设置

- 更改地图 CRS...
- 重命名视图...

7.3.3 导出地图视图

使用打印布局或报告的高级功能，可以将您制作的地图布局导出为各种格式，也可以不使用布局直接导出当前渲染。这个地图视图的快速“截图”有一些方便的功能。

要使用当前渲染导出地图画布，请执行以下操作：

1. 转到 工程 ► 导入/导出
2. 根据您的输出格式，选择
 -  导出地图为图像...
 - 或  导出地图为 PDF 文件...

这两个工具为您提供了一组通用选项。在打开的对话框中：

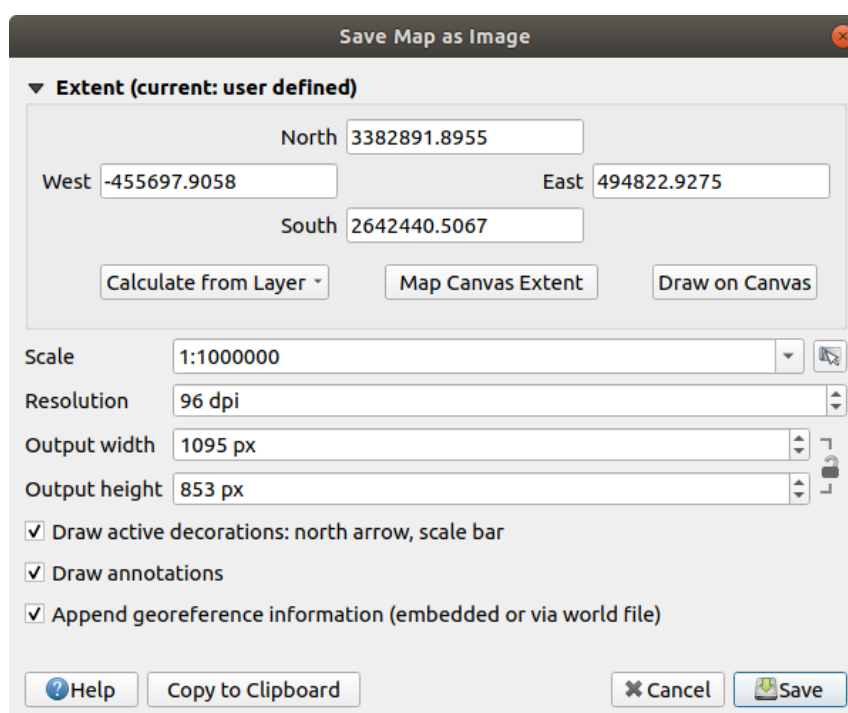


图 7.5: 地图另存为图像对话框

1. 选择要导出的 范围：可以是当前视图范围（默认）、图层范围或在地图画布上绘制的自定义范围。将显示所选区域的坐标，并可手动编辑。
2. 输入地图 比例或从预定义比例 选择：更改比例将调整要导出范围的大小（从中心开始）。
3. 设置输出的 分辨率
4. 以像素为单位控制图像的 输出宽度和 输出高度：默认是基于当前分辨率和范围，可以自定义并将调整地图范围的大小（从中心开始）。大小比值可以锁定，这在画布上绘制范围时可能特别方便。
5. 绘制活动装饰：使用中的装饰（比例尺、标题、网格、指北针等）与地图一起导出
6. 绘制注记导出全部注记

7. 附加配准信息 (嵌入或者借助于坐标信息文件): 根据输出格式, 一个同名的坐标信息文件 (扩展名为 PNGW 表示 PNG 图像, 扩展名为 JPGW 表示 JPG, ...) 与您的图像保存在同一文件夹中。PDF 格式将信息嵌入到 PDF 文件中。
8. 导出为 PDF 时, 地图另存为 *PDF...* 对话框提供了更多选项:

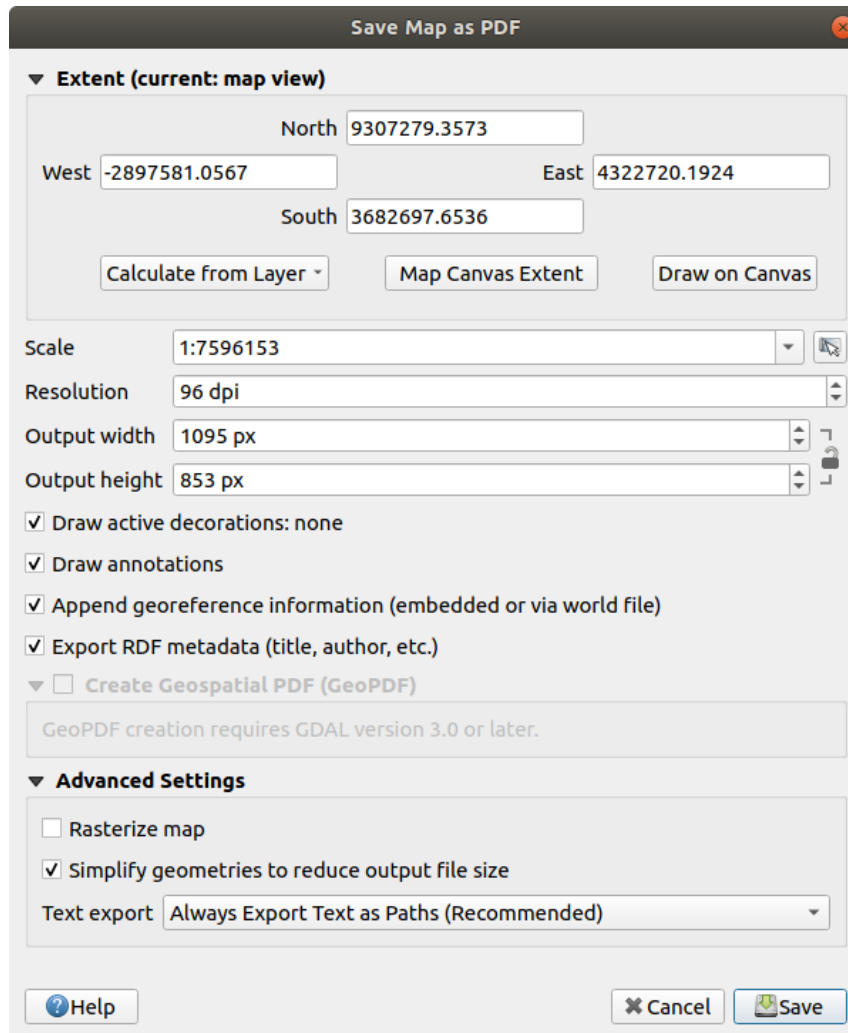


图 7.6: 地图另存为 PDF 的对话框

- 文档的 导出 *RDF* 元数据, 例如标题、作者、日期、描述...
- 创建地理空间 *PDF (GeoPDF)*: 生成地理配准信息的 *PDF* 文件 (需要 GDAL v3 或更高版本), 您可以:
 - 选择 GeoPDF 格式
 - 在 GeoPDF 文件中 包含矢量要素信息: 将在输出 GeoPDF 文件中包含地图中可见要素的所有几何图形和属性的信息。

注解: 从 QGIS 3.10 开始, GDAL 3 还可以使用 GeoPDF 文件作为数据源。有关 QGIS 中 GeoPDF 支持的更多信息, 请参阅 <https://north-road.com/2019/09/03/qgis-3-10-loves-geopdf/>。

- 地图栅格化
- 简化几何图形以减小输出文件的大小: 导出地图时, 几何图形将通过删除在导出分辨率下没有明显差异的顶点进行简化 (例如, 如果导出分辨率为 300 dpi, 相隔小于 1/600 英寸的

顶点将被删除)，这可以减小导出文件的大小和复杂性 (非常大的文件可能无法在其他应用程序中加载)。

- 设置 文本导出: 控制文本标注是作为正确的文本对象导出 (始终将文本导出为文本对象) 还是仅作为路径导出 (始终将文本导出为路径)。如果作为文本对象导出，可以在外部应用程序 (例如 Inkscape) 中作为普通文本进行编辑。但其副作用是渲染质量降低，并且当设置某些文本如描边效果时，渲染会出现问题。这就是为什么建议导出为路径的原因。

9. 单击 保存后可以选择文件位置、名称和格式。

导出到图像时，还可以将上述设置的预期结果 复制到剪贴板，并将地图粘贴到另一应用程序中，如 LibreOffice、GIMP...

7.4 3D 地图视图

通过 3D 地图视图提供 3D 可视化支持。可以使用 视图 ▶ 新建 3D 地图视图创建并打开 3D 地图视图，这时将弹出一个浮动 QGIS 面板，面板可以停靠。

首先，3D 地图视图的范围和视图与 2D 主地图画布相同。可以使用一组浏览工具将视图转换为 3D 视图。

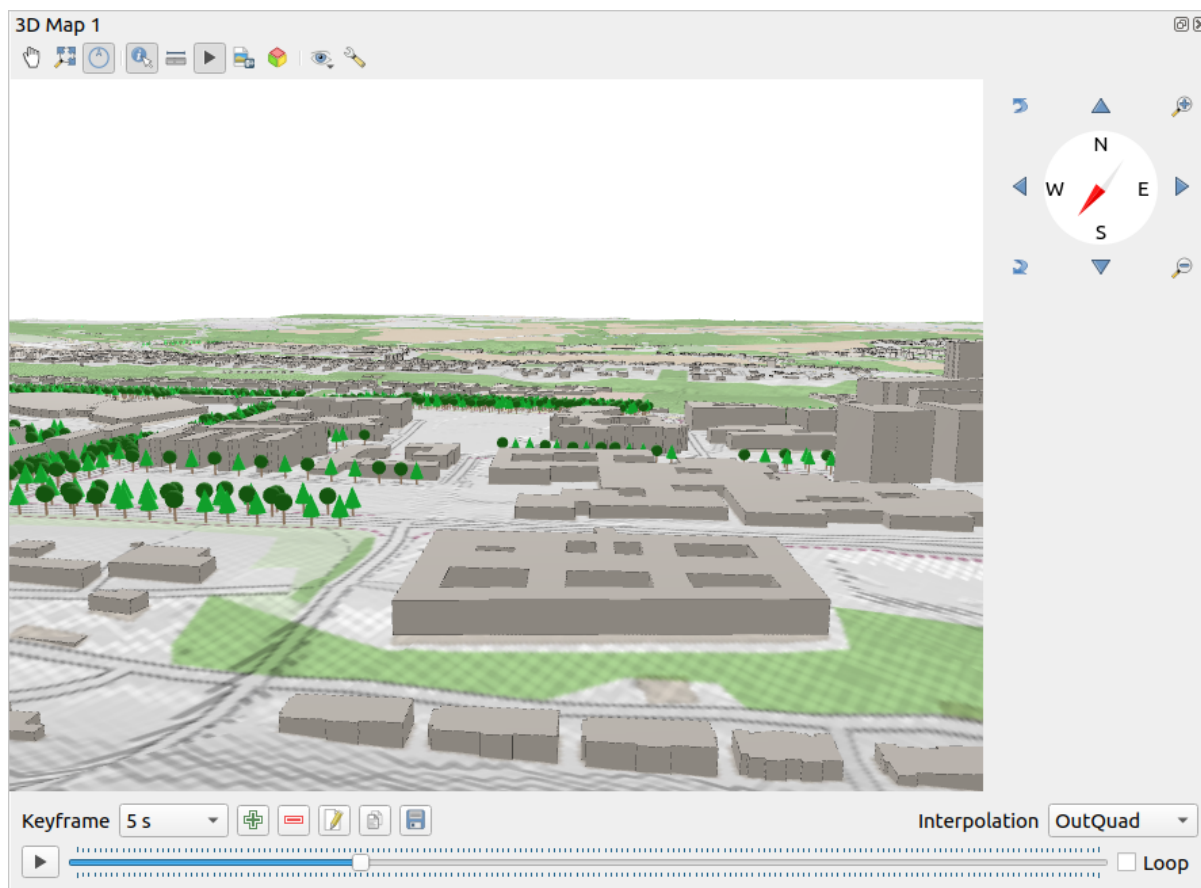









图 7.7: 3D 地图视图对话框




3D 地图视图面板顶部提供了以下工具：

- 相机视角控制: 移动视图，保持相机的角度和方向不变
- 全图显示: 将视图调整为整个图层的范围
- 切换导航工具: 显示/隐藏浏览控件 (旨在简化地图视图的控制)



-  识别: 返回有关地形的单击点或单击 3D 要素的信息 -- 更多详细信息请访问 [Identifying Features](#)
-  测量线: 测量点之间的水平距离
-  动画: 显示/隐藏动画播放 控件
-  另存为图像...: 将当前视图导出为图像文件格式
-  导出 3D 场景...: 将当前视图导出为 3D 场景 (.obj 文件), 允许在 **Blender** 等应用程序中进行后期处理。地形和矢量要素将导出为 3D 对象, 重写图层属性 或 地图视图配置 的导出设置, 包括:
 - 场景名称和目标 文件夹
 - 地形分辨率
 - 地形纹理分辨率
 - 模型比例
 - 平滑边
 - 导出法线
 - 导出纹理
-  设置视图主题: 允许您从预定义的地图主题 中选择要在地图视图中显示的图层集。
-  配置 地图视图设置


7.4.1 导航选项

要在 3D 中浏览地图视图, 请执行以下操作:

- 倾斜地形 (围绕穿过窗口中心的水平轴旋转地形)
 - 按下  向上倾斜 和  向下倾斜 工具
 - 按下 Shift 并使用向上/向下方向键
 - 按住鼠标中键, 向前/向后拖动鼠标
 - 按住 Shift 并在按住鼠标左键的情况下向前/向后拖动鼠标
- 旋转地形 (围绕穿过窗口中心的垂直轴旋转地形)
 - 将导航控件的罗盘转到观察方向
 - 按住 Shift 并使用左/右键
 - 按住鼠标中键, 左右拖动鼠标
 - 按住 Shift 并在按住鼠标左键的情况下左右拖动鼠标
- 更改相机位置 (和视图中心), 在水平平面移动相机
 - 按住鼠标左键拖动鼠标, 并启用  相机视角控制
 - 按导航控件的方向箭头
 - 使用上/下/左/右键分别向前、向后、向右和向左移动相机
- 更改相机高度: 按 Page Up/Page Down 键
- 更改相机方向 (相机保持在其位置, 但视图中心点移动)
 - 按住 Ctrl 并使用方向键上下左右转动相机
 - 按住 Ctrl 并在按住鼠标左键的情况下拖动鼠标




- 放大和缩小


- 按导航控件中相关的  放大 和  缩小 工具
- 滚动鼠标滚轮 (按住 Ctrl 键可获得更精细的缩放效果)
- 按住鼠标右键拖动鼠标进行放大 (向下拖动) 和缩小 (向上拖动)



要重置相机视图, 请单击 3D 画布面板顶部的  全图显示 按钮。


7.4.2 创建动画

动画基于一组关键帧 - 特定时间的相机位置。要创建动画, 请执行以下操作:

1. 切换  动画 工具, 显示动画播放控件
2. 单击  添加关键帧 按钮并以秒为单位输入 关键帧时间, 关键帧组合框现在显示时间设置。
3. 使用导航工具, 将相机移动到与当前关键帧时间相关联的位置。
4. 重复前面的步骤, 根据需要添加若干关键帧 (包括时间和位置)。
5. 单击  按钮预览动画。QGIS 将使用相机在设定时间的位置或旋转生成场景, 并在这些关键帧之间进行插值。可以使用各种动画 插值模式 (例如, linear、inQuad、outQuad、inCirc…… 更多详细信息, 请访问 <https://doc.qt.io/qt-5/qeasingcurve.html#EasingFunction-typedef>)。

也可以通过移动时间滑块预览动画。按住  重复 按钮将重复运行动画, 而单击  将停止运行动画。

可以使用 关键帧列表浏览相机不同的视图。每当时间处于活动状态时, 更改地图视图将自动更新相关位置。也可以使用  编辑关键帧 (仅限时间) 或  移除关键帧。

单击  导出动画帧 将生成一系列表示场景的图像。除了文件名 模板和 输出目录, 还可以设置 帧率数、输出宽度和 输出高度。

7.4.3 场景配置

3D 地图视图打开后，其中包含一些可自定义的默认设置。要执行此操作，请单击 3D 画布面板顶部的配置... 按钮，打开 3D 配置窗口。

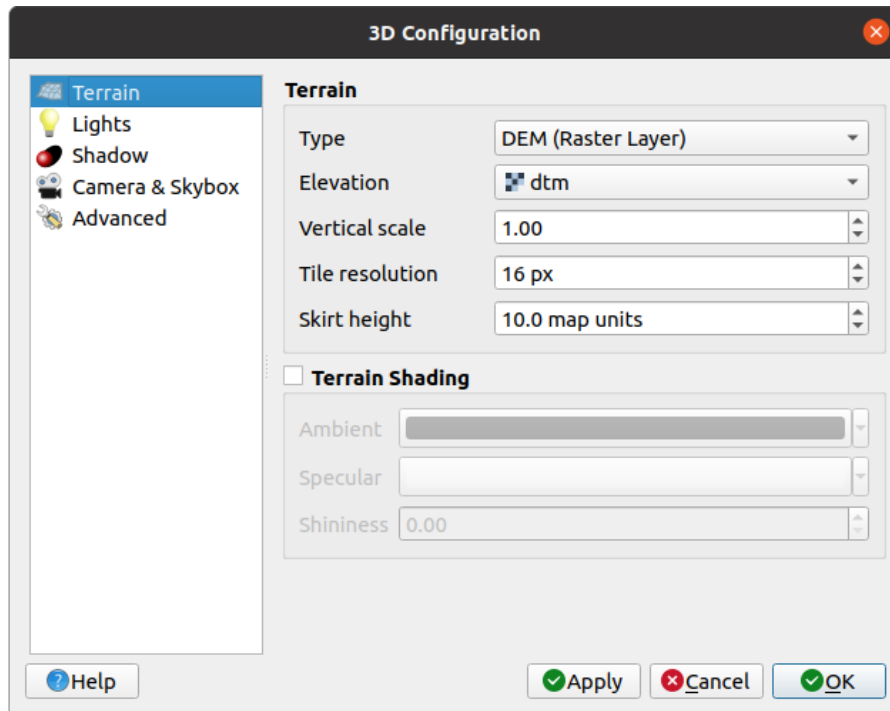


图 7.8: 3D 地图配置对话框


在 3D 配置窗口，有各种选项可微调 3D 场景：

地形

- 地形: 在深入研究细节之前，值得注意的是，3D 视图中的地形是由地形瓦片的层次结构表示，随着相机靠近地形，没有足够细节的现有瓦片将被具有更多细节的较小瓦片替换。每个瓦片都是源自高程栅格图层的网孔几何图形和 2D 地图图层的纹理。
 - 高程地形 类型可以是：
 - * a 平坦地形
 - * 已加载的 DEM (栅格图层)
 - * 在线服务，加载由 Mapzen 工具生成的高程瓦片 -- 更多详细信息，请访问 <https://registry.opendata.aws/terrain-tiles/>
 - * 已加载的网孔数据集
 - 高程: 用于生成地形的栅格或网孔图层。栅格图层必须包含表示高程的波段；对于网孔图层，将使用顶点的 Z 值。
 - 垂直比例: 垂直轴的比例因子，增加比例会扩大地貌的高度。
 - 瓦片分辨率: 每个瓦片要使用的地形栅格图层中的样本数。16px 的值意味着每个瓦片的几何图形将由 16x16 个高程样本组成。数字越大，生成的地形瓦片越详细，但渲染复杂度就越高。
 - 裙板高度: 有时可能会在地形的瓦片之间看到小裂缝。提高该值将在地形瓦片周围添加垂直墙 (“裙板 - skirts”)，以隐藏裂缝。

- 当网孔图层用作地形时，可以配置 三角形设置 (线框显示、平滑三角形) 和 渲染颜色设置 (统一或取决于地形层级)。更多详细信息，请参见网孔图层属性 部分。
- 地形晕渲: 允许您选择地形的渲染方式：
 - 禁用晕渲 - 地形颜色仅由地图纹理确定
 - 启用晕渲 - 使用 Phong 的晕渲模型确定地形颜色，同时考虑地图纹理、地形法线矢量、场景灯光以及地形材质的 周围环境和 镜面反射颜色以及 光滑度

灯光

在 灯光选项卡中，按  菜单添加

- 最多八个 点光源: 向各个方向发光，就像一个填充区域的光球。离光源越近的对象越亮，而远离光源的对象就越暗。点光源具有设定的位置 (X, Y 和 Z)、颜色、强度和 衰减等属性
- 最多四个 平行光: 模拟从远离对象的巨型闪光灯中获得的照明，始终居中且永不熄灭 (例如太阳)。它在一个方向上发射平行光线，但光线延伸到无限远。平行光可以在给定 方位角、海拔、颜色和强度的参数下进行旋转。

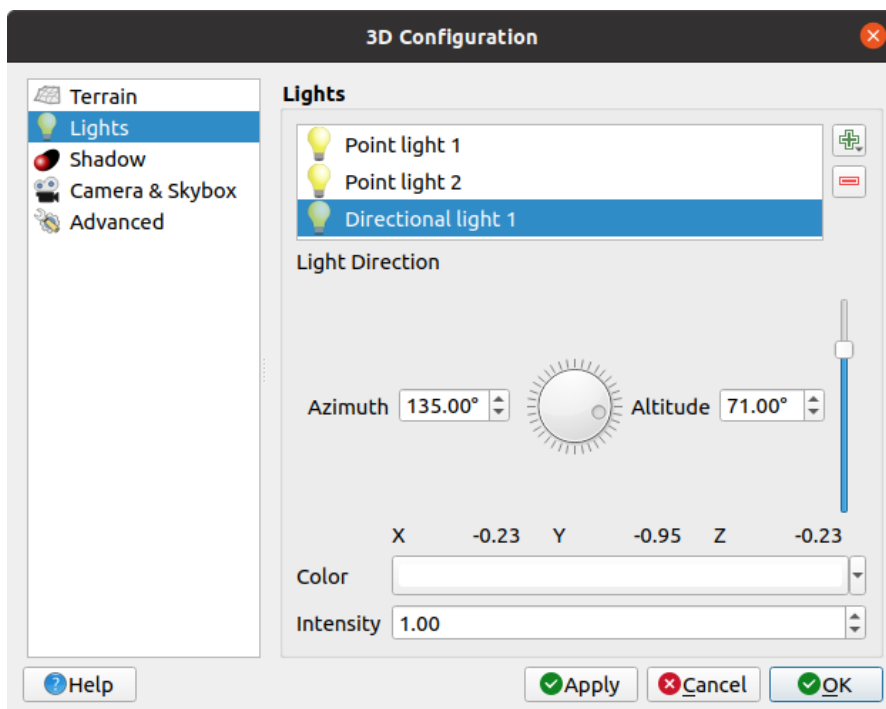


图 7.9: 3D 地图灯光配置对话框

阴影

勾选 显示阴影，在场景中显示阴影，前提是：

- 平行光
- 阴影渲染最大距离: 避免渲染太远对象的阴影，尤其是当相机沿地平线仰视时
- 阴影偏移: 避免由于地图大小不同而导致某些区域比其他区域更暗的自阴影效果，越低越好
- 阴影地图分辨率: 使阴影看起来更清晰。如果分辨率参数太高，可能会导致性能下降。

相机和天空盒

- 相机 视角: 允许创建全景场景, 默认值为 45°。
- 勾选 显示天空盒在场景中启用天空盒渲染, 天空盒类型可以是:
 - 全景纹理, 单个文件提供 360° 视野
 - 不同表面, 包含场景的盒子的六个面各有一个纹理文件
 纹理文件可以是磁盘上的文件、远程 URL 或嵌入到工程中的文件 ([更多详细信息](#))。

高级

- 地图瓦片分辨率: 用作地形瓦片纹理的 2D 地图图像的宽度和高度。256px 意味着每个瓦片被渲染成 256x256 像素的图像。数字越大, 生成的地形越详细, 但渲染复杂度越高。
- 最大屏幕误差: 确定将地形瓦片与更详细的地形瓦片交换的阈值 (反之亦然) - 即 3D 视图使用更高质量瓦片的时间。数字越小, 场景中的细节越多, 但渲染复杂度越高。
- 最大地面误差: 地形瓦片的分辨率, 在该分辨率下, 将瓦片拆分为更详细的瓦片将停止 (分割它们不会引入任何额外的细节)。该值限制了瓦片层次结构的深度: 值越低, 层次结构越深, 渲染复杂度越高。
- 缩放层级: 显示缩放层级的数量 (取决于地图瓦片分辨率和最大地面误差)。
- 显示标注: 打开或关闭地图标注
- 显示地图瓦片信息: 包括地形瓦片的边界和瓦片编号 (用于解决地形问题)
- 显示边界框: 显示地形瓦片的 3D 边界框 (用于解决地形问题)
- 显示相机视图中心
- 显示光源: 显示光源原点处的球体, 相对于场景内容更容易重新定位和放置光源

7.4.4 3D 矢量图层

通过选中矢量图层属性的 3D 视图部分中的 启用 3D 渲染器, 可以在 3D 地图视图中显示具有高程值的矢量图层。有许多选项可用于控制 3D 矢量图层的渲染。

7.5 状态栏

状态栏为您提供有关地图视图和已处理或可用操作的一般信息, 并提供管理地图视图的工具。

7.5.1 搜索栏 - 定位器

在状态栏的左侧, 搜索栏是一个快速搜索控件, 可帮助您查找并运行 QGIS 中的任何功能或选项:

1. 在文本控件中单击以激活搜索栏, 或按 Ctrl+K。
2. 键入与您要查找的工程相关联的文本 (名称、标签、关键字等)。默认情况下, 将启用搜索过滤的返回结果, 但您可以通过在文本前面加上 [搜索过滤](#) 前缀, 将搜索限制在指定范围内, 即键入 l cad 将仅返回名称包含 cad 的图层。

在访问搜索控件时, 还可以双击显示的菜单来选择过滤。


3. 根据项类型, 单击搜索结果执行相应的操作。

小技巧：将查找限制为当前图层的某个字段

默认情况下，使用“活动图层要素”过滤 (f) 的搜索将遍历图层的整个属性表。您可以使用 @ 前缀将搜索限制到特定字段。例如 f @name sal 或 @name sal 只返回其“name”属性包含’sal’的要素。书写时，文本自动补全处于活动状态，建议使用 Tab 键。

搜索是使用线程处理的，因此即使安装了慢速搜索过滤器，其结果也始终尽快可用。它们也会在遇到过滤时立即出现，这意味着，例如文件搜索过滤会在扫描文件树时逐个显示结果。这确保了用户界面始终响应，即使存在非常慢的搜索过滤（例如，使用在线服务的过滤）。

小技巧：快速访问搜索的配置



单击状态栏上搜索控件内的  图标，以显示可使用的过滤列表，以及打开 设置 ► 选项... 的搜索标签页的配置条目

7.5.2 汇报行为


在搜索栏旁边的区域中，将在需要时显示您已执行操作的摘要（例如选择图层中的要素、移除图层），或对您悬停工具的详细描述（并不适用于所有工具）。

如果需要执行较长时间的操作，例如在栅格图层中收集统计信息、执行处理算法或在地图视图中渲染多个图层，状态栏中将显示进度条。

7.5.3 控制地图画布


 坐标选项显示鼠标的当前位置，在地图视图中移动时跟随鼠标。可以在 工程 ► 工程属性... ► 常规选项卡中设置单位（和精度）。单击文本框右侧的小按钮，可以在坐标选项和  范围选项之间切换，该选项以地图单位显示地图视图当前左下角和右上角的坐标。

在显示坐标旁边，您会看到 比例显示，它显示地图视图的比例。有一个比例选择器，允许您在 预定义和自定义比例 之间进行选择。


在显示比例的右侧，按下  按钮锁定比例，以便使用放大镜进行放大或缩小操作。放大镜允许您在不改变地图比例的情况下放大地图，从而更容易准确调整标注和符号的位置。放大倍数以百分比表示，如果 放大镜的级别为 100%，当前地图不会被放大。此外，可以在 设置 ► 选项 ► 渲染 ► 渲染行为 中定义默认放大值，这对于高分辨率屏幕放大符号非常有用。

在放大镜工具右侧，可以为地图视图定义当前顺时针方向旋转角度，单位为度。

在状态栏右侧，有个小复选框，可以临时使用它来阻止图层被渲染到地图视图（请参见 [Rendering](#) 部分）。


在渲染功能右侧，可以找到显示当前工程 CRS 的  EPSG: 代码按钮，单击此按钮将打开 工程属性对话框，并允许您将另一个 CRS 应用于地图视图。



小技巧：计算地图画布的正确比例

启动 QGIS 时，默认的 CRS 为 WGS 84 (EPSG 4326)，单位为度。这意味着 QGIS 将按照指定的度数解释图层中的任何坐标。要获得正确的比例值，可以在 工程 ► 工程属性... 下的 常规选项卡中手动更改此设置（例如为米），或者您可以使用上面看到的  EPSG: 代码 图标。在后一种情况下，单位设置为工程投影指定的单位（例如 +units=us-ft）。

请注意，启动时的 CRS 选项可以在 设置 ► 选项 ► CRS 中设置。

7.5.4 实时信息

点击  按钮打开 日志信息面板，其中包含有关底层进程的信息 (QGIS 启动、插件加载、处理工具等)

根据 [插件管理器设置](#)，状态栏有时会在右侧显示图标信息，提示您有关新插件 () 或可升级插件 () 的相关信息。单击图标可以打开插件管理器对话框。

浏览器面板

QGIS 浏览器面板是浏览、搜索、检查、复制和加载 QGIS 资源的绝佳工具。只有 QGIS 知道如何处理的资源才会显示在浏览器中。

使用浏览器面板，您可以搜索、检查和添加数据，如 *The Browser Panel* 所述。此外，浏览器面板还支持拖放许多 QGIS 资源，例如工程文件、Python 脚本、处理脚本和处理模型。

Python 脚本、处理脚本和处理模型也可以在外部编辑器和图形建模器中打开进行编辑。

您可以将图层从图层面板拖放到浏览器面板，例如拖放到 GeoPackage 或 PostGIS 数据库中。

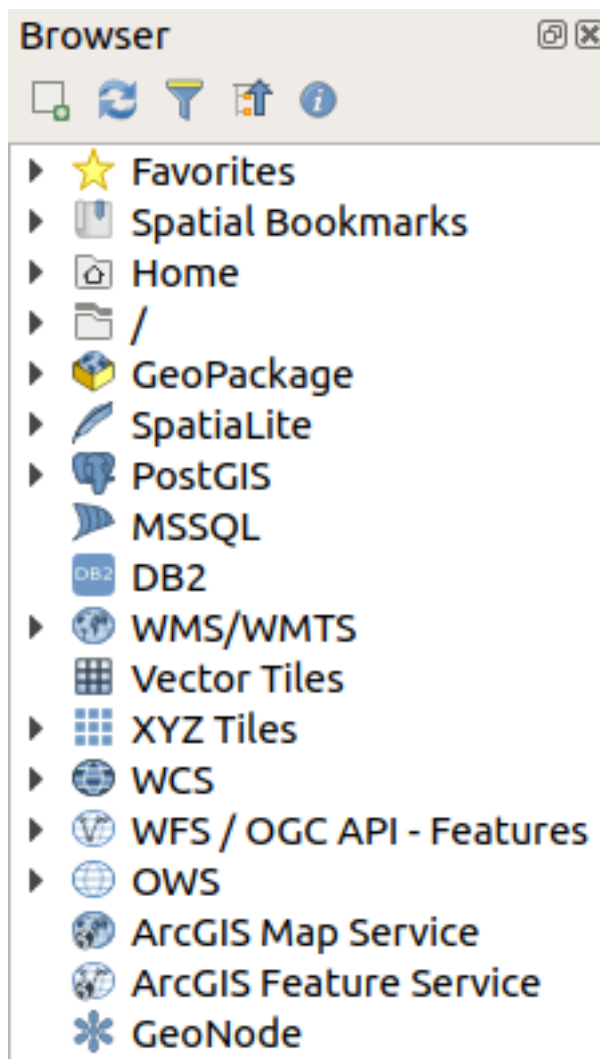










图 8.1: 浏览器面板

浏览器面板 (图 8.1) 构成一个可扩展的层次结构，其中包含一些固定的顶级条目，用于组织浏览器处理的资源。通过单击条目名称左侧的  展开节点条目，单击  可折叠分支， 全部折叠 按钮可折叠所有顶级条目。

在 设置  界面自定义中可以禁用资源。例如，如果您不想在浏览器中显示 Python 脚本，可以取消勾选 浏览器  *py* 条目；如果想在浏览器中排除您的主文件夹，可以取消勾选 浏览器  *special:Home* 条目。

过滤器 ( 过滤浏览器) 可用于根据条目名称 (层次结构中的目录条目和节点条目) 进行搜索。使用过滤器文本字段旁边的  选项 下拉菜单，可以

- 切换 区分大小写搜索
- 将 过滤模式语法设置为
 - 正常
 - 通配符
 - 正则表达式

属性控件显示有关条目或资源的实用信息，可以使用  启用/禁用属性控件 按钮启用或禁用。启用后，它将在浏览器面板的底部打开，如 图 8.2 所示。

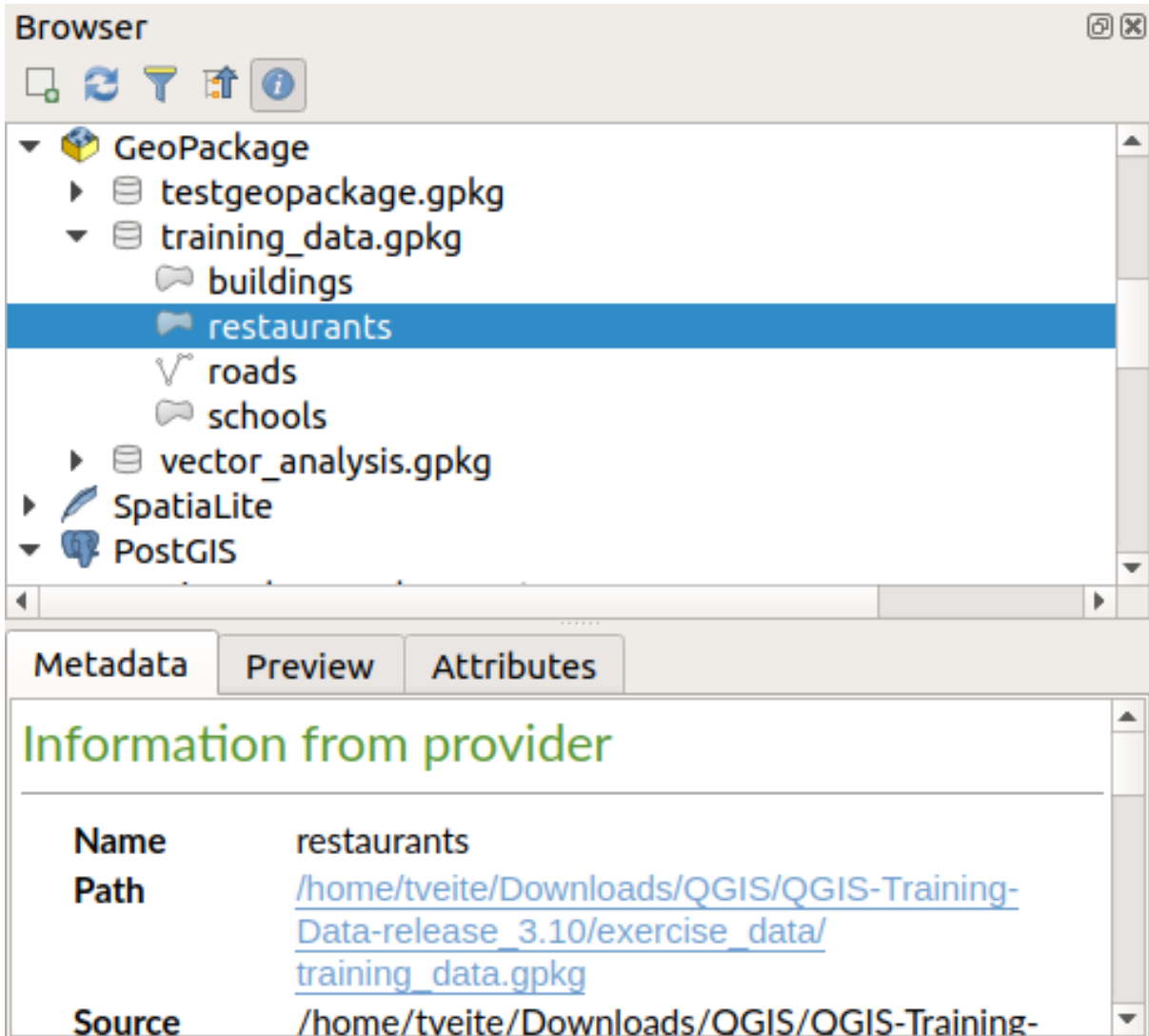



图 8.2: 属性控件

通过在 视图 ► 面板激活 浏览器 (2) 面板，可以打开第二个浏览器面板。当在位于浏览器层次结构的不同分支中的资源之间复制图层时，使用两个浏览器面板非常有用。

8.1 从浏览器打开或运行资源

在浏览器面板可以完成很多工作

- 通过双击、拖动到地图画布或单击  添加选中的图层 按钮 (选择图层后)，将矢量、栅格和网孔图层添加到地图
- 双击或拖动到地图画布，运行 Python 脚本 (包括处理算法)
- 双击或拖动到地图画布，运行模型
- 使用上下文菜单从 QGIS 工程文件中提取符号...
- 使用默认应用程序打开文件 (在上下文菜单中的 打开 <file type> 的外部编辑器...)。例如：HTML 文件、电子表格、图像、PDF、文本文件...
- 复制条目

以下列出了在顶级条目下排序的不同资源组的特定于资源的操作。

8.2 浏览器面板顶级条目

8.2.1 收藏夹

经常使用的文件系统位置可以标记为收藏夹，您标记的内容将显示在此处。

除了主目录中描述的操作外，上下文菜单还允许您 **重命名收藏条目...** 和 **移除收藏条目**。

8.2.2 空间书签

在这里可以找到您的空间书签，又细分为 **工程书签**和 **用户书签**。

从顶层上下文菜单中，可以创建书签 (**新建空间书签...**)，显示空间书签管理器，**导入空间书签...** 和 **导出空间书签...**。

对于书签条目，您可以 **缩放到书签位置**，**编辑空间书签...** 和 **删除空间书签**

8.2.3 主目录

您的文件系统主目录/文件夹。通过右键单击条目并选择 **添加到收藏夹**，该位置将添加到 **收藏夹**。从上下文菜单中，您还可以

- **添加目录、Geopackage 或 ESRI Shapefile 格式数据集 (添加)**
- **隐藏目录 (从浏览器中隐藏)**
- **切换快速扫描此目录**
- **在文件管理器中打开目录 (打开目录)**
- **在终端窗口中打开目录 (在终端打开)**
- **查看属性 (属性..., 目录属性...)**

8.2.4 /

您的文件系统根目录或文件夹。

8.2.5 Geopackage

Geopackage 文件或数据库。从顶层上下文菜单，可以创建 **Geopackage 文件 / 数据库 (创建数据库...)** 或者添加现有 **Geopackage 文件 / 数据库 (新建连接...)**。

通过每个 Geopackage 的上下文菜单，可以将其从列表中移除 (**移除连接...**)，将新图层或表添加到 Geopackage (**创建新图层或表...**)，删除 Geopackage (**删除 <name of geopackage>**) 和 **压缩数据库 (Vacuum)**。

对于图层/表条目，您可以

- **重命名 (重命名图层 <layer name>...)**
- **导出 (导出图层 ► 到文件)**
- **添加到工程 添加图层到工程**
- **删除 (删除图层)**
- **查看属性 (图层属性..., 文件属性...)**

8.2.6 SpatiaLite

SpatiaLite 数据库连接。

从顶层上下文菜单中，可以创建 SpatiaLite 文件 / 数据库 (创建数据库...) 或添加现有 SpatiaLite 文件 / 数据库 (新建连接...)。

每个 SpatiaLite 文件的上下文菜单都允许将其删除 (删除)。

对于图层/表条目，您可以

- 导出 (导出图层 ► 到文件)
- 添加到工程 添加图层到工程
- 删除 (删除图层)
- 查看属性 (图层属性...)

8.2.7 PostGIS

PostGIS 数据库连接。

从顶层上下文菜单中，您可以添加新连接 (新建连接...)。

每个连接的上下文菜单允许您 刷新、编辑 编辑连接...、删除 (删除连接) 或 创建架构...。

每个架构的上下文菜单允许您 刷新, 重命名架构... 或 删除架构。

对于图层 / 表，您可以

- 重命名 (重命名表格...)
- 移除其中的内容 (快速清空表格...)
- 导出 (导出图层 ► 到文件)
- 将其添加到工程 (添加图层到工程)
- 删除 (删除图层)
- 查看属性 (图层属性...)

8.2.8 MSSQL

Microsoft SQL Server 连接。

从顶层上下文菜单中，您可以添加新连接 (新建连接...)。

每个连接的上下文菜单允许您 刷新、编辑 编辑连接...、删除 (删除连接) 或 创建架构...。

每个架构的上下文菜单允许您 刷新, 重命名架构... 或 删除架构。

对于图层 / 表，您可以

- 重命名 (重命名表格...)
- 移除其中的内容 (快速清空表格...)
- 导出 (导出图层 ► 到文件)
- 将其添加到工程 (添加图层到工程)
- 删除 (删除图层)
- 查看属性 (图层属性...)

8.2.9 DB2

IBM DB2 数据库连接。

从顶层上下文菜单中，您可以添加新连接 (新建连接...)。

每个连接的上下文菜单允许您 刷新、编辑 编辑连接...、删除 (删除连接) 或 创建架构...。

每个架构的上下文菜单允许您 刷新, 重命名架构... 或 删除架构。

对于图层 / 表，您可以

- 重命名 (重命名表格...)
- 移除其中的内容 (快速清空表格...)
- 导出 (导出图层 ► 到文件)
- 将其添加到工程 (添加图层到工程)
- 删除 (删除图层)
- 查看属性 (图层属性...)

8.2.10 WMS/WMTS

Web 地图服务 (WMS) 和 Web 地图瓦片服务 (WMTS)

从顶层上下文菜单中，您可以添加新连接 (新建连接...)。

每个 WSM/WMTS 服务的上下文菜单允许您 刷新, 编辑... 和删除 (删除)。

通过将图层拖动到地图画布上来添加图层组。

对于 WMS/WMTS 图层条目，您可以

- 导出 (导出图层 ► 到文件)
- 将其添加到工程 (添加图层到工程)
- 查看属性 (图层属性...)

8.2.11 矢量瓦片

矢量瓦片服务

从顶层上下文菜单中，您可以添加现有服务 (新建连接...), 也可以 保存连接... 到从 XML 文件或从 XML 文件 加载连接...。

8.2.12 XYZ 瓦片

XYZ 瓦片服务

从顶层上下文菜单中，您可以添加现有服务 (新建连接...), 也可以 保存连接... 到从 XML 文件或从 XML 文件 加载连接...。

对于 XYZ 瓦片服务条目，您可以

- 编辑 (编辑...)
- 删除 (删除)
- 导出 (导出图层 ► 到文件)
- 添加到工程 添加图层到工程
- 查看属性 (图层属性...)

8.2.13 WCS

Web Coverage 服务

从顶层上下文菜单中，您可以添加新连接 (新建连接...)。

每个 WCS 的上下文菜单允许您 刷新、编辑... 和删除 (删除)。

对于 WCS 图层条目，您可以

- 导出 (导出图层 ► 到文件)
- 将其添加到工程 (添加图层到工程)
- 查看属性 (图层属性...)

8.2.14 WFS / OGC API - 要素

Web 要素服务 (WFS) 和 OGC API - 要素服务 (aka WFS3)

从顶层上下文菜单中，您可以添加新连接 (新建连接...)。

每个 WFS 的上下文菜单允许您 刷新、编辑... 和删除 (删除)。

对于 WFS 图层条目，您可以

- 导出 (导出图层 ► 到文件)
- 将其添加到工程 (添加图层到工程)
- 查看属性 (图层属性...)

8.2.15 OWS

在这里，您将找到所有开放式 Web 服务 (OWS) 的只读列表 - WMS / WCS / WFS / ...

8.2.16 ArcGIS 地图服务器

8.2.17 ArcGIS 要素服务器

8.2.18 GeoNode

从顶层上下文菜单中，您可以添加新连接 (新建连接...)。

每个服务器的上下文菜单允许您 刷新、编辑... 和删除 (删除)。

对于服务器图层条目，您可以

- 导出 (导出图层 ► 到文件)
- 将其添加到工程 (添加图层到工程)
- 查看属性 (图层属性...)

8.3 资源

- 工程文件，QGIS 工程文件的上下文菜单允许您：
 - 打开 (打开工程)
 - 提取符号 (提取符号...) - 打开样式管理器，允许您将符号导出为 XML 文件，将符号添加到默认样式或导出为 PNG 或 SVG。
 - 查看属性 (文件属性...)

可以展开工程文件以查看其图层，图层的上下文菜单提供与浏览器中其他位置相同的操作。






- QGIS 图层定义文件 (QLR)。上下文菜单中提供了以下操作：
 - 导出 (导出图层 ► 到文件)
 - 将其添加到工程 (添加图层到工程)
 - 查看属性 (图层属性...)
- 数据处理模型 (.model3)。上下文菜单中提供了以下操作：
 - 运行模型...
 - 编辑模型...
- QGIS 打印设计器模板 (QPT)。从上下文菜单中可以执行以下操作：
 - (由模板新建布局)
- Python 脚本 (.py)。上下文菜单中提供了以下操作：
 - (运行脚本...)
 - (使用外部编辑器打开)
- 可识别的栅格格式。上下文菜单中提供了以下操作：
 - 删除 (删除文件 <dataset name>)
 - 导出 (导出图层 ► 到文件)
 - 将其添加到工程 (添加图层到工程)
 - 查看属性 (图层属性..., 文件属性...)

对于某些格式，您还可以打开 <file type> 外部编辑器...



- 可识别的矢量格式。上下文菜单中提供了以下操作：
 - 删除 (删除文件 <dataset name>)
 - 导出 (导出图层 ► 到文件)
 - 将其添加到工程 (添加图层到工程)
 - 查看属性 (图层属性..., 文件属性...)

对于某些格式，您还可以打开 <file type> 外部编辑器...

QGIS 是高度可配置的。通过 设置菜单，提供了以下不同工具：

-  样式管理器...: 创建和管理符号, 样式和颜色渐变.
-  自定义投影...: 创建您自己的坐标参照系.
-  键盘快捷键...: 定义您自己的一组键盘快捷键. 此外, 工程属性 可以在每个 QGIS 会话期间替换它们 (可在 工程菜单下访问).
-  界面自定义...: 配置应用程序界面, 隐藏您可能不需要的对话框或工具.
-  选项...: 设置全局选项 以应用于软件的不同领域. 这些首选项保存在当前用户配置 设置中, 并在使用此配置文件打开新工程时默认应用.

9.1 选项

 可以使用 选项对话框选择 QGIS 的一些基本选项。选择菜单选项 设置 ►  选项, 您可以根据需要修改选项, 某些更改可能需要重新启动 QGIS 才能生效。

下面介绍了可以自定义选项的选项卡。

注解：插件可以在选项对话框中嵌入它们的设置

虽然以下只展示了核心设置，但请注意，通过 *installed plugins* 在标准选项对话框中插件可展现自己的选项，并以此扩展列表。这避免了每个插件都有自己的配置对话框，并为它们提供额外的菜单项...

9.1.1 通用设置




本地化语言环境

默认情况下，QGIS 依赖于您的操作系统配置来设置语言和控制数值。启用该组允许您自定义自己的行为习惯。


- 从用户界面语言选择要应用于 GUI 的语言
- 在本地化 (数字、日期和货币格式) 选择输入和呈现日期和数值的语言系统
- 显示数字分组 (千位) 分隔符

框架底部会显示选中设置的摘要以及这些设置的作用。



应用软件

- 选择样式 (重启 QGIS 生效), 即窗口控件在对话框中的外观和位置。可能的值取决于您的操作系统。
- 定义用户界面主题 (重启 QGIS 生效) . 可以是 'default', 'Night Mapping' 或 'Blend of Gray'
- 定义图标大小 
- 定义字体和字号. 字体可以是  Qt 默认或用户从操作系统安装的字体进行选择
- 更改限时信息或对话框超时
- 启动时隐藏初始界面
- 在欢迎页面显示 QGIS 动态资讯: 在欢迎页面上显示精选的 QGIS 动态消息, 让您直接了解项目新闻 (用户/开发人员会议日期和摘要、社区调查、发布公告、各种提示...)
- 打开 QGIS 软件时检查更新如果发布了新版本, 将在启动软件时检查并通知您
- 使用操作系统颜色选择对话框 (参见 *Color Selector*)
- 非模态数据源管理器对话框保持数据源管理器对话框处于打开状态, 并允许向工程添加图层时与 QGIS 界面进行交互

工程文件

- 启动时打开工程为
 - '欢迎页面' (默认): 可以显示 [用户配置](#) 的动态资讯、工程模板和最近工程 (带有缩略图)。默认情况下不打开任何工程。
 - '新建': 基于默认模板打开一个新工程
 - '最近': 重新打开上次保存的工程
 - '特定': 打开一个特定的工程, 使用 ... 按钮定义默认情况下要使用的工程文件。
- 由默认工程创建新工程. 您可以选择 [设置当前工程为默认工程](#) 或 [重置为默认](#). 可以浏览文件并定义目录, 并在其中找到用户定义的工程模板。如果您先激活 由默认工程创建新工程, 然后将工程保存在工程模板文件夹中, 这将被添加到 [工程 ▶ 由模板新建列表](#) 中。
- 需要时提示保存工程和数据源更改情况避免丢失所做的更改。
- 移除图层时提示确认
- 当打开旧版 QGIS 格式的工程文件时提出警告. 您始终可以打开使用较旧版本 QGIS 创建的工程, 但一旦保存该工程, 尝试使用较低版本打开可能会失败, 这是因为该版本没有相对于高版本可用的功能。
- 启用宏 . 创建此选项是为了处理用于对工程事件执行操作的宏。您可以在 '从不'、'询问'、'仅在该会话中' 和 '始终 (不建议)' 之间进行选择。

- 默认工程文件格式

-  **QGS**: 存档文件格式，嵌入辅助数据 (请参见辅助数据)
-  **QGZ**: 工程文件以明码文本保存，未嵌入辅助数据: 辅助数据存储于单独的 .qgd 文件，工程文件则在同一目录存储不同后缀名的同名文件。

9.1.2 系统设置

SVG 路径

添加或移除可伸缩矢量图形 (SVG) 符号的搜索路径。这些 SVG 文件可用于符号化、标注要素或装饰地图构图。

在符号或标注中使用 SVG 文件时，QGIS 允许您：

- 从文件系统加载文件：通过文件路径识别文件，QGIS 需要解析路径以显示相对应的图像
- 从远程 URL 加载文件：如上所述，只有成功检索远程资源时才会加载图像
- 将 SVG 文件嵌入到项：文件嵌入到当前工程、样式数据库或打印布局模板，SVG 文件始终作为项的一部分进行渲染。这是一种使用自定义 SVG 符号创建自包含工程的便捷方法，可在 QGIS 的不同用户和安装之间轻松共享。

还可以从符号或标注提取嵌入的 SVG 文件，并将其保存在磁盘上。

注解： 上述用于在工程中加载和存储 SVG 文件的选项也适用于您可能希望用于自定义符号、标注或装饰的栅格图像。

插件路径

添加或移除附加 C++ 插件库的搜索路径。

文档路径

添加或移除用于 QGIS 帮助的文档路径。默认情况下，会添加与所用版本对应的官方在线用户手册的链接。但是，您可以添加其他链接并从上到下排列它们的优先级：每次单击对话框中的帮助按钮时，都会优先检查最上面的链接，如果没有找到相应的页面，则会尝试下一个链接，依此类推。

注解： 文档仅针对 QGIS 长期发行版 (LTR) 进行版本控制和翻译，这意味着如果您运行的是常规版本 (如 QGIS 3.0)，帮助按钮默认打开下一版 LTR 手册页面 (即 3.4 LTR)，其中可能包含较新版本 (3.2 和 3.4) 的功能描述。如果没有可用的 LTR 文档，则使用较新和开发版本功能描述的测试文档。

设置

如果进行了任何自定义设置，它可以帮助您重置用户界面为默认设置 (重启 QGIS 生效)。

环境

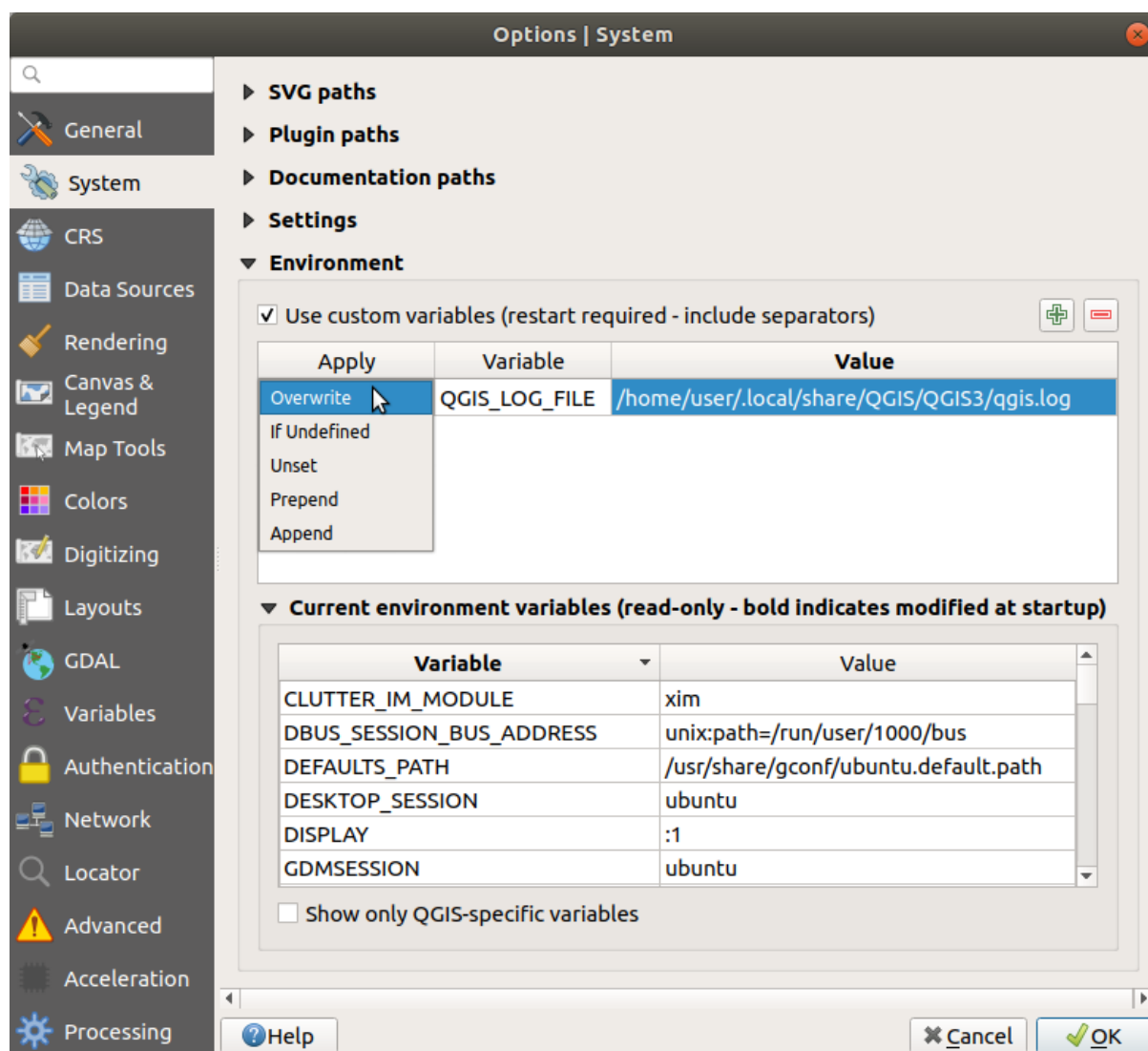


图 9.1: QGIS 中的系统环境变量

可以在 **环境**组中查看和配置系统环境变量。这对平台很有用，例如 Mac，因为 GUI 应用程序不一定继承用户的 Shell 环境，它还可以用于设置和查看由处理工具箱 (例如 SAGA、GRASS) 控制的外部工具集的环境变量，以及打开源代码特定部分的调试输出。

使用自定义变量 (重启 *QGIS* 生效 - 包括分隔符)。您可以添加和 移除变量。已定义的环境变量显示在当前环境变量中，可以通过勾选 仅显示 *QGIS* 特殊变量进行过滤。

9.1.3 CRS 设置

注解：有关 QGIS 如何处理图层投影的更多信息，请阅读 *Working with Projections* 专区。

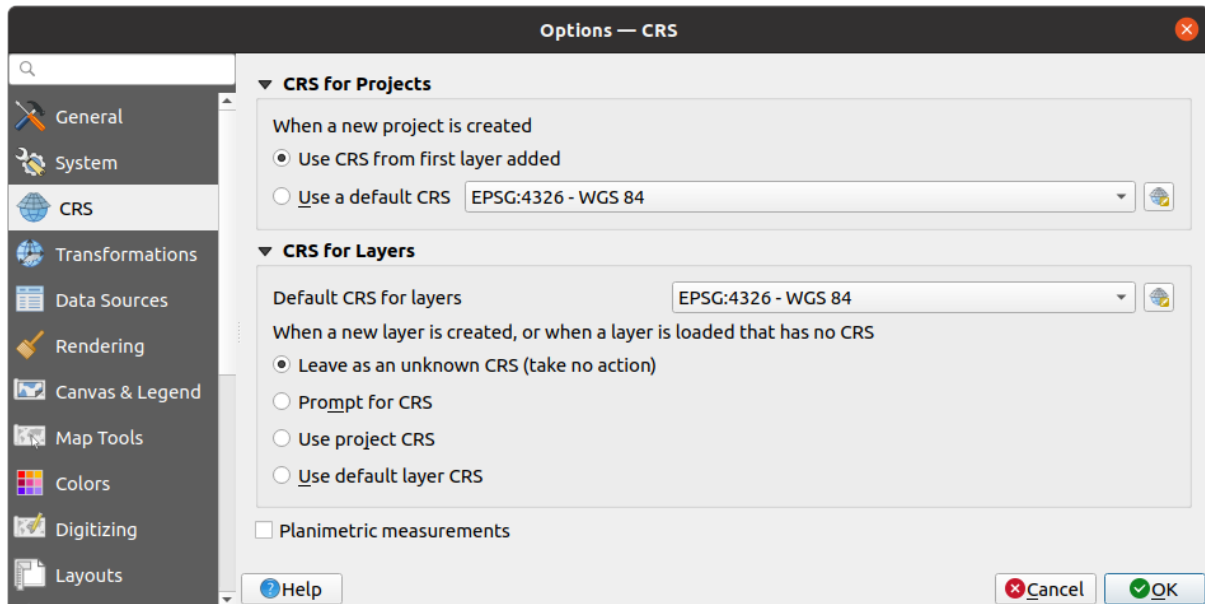


图 9.2: QGIS 中的 CRS 设置

工程的 CRS

有个选项可以自动设置新建工程的 CRS：

- 使用添加的第一个图层的 CRS: 工程的 CRS 将设置为加载到其中的第一个图层的 CRS
- 使用默认 CRS: 预选 CRS 将默认应用于新建工程，并在向工程添加图层时保持不变。

该选项将保存供后续 QGIS 会话使用，工程的坐标参照系仍然可以从 工程 ► 工程属性... ► CRS 选项卡进行重新定义。

图层 CRS


默认图层 CRS: 选择创建图层时要使用的默认 CRS

还可以定义创建新图层或加载不含 CRS 的图层时要执行的操作。

- 保留未知 CRS (无动作)
- 提示选择 CRS
- 使用工程 CRS
- 使用默认 CRS

平面测量: 为新创建的工程设置“平面测量”属性的默认值。

9.1.4 变换设置

 变换选项卡可帮助您将图层加载到工程或重投影图层时要应用的设置坐标变换和操作。

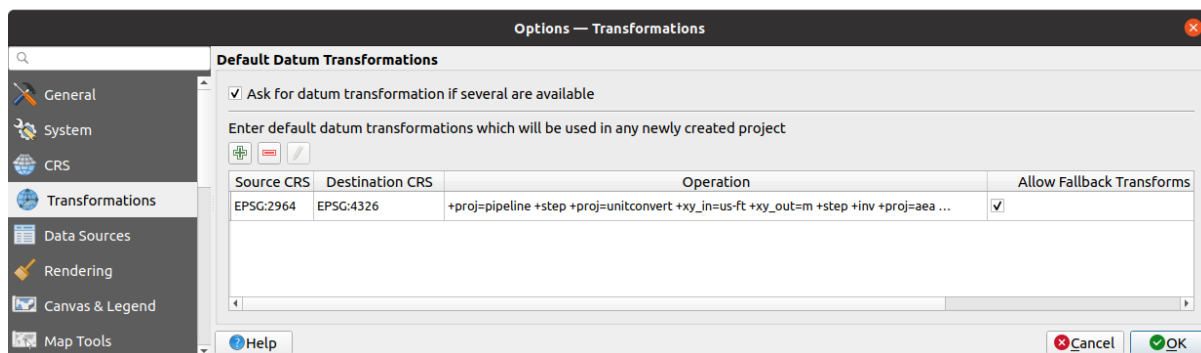


图 9.3: 变换设置

默认基准面变换

在该组中，您可以控制是否应将图层重投影到另一个 CRS：

- 使用 QGIS 默认变换设置自动处理；
- 和/或更多由您通过自定义首选项控制，例如：
 - 存在多个时询问基准面变换
 - 默认情况下要应用的预定义基准变换列表。更多信息，请参阅 *Datum Transformations* 。

9.1.5 数据源设置

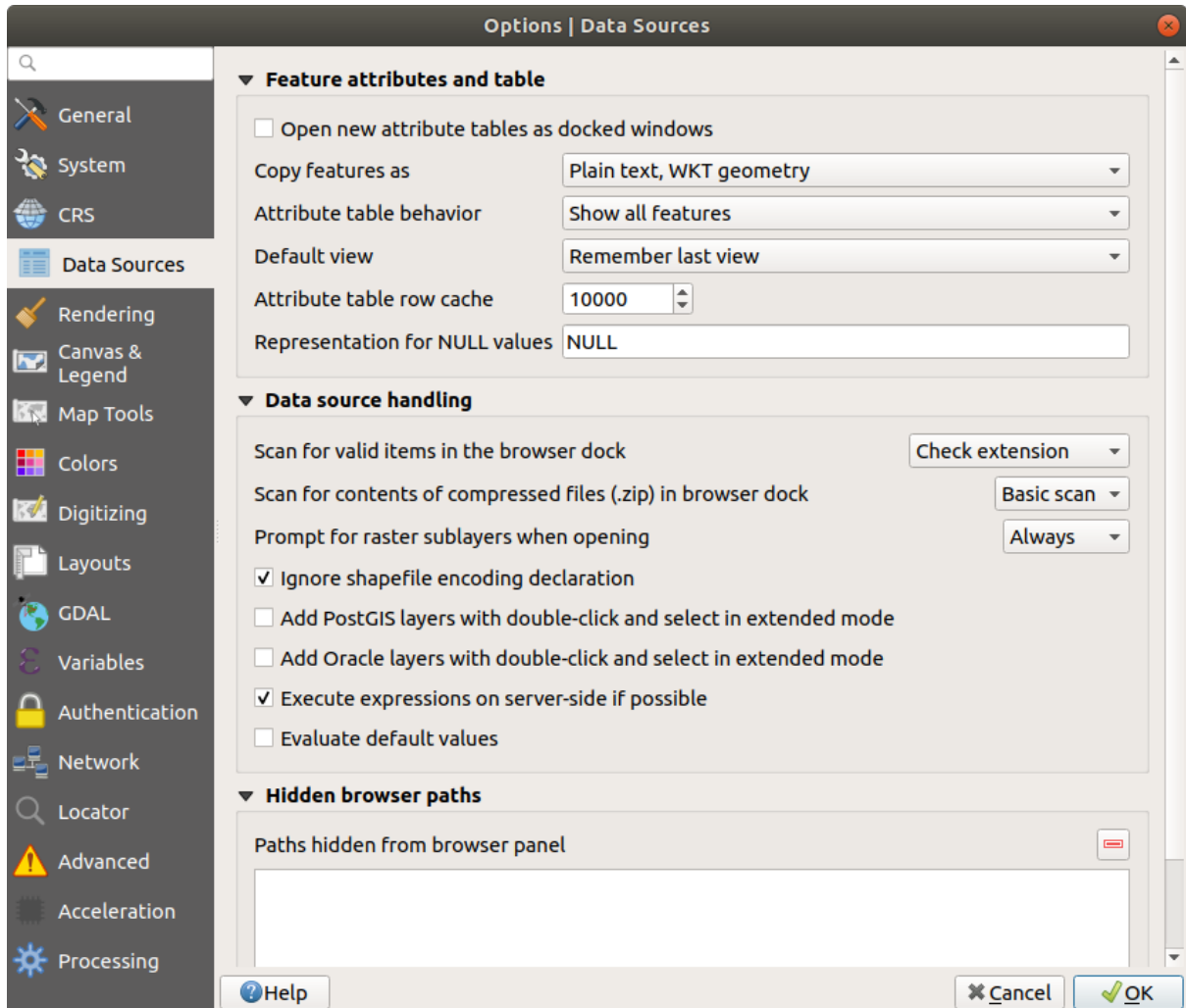


图 9.4: QGIS 中的数据源设置

要素属性和表格



- 以停靠窗口方式打开属性表
- 在其他应用程序中粘贴要素时 要素复制为 '纯文本, 无几何图形', '纯文本, WKT 几何图形', 或 'GeoJSON'
- 属性表行为 : 默认打开属性表时设置过滤选项。有三种可能: '显示所有要素', '显示选中的要素' 和 '显示地图上可见的要素'。
- 默认视图: 定义每次打开属性表的视图模式。它可以是 '记住最后视图', '表格视图' 或 '表单视图'。
- 属性表记录缓存 。该行缓存可以保存最后加载的 N 个属性记录, 以便更快地处理属性表。关闭属性表时, 缓存将被删除。
- 空值表示为. 在此, 您可以为包含空值 (NULL) 的数据字段定义一个值。

小技巧: 改进大数据属性表的打开速度

处理有大量记录的图层时, 打开属性表的速度可能会很慢, 因为对话框会索取图层中的所有行。将 属性表行为 设置为 **显示地图可见要素** 将使 QGIS 在打开表时仅请求当前地图画布中的要素, 从而允许快速加载数据。

请注意，此属性表实例中的数据将始终关联到打开时使用的画布范围，这意味着选择 **显示所有要素** 将不会显示新要素。但是，您可以通过更改画布范围并在属性表选择 **显示地图可见要素** 选项来更新显示的要素集。

数据源处理

- 在浏览器停靠区扫描有效项 。您可以在‘检查扩展名’和‘检查文件内容’之间进行选择。
- 在浏览器停靠区扫描压缩文件 (.zip) 的内容  定义在查询此类文件时浏览器面板底部的控件信息的详细程度，‘否’，‘基本扫描’和‘深度扫描’是可能的选项。
- 打开栅格子图层时显示提示。有些栅格支持子图层 --- 它们在 GDAL 中被称为子数据集。例如 netCDF 文件 --- 如果有许多 netCDF 变量，GDAL 会将每个变量视为一个子数据集。该选项允许您控制在打开包含子图层的文件时如何处理。您有以下选择：
 - ‘始终’：始终询问 (如果存在子图层)
 - ‘需要时’：询问图层是否没有波段，但有子图层
 - ‘从不’：从不提示，不会加载任何内容
 - ‘全部加载’：从不提示，但加载所有子图层
- 忽略 Shapefile 编码声明。如果 Shapefile 包含编码信息，QGIS 将忽略该信息。
- 如果可能的话，在服务器端执行表达式：当从数据源请求要素时，QGIS 将尝试通过直接向服务器发送过滤条件来优化请求，并且只下载符合条件的要素。例如，如果用户界面上的列表只应列出居住在伯尔尼的农民，QGIS 将向数据库发送 WHERE "hometown" = 'Bern'。在某些情况下，筛选条件过于复杂，无法从 QGIS 表达式转换为与数据库兼容的 SQL。这种情况下，QGIS 将下载整个数据，并在本地进行过滤以确保安全，这将大大降低性能。
禁用此选项，可能会强制 QGIS 始终下载整个数据并在本地进行过滤，这会降低性能。此选项是一种安全中断，只有您在识别出 QGIS 表达式翻译引擎的错误行为时，才应禁用。

隐藏浏览器路径

此控件列出了您在浏览器面板中选择隐藏的所有文件夹，从列表中移除文件夹将使其在浏览器面板中可用。

本地化数据路径

可以对任何类型的基于文件的数据源使用本地路径，它们是用于抽象数据源位置的路径列表。例如，如果 C:\my_maps 列在本地路径中，则 C:\my_maps\my_country\ortho.tif 作为数据源的图层将在工程中保存后使用 localized:my_country\ortho.tif。

路径按优先顺序列出，换句话说，QGIS 将首先在第一个路径中查找文件，然后在第二个路径中查找文件，以此类推。

9.1.6 渲染设置

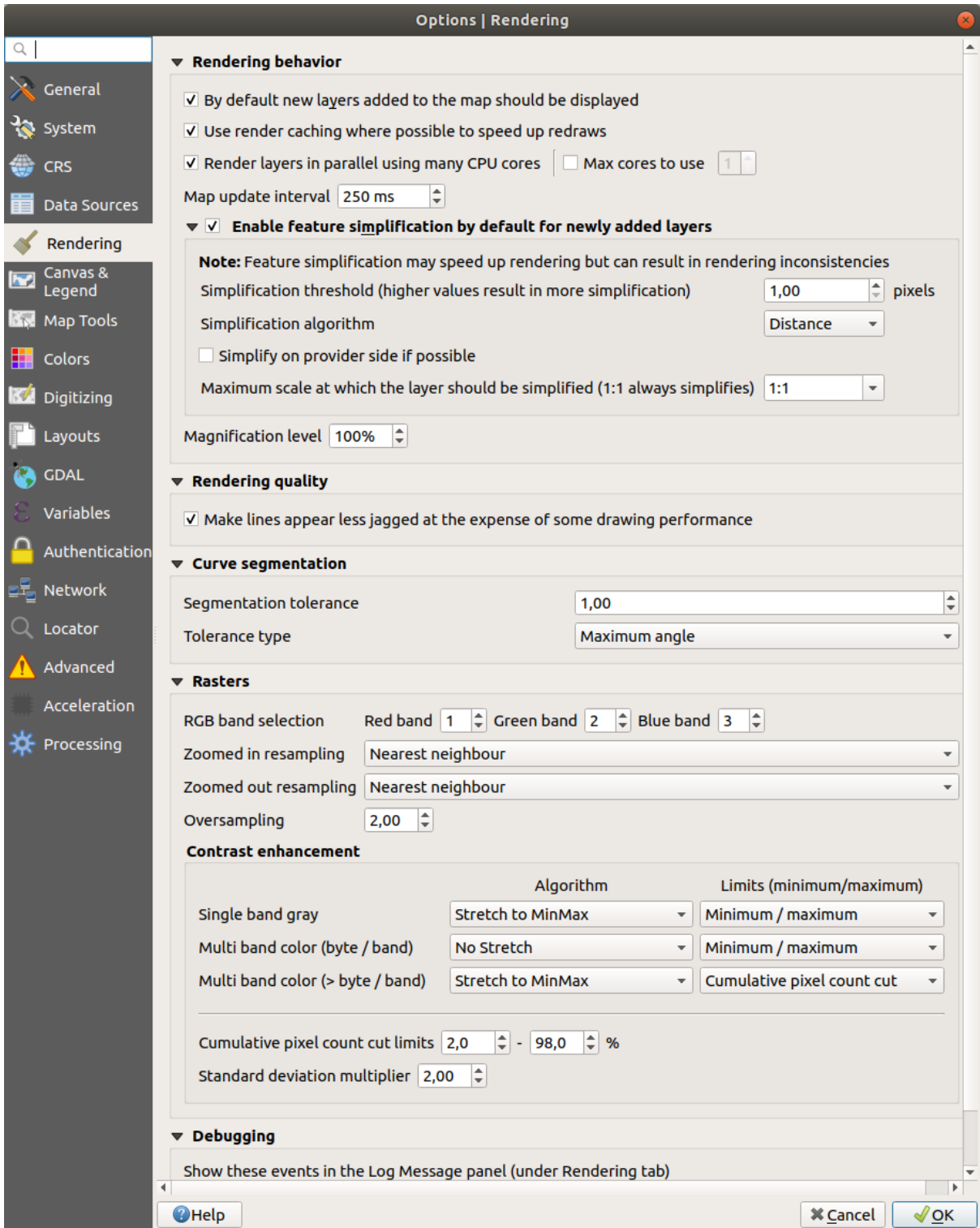


图 9.5: 工程属性对话框的渲染选项卡

渲染行为

- 默认显示地图中新添加的图层: 在加载多个图层时, 取消勾选此选项会很有用, 这可以避免在画布中渲染每个新图层和减慢处理过程

- 在可能的情况下使用渲染缓存以加速图形重绘
- 使用多个 CPU 核心并行渲染图层
- 最大可用核心数
- 地图刷新间隔 (默认是 250 毫秒)
- 默认对新添加的图层启用要素简化
- 简化阈值
- 简化算法: 该选项会对要素进行局部“动态 (on-the-fly)”简化, 并加快几何图形渲染速度。它不会更改从数据提供程序获取的几何图形。当您有使用要素几何图形的表达式 (例如面积计算) 时, 这一点很重要 - 它确保这些计算是在原始几何图形上而不是在简化几何图形上完成的。为此, QGIS 提供了三种算法: ‘距离’ (默认)、‘捕捉到网格’ 和 ‘Visvalingam’。
- 若可能, 在数据源端简化: 几何图形由提供程序 (PostGIS、Oracle 等) 进行简化, 与本地简化不同, 基于几何图形的计算可能会受到影响
- 图层简化最大比例
- 放大倍率 (参见 *magnifier*)

注解: 除了全局设置外, 还可以从其 图层属性 ► 渲染菜单中为任何特定图层设置要素简化

渲染质量

- 牺牲绘制效率使线的显示更光滑

曲线分割

- 分割容差: 此设置控制圆弧的渲染方式。**较小的**最大角度 (两个连续顶点与曲线中心之间的角度, 以度为单位) 或最大差值 (两个顶点的线段与曲线之间的距离, 单位为地图单位), 渲染时将使用 **更多** 直线段。
- 容差类型: 可以是最大角度, 也可以是近似值和曲线之间的最大差值。

栅格

- 通过 RGB 波段选择, 您可以定义红色、绿色和蓝色波段的编号。
- 可以定义放大重采样和缩小重采样方法。对于放大重采样您可以在三种重采样方法中选择: “最近邻插值”、“双线性”和“三次方”。对于: `guilabel:` 缩小重采样可以在“最近邻插值”和“平均”之间进行选择。您还可以设置过采样值 (介于 0.0 和 99.99 之间 - 较大的值意味着 QGIS 更大的工作量 - 默认值为 2.0)。

对比度增强

对比度增强选项可应用于单波段灰度, 多波段颜色 (字节/波段) 或多波段颜色 (> 字节/波段)。对于每个选项, 您可以设置:

- 要使用的算法, 其值可以是 ‘不拉伸’, ‘拉伸至极小极大’, ‘拉伸并裁剪至极小极大’ 或 ‘裁剪至极小极大’
- 要应用的限制 (最小/最大值), 其值包括 ‘累计像素总数裁剪’, ‘最小值/最大值’, ‘平均 +/- 标准差’。

对于栅格渲染, 还可以定义以下选项:

- 累计像素总数裁剪限制
- 标准差乘数

调试

- 地图画布刷新到 日志信息面板的调试渲染持续时间。

9.1.7 画布和图例设置

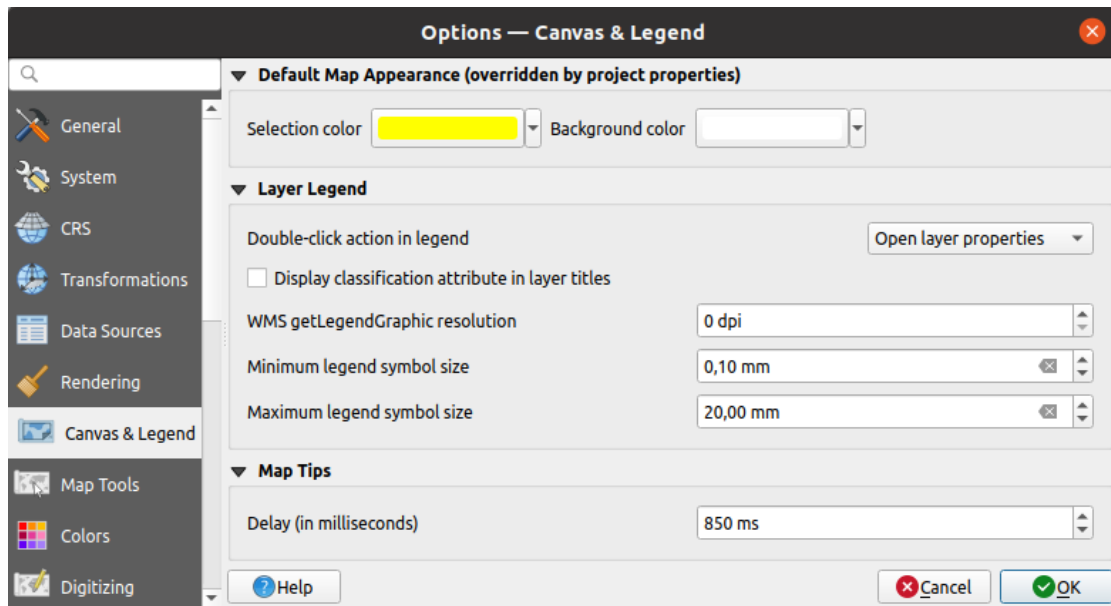




图 9.6: 画布和图例设置

这些属性允许您设置：

- **默认地图外观 (由工程属性覆盖):** 包括选中要素颜色和背景颜色。
- **图层图例交互操作:**
 - 双击图例的操作 . 您可以双击'打开图层属性对话框','打开属性表'或'打开图层样式停靠栏'。
 - 在图层面板  显示类别属性名称, 例如, 在应用分类或基于规则的渲染器时 (有关更多信息, 请参阅 *Symbology Properties*)。
 - WMS 获取图例图形分辨率
 - 控制 图层面板符号大小显示的最小图例符号尺寸和最大图例符号尺寸
- 以毫秒为单位的图层地图提示 显示 延迟

9.1.8 地图工具设置

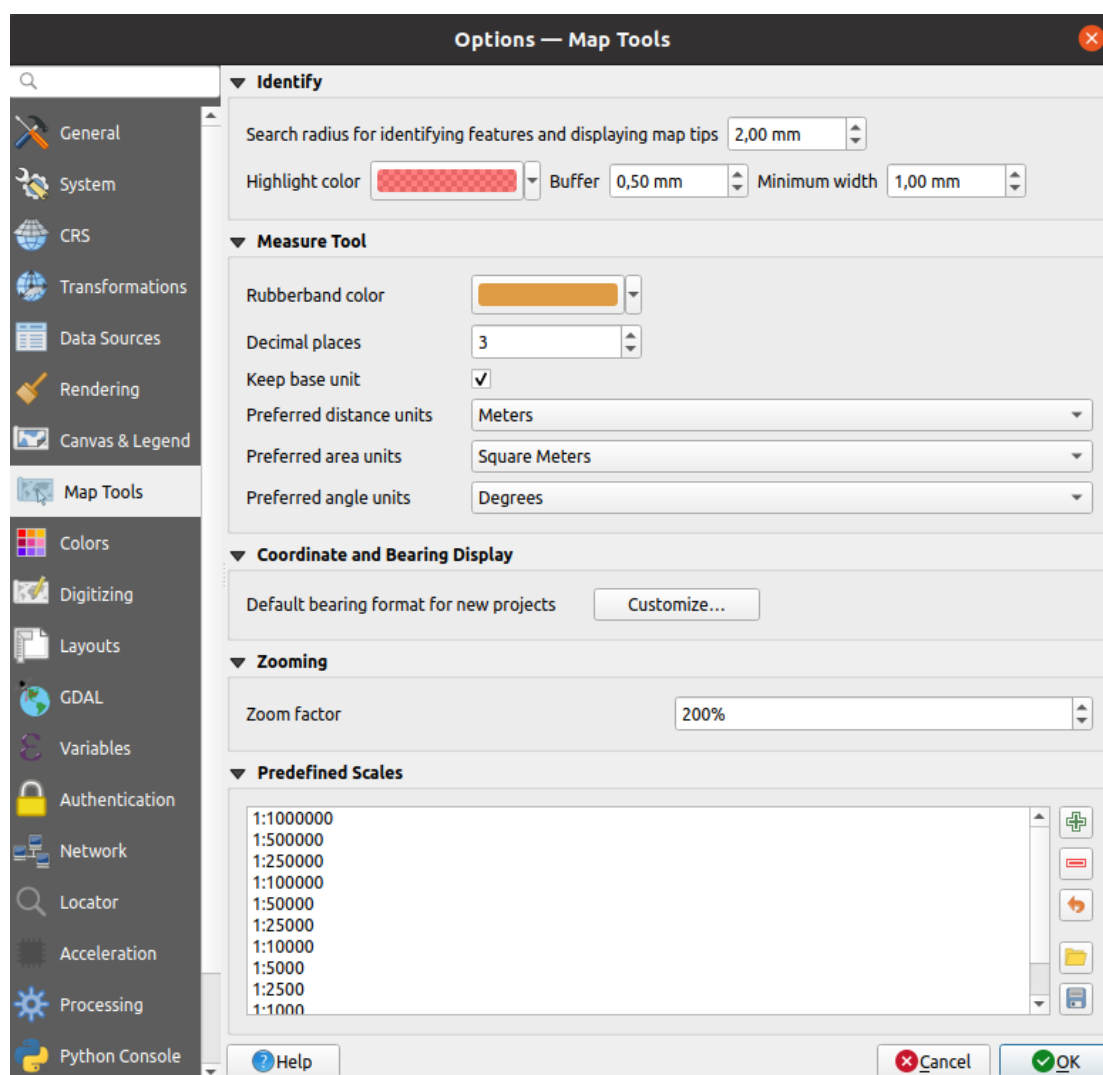


图 9.7: QGIS 中的地图工具设置

此选项卡提供了有关识别工具行为的一些选项。

- 识别要素和显示地图提示的搜索半径是一个容差距离，只要在该容差范围内单击，识别工具就会描述结果。
- 突出显示颜色允许您选择突出显示所识别要素的颜色。
- 轮廓确定要从识别突出显示的轮廓渲染的缓冲距离。
- 最小宽度决定突出显示对象轮廓的宽度。

测量工具

- 定义测量工具的橡皮筋颜色
- 定义小数位数
- 保持基本单位不自动转换大数目单位 (例如，米到千米)
- 首选距离单位: 选项有 '米', '千米', '英尺', '码', '英里', '海里', '厘米', '毫米', '度' 或 '地图单位'
- 首选面积单位: 选项有 '平方米', '平方千米', '平方英尺', '平方码', '平方英里', '公顷', '英亩', '平方海里', '平方厘米', '平方毫米', '平方度' 或 '地图单位'

- 首选角度单位: 选项有'度','弧度','百分度','角分','角秒','转数/圈数 (Turns/revolutions)', 毫弧度 (SI 定义) 或密位 (北约军用定义)



坐标和方位角显示

- 定义 新建工程默认方位角格式: 用于平移地图画布时在状态栏显示鼠标坐标。可以在工程属性对话框中对其覆盖设置。

平移和缩放

- 为缩放工具或鼠标滚轮定义 缩放因子

预定义比例

在此, 可以找到预定义比例列表。使用  和  按钮, 您可以添加或移除自定义比例, 还可以导入或导出比例从 / 到 .XML 文件。请注意, 您仍然可以删除更改并重置为预定义列表。

9.1.9 颜色设置

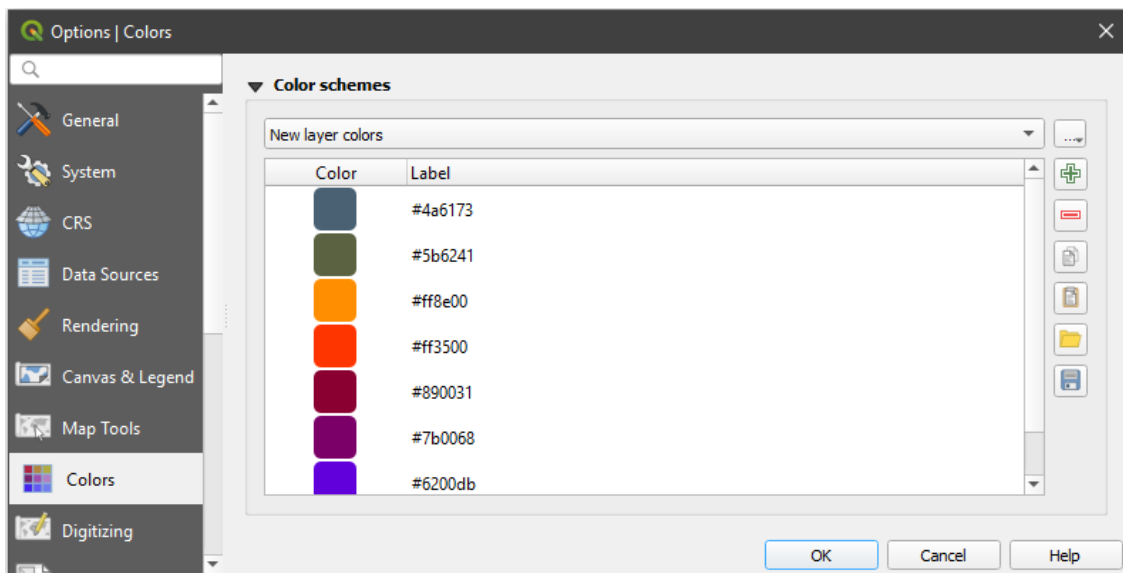








图 9.8: 颜色设置

此菜单允许您在颜色选择控件 中创建或更新整个应用程序使用的颜色调色板。您可以选择:

- 最近使用过的颜色显示最近使用的颜色
- 标准颜色, 默认调色板的颜色
- 工程颜色, 特定于当前工程的一组颜色 (更多信息, 请参阅默认样式属性)
- 新建图层颜色, 将新图层添加到 QGIS 时默认使用的一组颜色
- 或自定义调色板, 您可以使用调色板组合框旁边的 ... 按钮创建或导入调色板。

默认情况下, 不能移除 最近使用过的颜色, 标准颜色和 工程颜色调色板, 并将其设置为显示在颜色按钮下拉列表中。使用 在颜色按钮中显示选项, 还可以将自定义调色板添加到此控件中。

对于任何调色板, 都可以使用框架旁边的工具组管理颜色列表, 即:

-  添加或  移除颜色
-  复制或  粘贴颜色
-  导入或  导出颜色集从/到 .gpl 文件。

双击列表中的颜色，在 **选择颜色** 对话框中对其进行调整或替换，也可以通过双击 标注列 对其进行重命名。

9.1.10 数字化设置

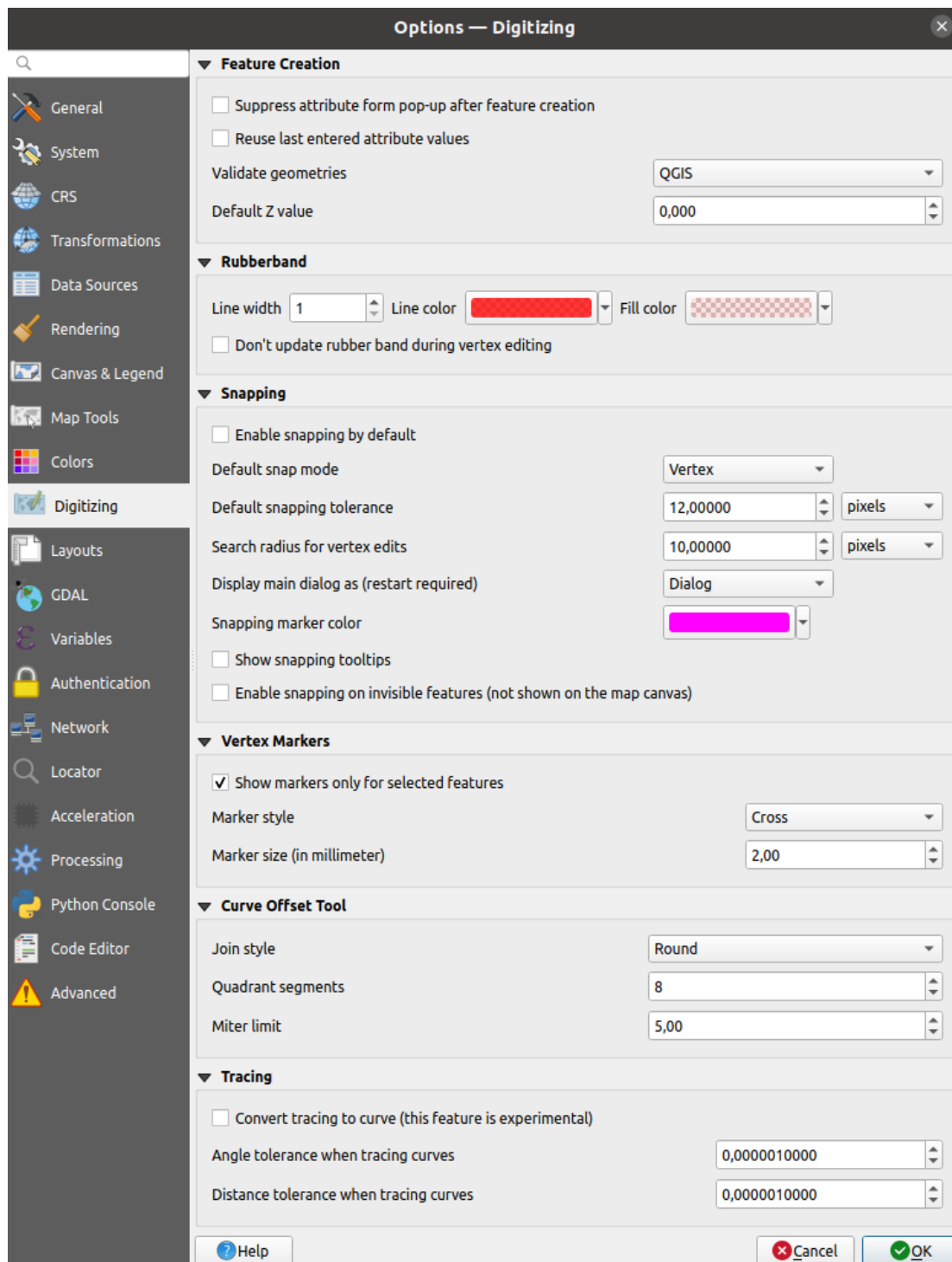


图 9.9: QGIS 中的数字化设置

此选项卡可帮助您在 **编辑矢量图层** (属性和几何图形) 时配置常规设置。


创建要素

- 要素创建后不要弹出属性表单: 可以在每个图层属性对话框中重写此选项。
- 重用最后输入的属性值: 记住每个属性最后使用的值, 并将其用作下一个数字化要素的默认值。每个图层都有效。
- 验证几何图形. 编辑具有多个节点的复杂线和多边形可能会导致渲染速度非常慢。这是因为 QGIS 中的默认验证过程可能需要很多时间。要加快渲染速度, 可以选择 GEOS 几何图形验证 (从 GEOS 3.3 开始) 或将其关闭。GEOS 几何图形验证要快得多, 但缺点是只报告第一个几何图形问题。
请注意, 根据选择, 几何图形错误报告可能会有所不同 (请参阅 *Types of error messages and their meanings*)
- 默认 Z 值用于创建新的 3D 要素。


橡皮筋

- 定义橡皮筋的 线宽, 线颜色和 填充颜色。
- 编辑顶点时不更新橡皮筋。

捕捉

- 默认启用捕捉打开工程时激活捕捉
- 定义 默认捕捉模式  (' 顶点', ' 顶点和线段', ' 线段')
- 以地图单位或像素为单位定义 默认捕捉容差
- 以地图单位或像素为单位定义 顶点编辑的搜索半径
- 主对话框显示为 (重启 QGIS 生效): 设置高级捕捉对话框应显示为" 对话框" 还是" 停靠"。
- 捕捉标记颜色
- 显示捕捉工具提示例如您要捕捉其要素图层的名称, 当多个要素重叠时很有帮助。
- 对不可见的要素 (未在地图画布中显示) 也启用捕捉

顶点标记

- 仅为选中的要素显示标记
- 定义顶点的 标记样式  (' 十字' (默认), ' 半透明圆圈' 或' 无')
- 定义顶点的 标记尺寸 (毫米)

曲线偏移工具

接下来的 3 个选项是指 *Advanced digitizing* 中的  偏移曲线 工具。通过各种设置, 可以影响线偏移的形状。从 GEOS 3.3 开始可以使用这些选项。

- 连接样式: ' 圆角', ' 尖角' 或' 斜角'
- 相限区段
- 尖角限制

追踪

通过激活 将追踪转换为曲线您可以在数字化时创建曲线段。请记住, 您的数据提供程序必须支持此功能。

9.1.11 布局设置

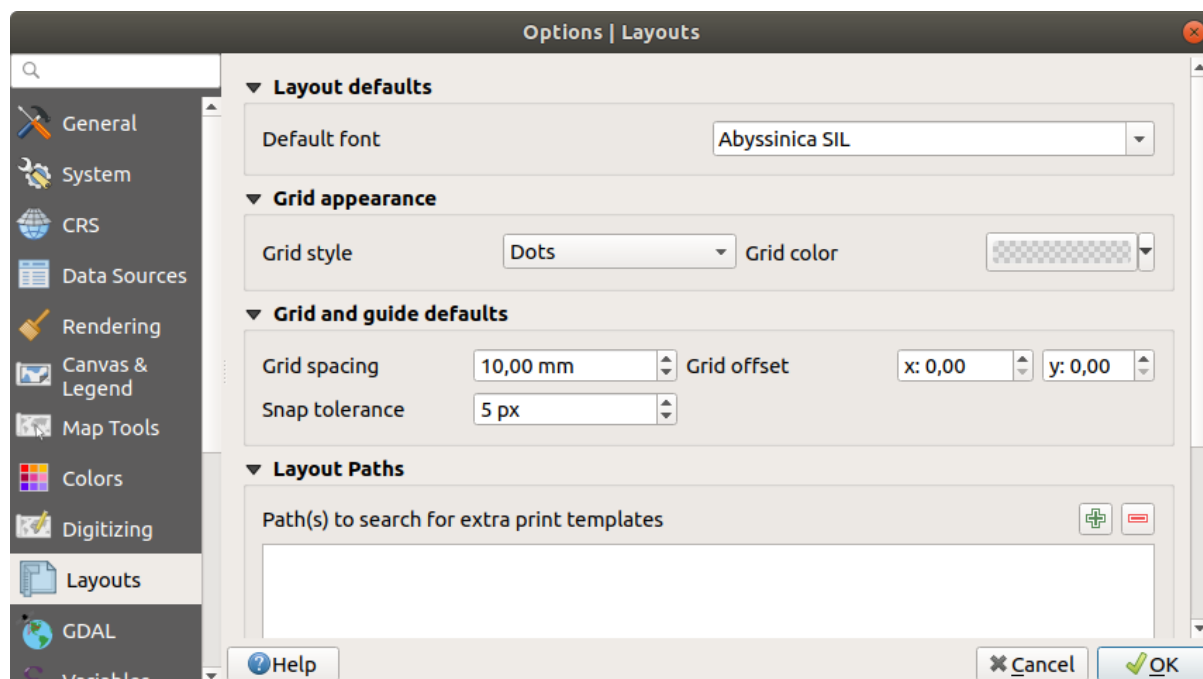


图 9.10: QGIS 中的布局设置

缺省布局

您可以定义在打印布局中使用的默认字体。

网格外观

- 定义网格样式 (‘实线’, ‘点’, ‘十字’)
- 定义网格颜色

网格和参考线默认值

- 定义网格间距 1,00
- 定义 X 和 Y 的网格偏移 1,00
- 定义捕捉容差 1,00

布局路径

- 定义附加打印模板的搜索路径: 包含自定义布局模板的文件夹列表, 可在创建新模板时使用。

9.1.12 GDAL 设置

GDAL 是一个支持大量矢量和栅格格式的地理空间数据的数据交换库。它提供了以这些格式读取和 (通常) 写入数据的驱动程序。GDAL 选项卡显示了栅格和矢量格式的驱动程序及其功能。

栅格驱动选项

此框架提供了自定义支持读写访问的栅格驱动程序行为的方法：

- 编辑创建选项：允许您编辑或添加不同的文件转换配置文件，即在输出栅格文件时使用一组预定义的参数组合（压缩类型和层级、块大小、概述、色度、透明度…）。参数取决于驱动程序。

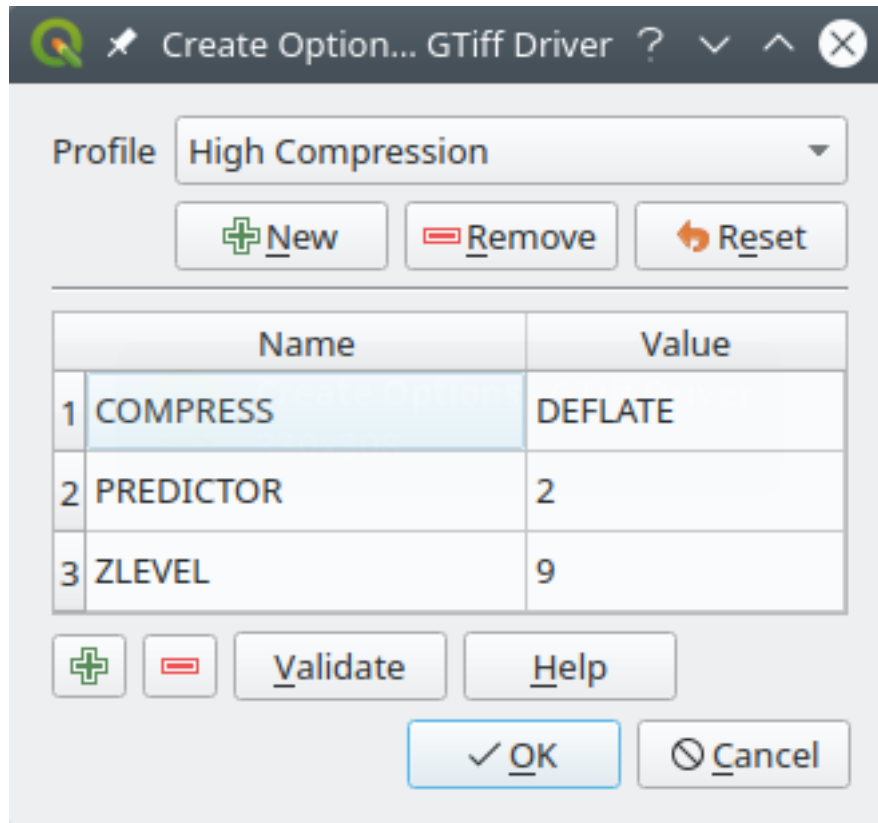




图 9.11: 创建选项配置文件示例 (适用于 GeoTiff)

对话框的上半部分列出了当前配置文件，并允许您添加新配置文件或删除其中任何一个。如果您更改了配置文件的默认参数，也可以将其重置为默认参数。一些驱动程序（例如 GeoTiff）有一些可以使用的配置文件样本。

在对话框底部：

-  按钮允许您添加行以填充参数名称和值
-  按钮删除所选参数
- 单击 验证按钮，检查为给定格式输入的创建选项是否有效
- 使用 帮助按钮查找要使用的参数，或参考 [GDAL 栅格驱动程序文档](#)。

- 编辑金字塔选项

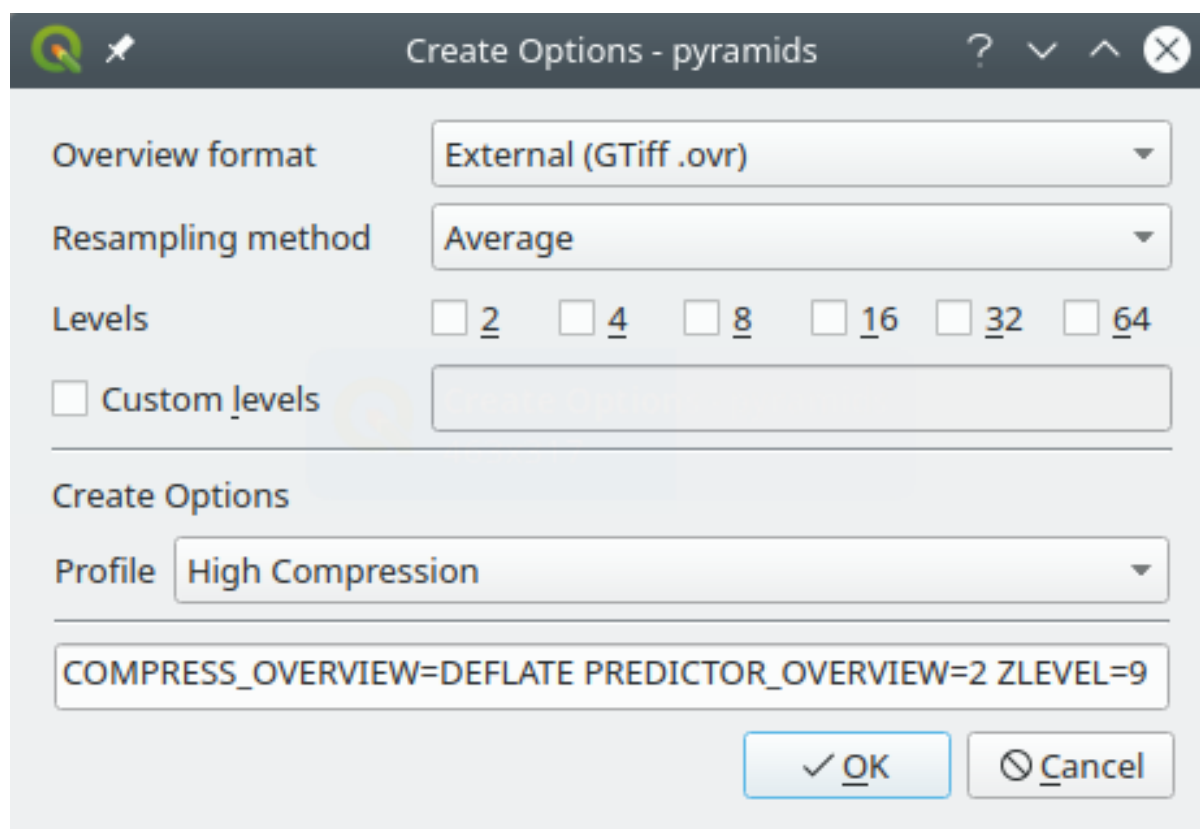


图 9.12: 金字塔配置文件示例

GDAL 栅格和矢量驱动

栅格驱动和 矢量驱动 (在单独的选项卡中) 允许您定义哪个 GDAL 驱动能够读取和/或写入文件, 因为在某些情况下, 有多个 GDAL 驱动可用。

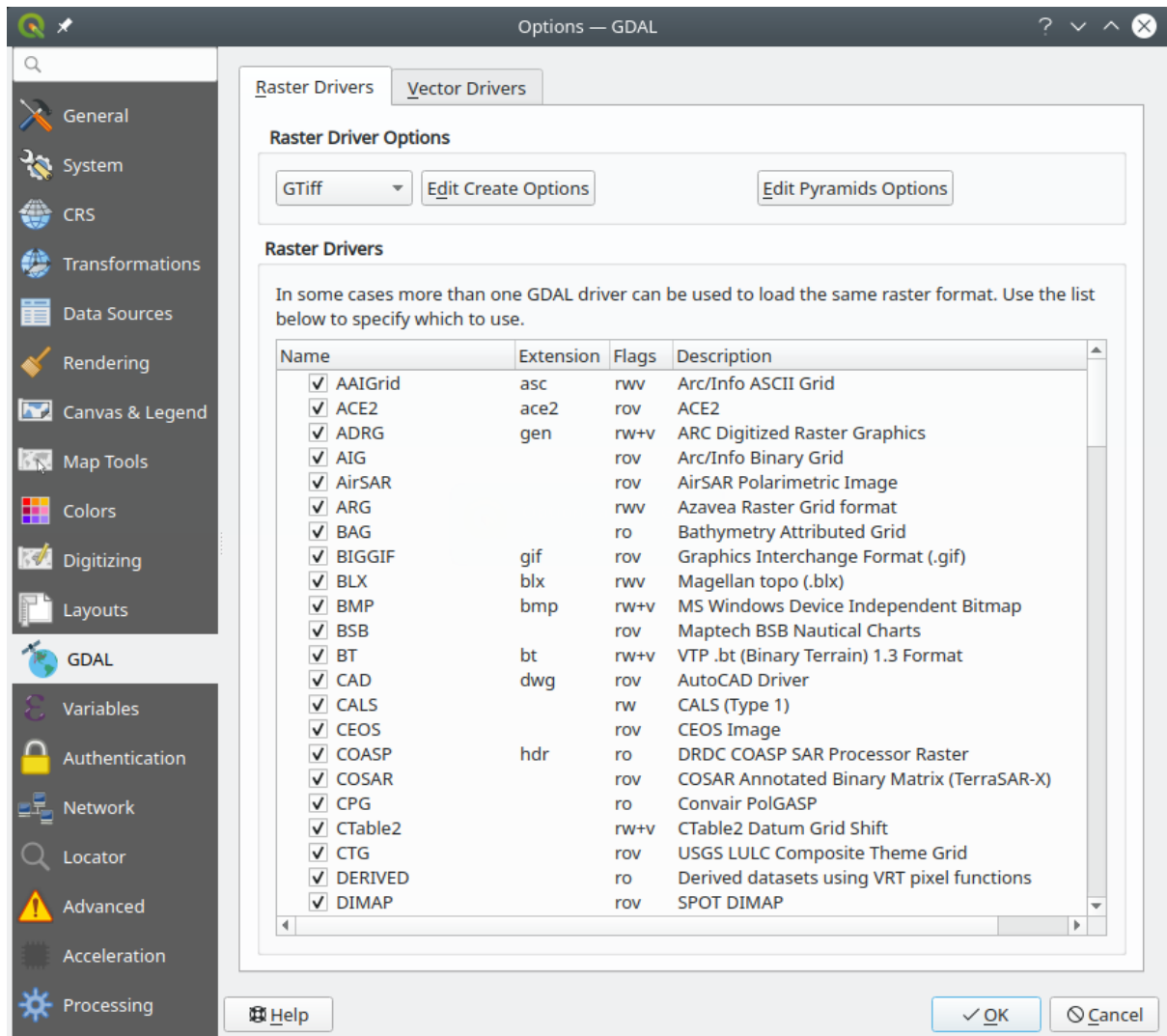




图 9.13: QGIS 中的 GDAL 设置 - 栅格驱动

小技巧: 双击允许读写访问的栅格驱动 (rw+ (v)) 打开编辑创建选项对话框进行自定义编辑。

9.1.13 变量设置

变量选项卡列出了全局级别的所有可用变量。

它还允许用户管理全局级别的变量。单击  按钮添加新的自定义全局级别变量；同样，从列表选择一个自定义全局级别变量，然后单击  按钮将其删除。

有关变量的更多信息，请参见 *Storing values in Variables* 章节。

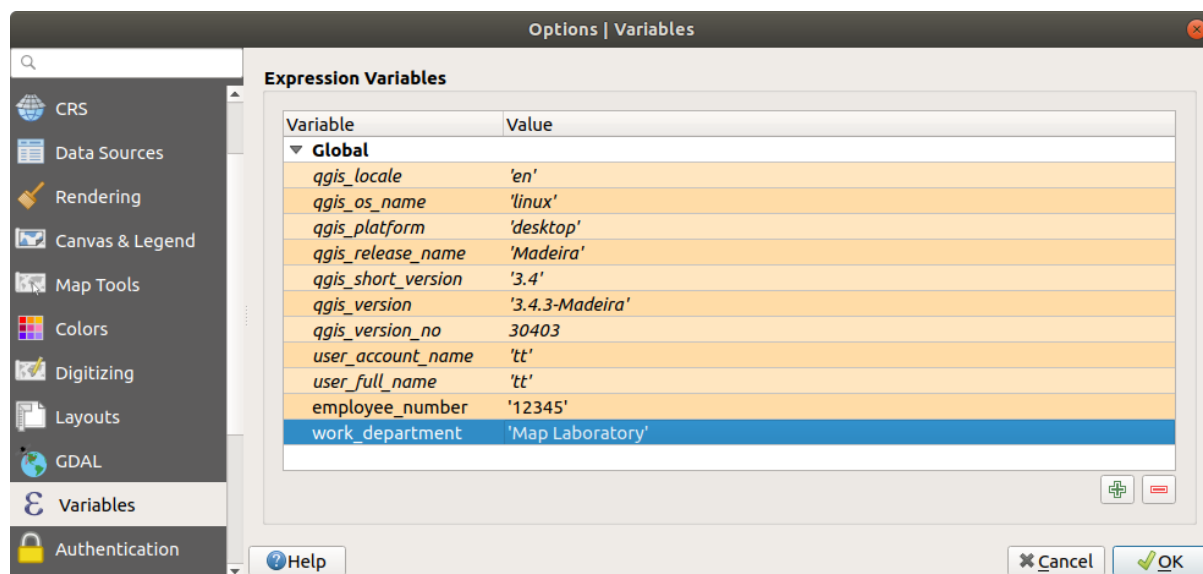


图 9.14: QGIS 中的变量设置

9.1.14 认证设置

在认证选项卡中，可以设置认证配置和管理 PKI 证书。更多信息，请参阅[认证系统](#)。

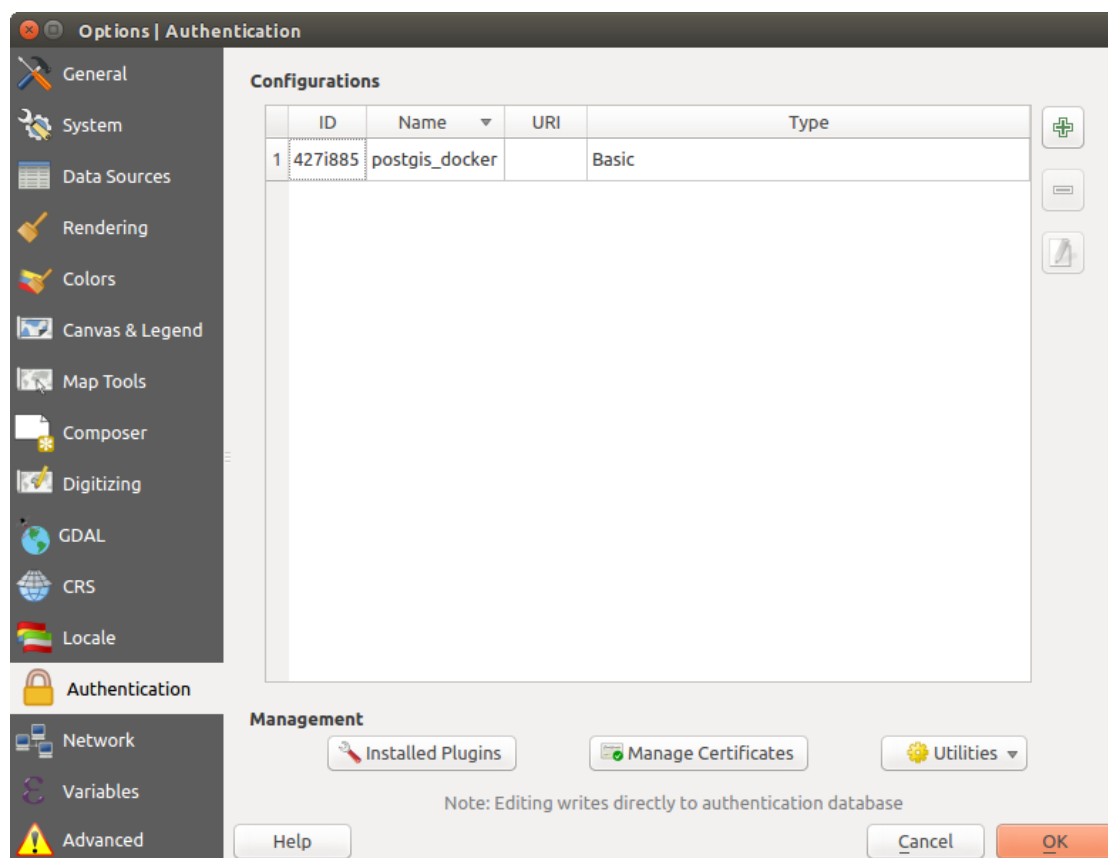


图 9.15: QGIS 中的认证设置

9.1.15 网络设置

通用

- 定义网络请求超时(毫秒) - 默认是 60000
- 定义 WMS 功能描述的默认有效期(小时) - 默认是 24
- 定义 WMS-C/WMTS 瓦片的默认过期时间(小时) - 默认是 24
- 定义瓦片或要素请求遇到错误时的最大重试次数
- 定义用户代理

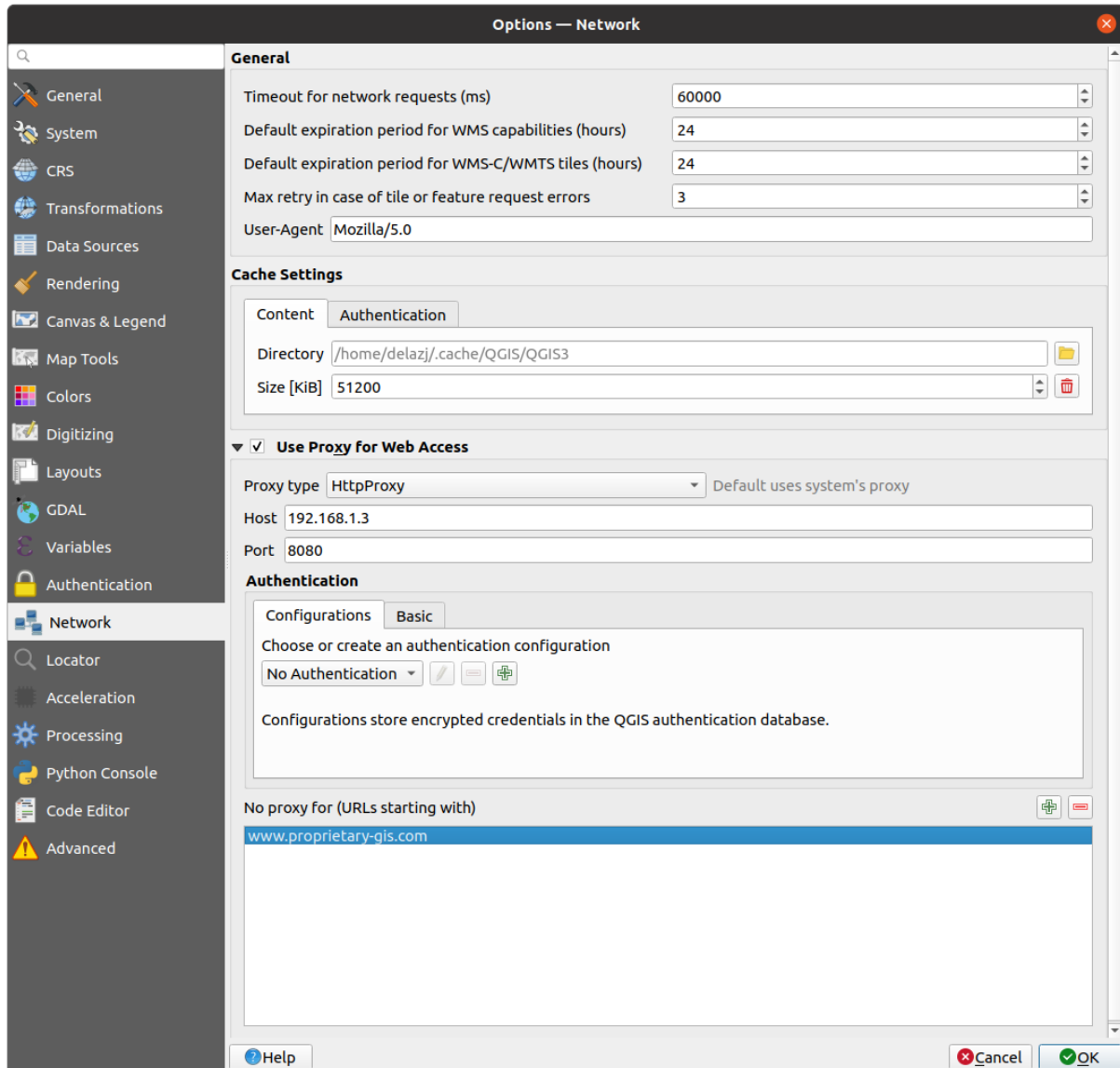



图 9.16: QGIS 中的代理设置

缓存设置

定义缓存的目录和大小,还提供了发生 SSL 错误时自动清除连接认证缓存(推荐)的工具。

Web 访问代理

- Web 访问使用代理

- 根据需要设置代理类型 ，并定义“主机”和“端口”。可用的代理类型包括：
 - 默认代理: 代理根据系统的代理确定
 - *Socks5Proxy*: 任何类型连接的通用代理，支持 TCP、UDP、绑定到端口 (传入连接) 和认证。
 - *HttpProxy*: 使用“CONNECT”命令实现，只支持传出 TCP 连接；支持认证。
 - *HttpCachingProxy*: 使用普通的 HTTP 命令实现，仅在 HTTP 请求的上下文中有用。
 - *FtpCachingProxy*: 使用 FTP 代理实现，仅在 FTP 请求的上下文中有用。

使用认证控件 设置代理的凭据。


排除部分 URL 可以添加到代理设置下方的文本框中 (请参阅 图 9.16)。如果目标 URL 以此文本框中列出的字符串之一开头，则不会使用代理。

如果需要有关不同代理设置的更多信息, 请参阅底层 QT 库文档手册: <https://doc.qt.io/qt-5.9/qnetworkproxy.html#ProxyType-enum>

小技巧: 使用代理

使用代理有时会很棘手，通过对上述代理类型进行“反复试验”来检查它们在您的实例中是否成功，是很必要的。

9.1.16 搜索设置

 The 搜索选项卡允许您配置搜索栏, 这是状态栏上的一个快速搜索控件, 可帮助您在应用程序中执行搜索。它提供了一些默认过滤 (带有前缀) 供使用:

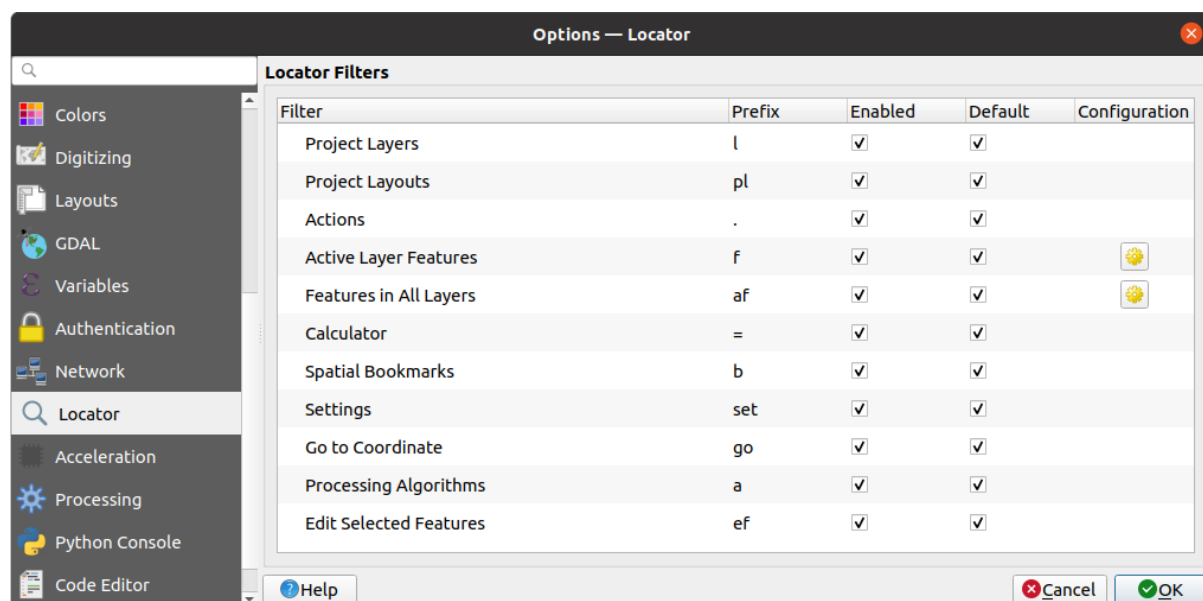




图 9.17: QGIS 中的搜索设置

- 工程图层 (l): 在 图层面板中查找并选择图层。
- 工程布局 (pl): 查找并打开打印布局。
- 动作 (.): 查找并执行 QGIS 动作; 动作可以是 QGIS 中的任何工具或菜单、打开面板...
- 活动图层要素 (f): 在当前活动图层的任何字段中搜索匹配的属性, 并缩放到选中的要素。按  配置最大结果数。

- 所有图层中的要素 (af): 在每个可搜索图层的显示名称中搜索匹配的属性，并缩放到选中的要素。按  可配置“每个图层最大结果数”和“最大结果数”。
- 计算器 (=): 允许计算任何 QGIS 表达式，如果有效，则提供将结果复制到剪贴板的选项。
- 空间书签 (b): 查找并缩放到书签范围。
- 设置 (set): 浏览并打开工程和应用程序范围内的属性对话框。
- 前往坐标 (go): 将地图画布平移到由逗号或空格分隔的 x 和 y 坐标对或格式化 URL (例如 OpenStreetMap、Leaflet、OpenLayer、谷歌地图等) 定义的位置。坐标应位于 WGS 84 (epsg:4326) 和/或地图画布 CRS 中。
- 处理算法 (a): 搜索并打开处理算法对话框。
- 编辑选中的要素 (ef): 提供快速访问，并在活动图层上运行兼容的 *modify-in-place* 处理算法。

在对话框中，您可以：

- 自定义过滤 前缀, 即用于触发过滤的关键词
- 设置是否 启用过滤: 可以在搜索中使用过滤，并在搜索栏菜单中提供快捷键
- 设置过滤是否为 默认: 不使用过滤的搜索只返回默认过滤类别的结果。
- 一些过滤提供了配置搜索结果数量的方法。

默认搜索过滤集可以通过插件进行扩展，例如用于 OSM Nominatim 搜索、直接数据库搜索、图层目录搜索等

9.1.17 高级设置

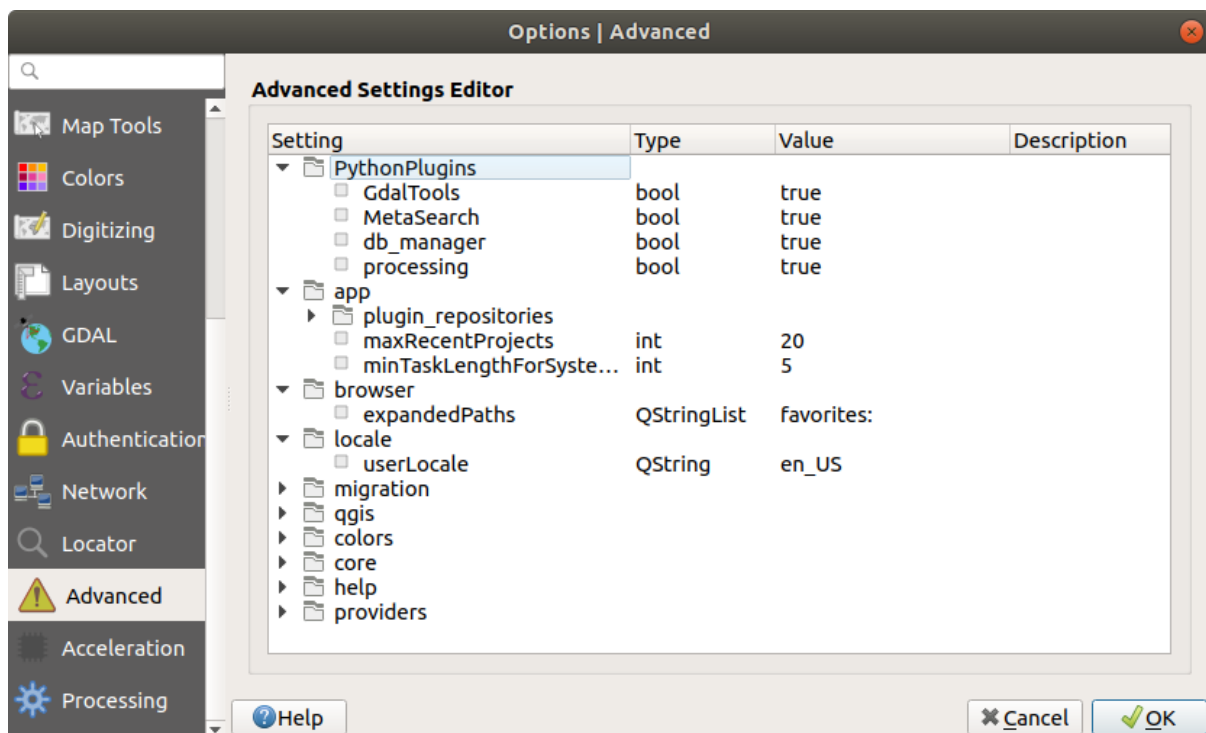


图 9.18: QGIS 中的高级设置选项卡

所有与 QGIS 相关的设置 (UI、工具、数据提供程序、处理配置、默认值和路径、插件选项、表达式、几何图形检查...) 都保存在当前用户配置目录下的 QGIS/QGIS3.ini 文件中。通过将此文件复制到其他设备，可以共享配置。

在 QGIS 中, 高级选项卡提供了通过高级设置编辑器管理这些设置的方法。在您保证要小心之后, 这个控件将被一个包含所有现有设置的树所填充, 您可以编辑它们的值。右键单击设置或组, 可以将其删除(要添加设置或组, 必须编辑 QGIS3.ini 文件)。所做更改会自动保存在 QGIS3.ini 文件中。

警告: 避免盲目使用高级选项卡设置

鉴于更改会自动应用, 因此修改此对话框中的项时一定要小心。在不知情的情况下进行更改可能会以各种方式破坏 QGIS 设置。

9.1.18 加速设置

OpenCL 加速设置。

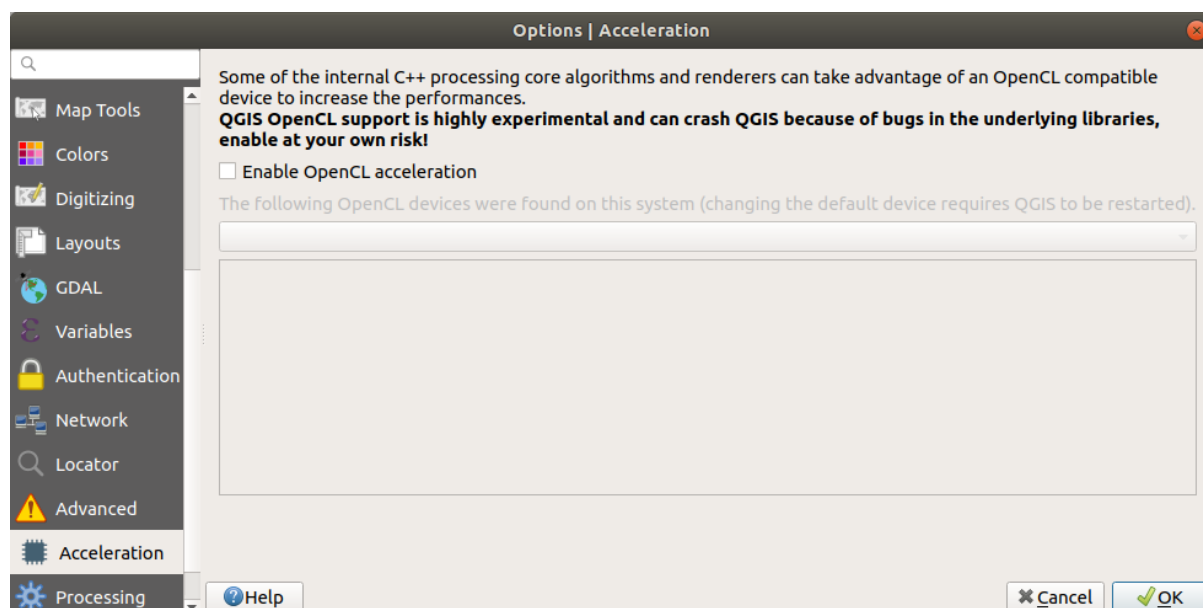



图 9.19: 加速选项卡

根据您的硬件和软件情况, 可能需要安装其他库来启用 OpenCL 加速。

9.1.19 数据处理设置

 数据处理选项卡为您提供 QGIS 处理框架中使用的工具和数据提供程序的常规设置。更多信息请访问 [QGIS 处理框架](#)。

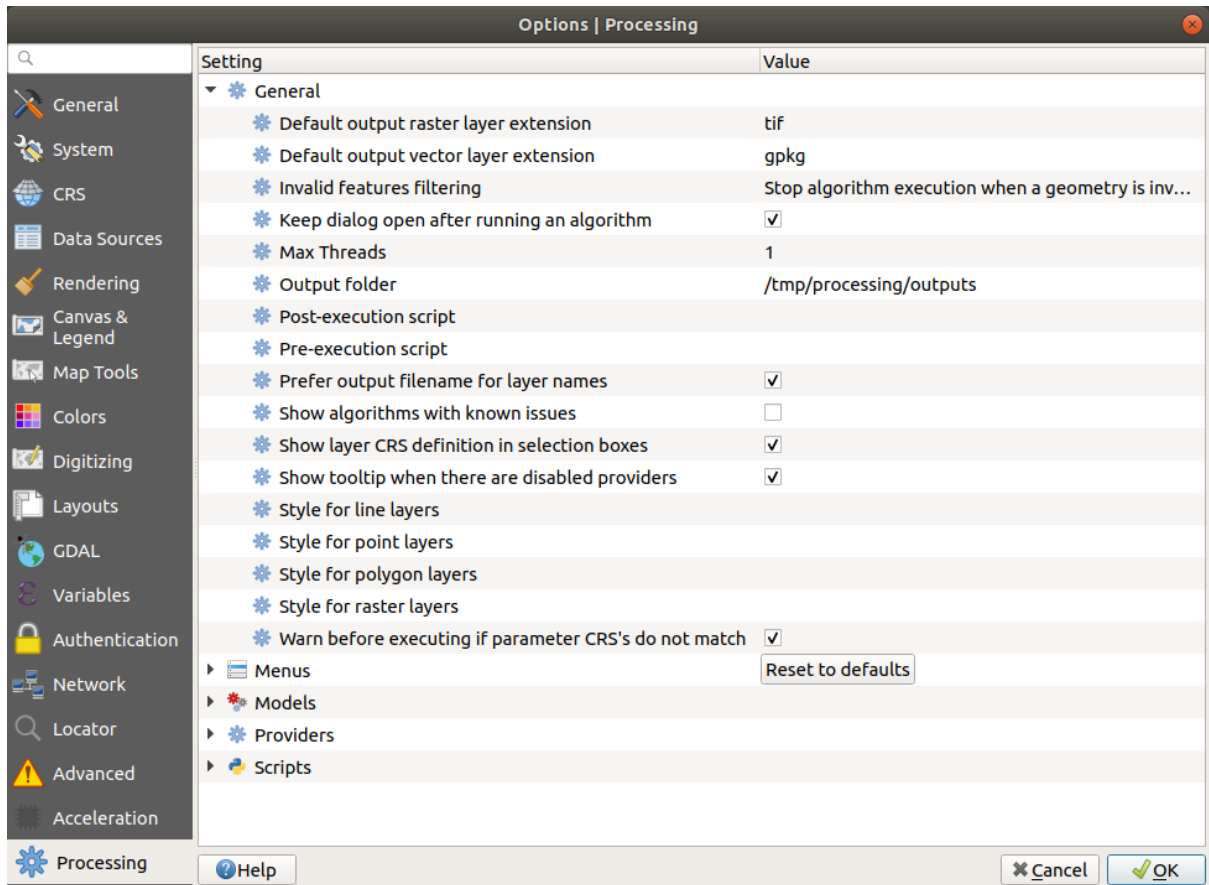




图 9.20: QGIS 中的数据处理设置选项卡

9.1.20 Python 控制台设置

 Python 控制台设置可帮助您管理和控制 Python 编辑器 (交互式控制台, 代码编辑器, 工程宏, 自定义表达式, ...) 的行为, 也可以使用以下  选项... 按钮进行访问:

- Python 控制台工具栏
- Python 控制台控件的上下文菜单
- 以及代码编辑器的上下文菜单。

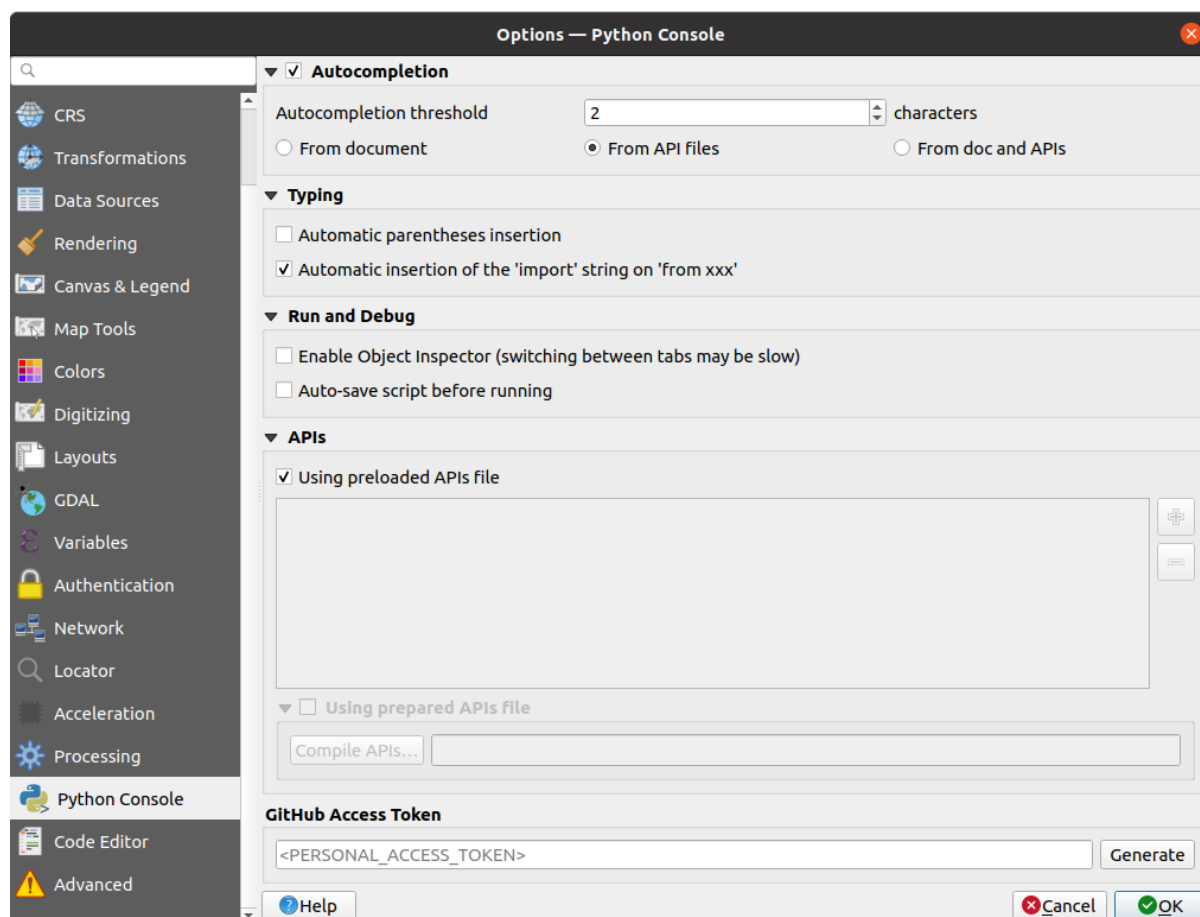


图 9.21: Python 控制台设置选项卡


您可以指定：

- 自动补全: 启用代码自动完成。您可以从当前文档、已安装的 API 文件或两者中获取自动补全阈值。
 - 自动补全阈值: 设置显示自动补全列表的阈值 (以字符为单位)
- 在 键盘输入 下面
 - 自动插入括号: 启用括号的自动关闭功能
 - 遇到 `'from xxx'` 字符串时自动插入 `'import'`: 在指定导入时允许插入 `'import'` 字符
- 在 运行和调试 下面
 - 启用对象检查器 (标签页间切换时可能会较慢): 启用对象检查器。
 - 运行之前自动保存的脚本: 执行时自动保存脚本。此操作将储存一个临时文件 (在临时系统目录中), 该文件将在运行后自动删除。

对于 API 您可以指定：

- 使用预加载的 API 文件: 您可以选择是否要使用预加载的 API 文件。如果未勾选此选项, 可以添加 API 文件, 还可以选择是否使用准备好的 API 文件 (请参见下一选项)。
- 使用预备的 API 文件: 如果勾选, 所选 `*.pap` 文件将用于代码完成。要生成预备的 API 文件, 必须至少加载一个 `*.api` 文件, 然后通过单击 `编译 API...` 按钮对其进行编译。
- 在 [GitHub](#) 访问令牌下, 您可以生成个人令牌, 允许您在 Python 代码编辑器中共享代码片段。更多信息, 请参阅 [GitHub 认证](#)

9.1.21 代码编辑器设置

在  代码编辑器选项卡，可以控制代码编辑器控件 (Python 交互式控制台和编辑器、表达式控件和函数编辑器等) 的外观和行为。

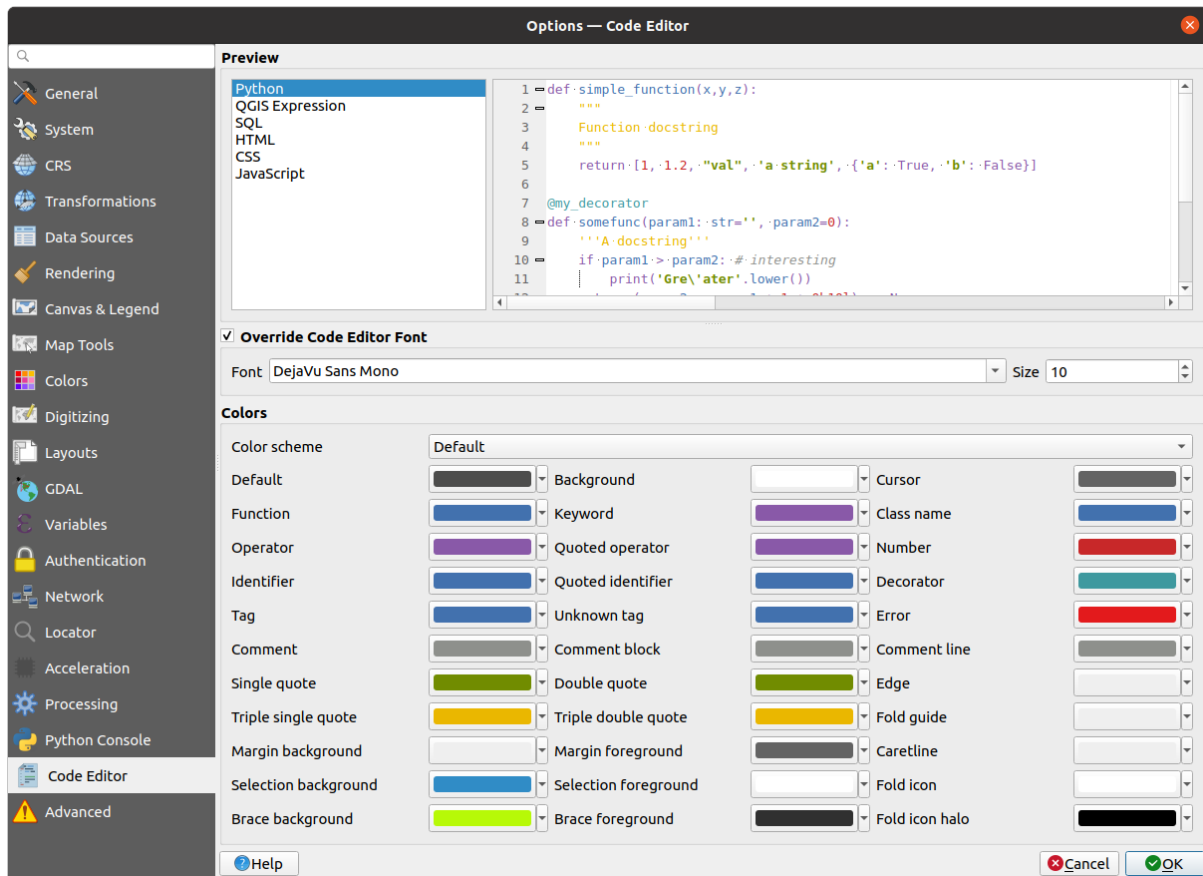


图 9.22: 代码编辑器设置选项卡

在对话框顶部，控件以各种编码语言 (Python, QGIS 表达式, HTML, SQL, JavaScript) 提供当前设置的实时预览。调整设置的个性化方法如下：

- 勾选 更换代码编辑器默认字体可以修改默认字体和字号。
- 在颜色组下，您可以：
 - 选择颜色方案：预定义方案的设置为默认，曝光暗和曝光亮。一旦修改颜色，就会触发自定义方案，并且可以通过选择预定义方案进行重置。
 - 在编写代码时更改每个元素的颜色，例如用于注释、引号、函数、背景等的颜色。

9.2 使用用户配置文件




设置 ► 用户配置菜单提供设置和访问用户配置文件的功能。用户配置文件是统一的应用程序配置，允许在单个文件夹中储存：

- 所有全局设置, 包括语言环境、投影、认证设置、调色板、快捷键等
- GUI 配置和定制
- 为基准变换而安装的网格文件和其他 Proj 辅助文件
- 已安装的插件 及其配置
- 工程模板和保存工程的历史记录及其图像预览
- 数据处理设置、日志、脚本和模型。

默认情况下，QGIS 安装包含一个名为 default 的单用户配置文件夹。但您可以创建任意数量的用户配置文件：

1. 单击 新建配置... 条目。
2. 系统将提示您提供配置文件名，并在 ~/<UserProfiles>/ 下创建同名文件夹，其中：

- ~ 表示主目录，在 Windows 上的主目录通常类似于 C:\Users\<user>。
- 和 <UserProfiles> 表示主配置文件夹，即：

-  .local/share/QGIS/QGIS3/profiles/
-  AppData\Roaming\QGIS\QGIS3\profiles\
-  Library/Application Support/QGIS/QGIS3/profiles/

可以使用 打开当前配置文件夹从 QGIS 打开用户配置文件夹。

3. 使用原始的配置启动 QGIS 新实例，之后您可以设置自定义配置。

如果您的 QGIS 设置中有多个配置文件，当前配置文件的名称将显示在应用程序标题栏的方括号之间。

由于每位用户配置文件都包含独立的设置、插件和历史记录，因此非常适合不同的 workflow、演示、同一台计算机的用户或测试设置等。您可以通过 设置 ► 用户配置菜单，或从命令行 使用特定用户的配置文件运行 QGIS。

除非更改，否则最后关闭 QGIS 会话的配置文件将用于之后的 QGIS 会话。

小技巧：在新用户配置文件下运行 QGIS，以检查 Bug 的持久性

当您在 QGIS 遇到某些功能有奇怪的行为时，请创建一个新的用户配置文件，然后再次运行这些命令。有时，Bug 与当前用户配置文件中的某些遗留问题有关，创建一个新用户配置文件可能会进行修复，因为现在使用新的 (原始) 配置文件重新启动 QGIS。

9.3 工程属性

在 工程 ► 工程属性下的工程属性窗口中，可以设置工程特定的选项。特定于工程的选项将重写上述选项对话框中的等效选项。

9.3.1 通用属性

在通用选项卡的通用设置中允许您：

- 查看工程文件的位置
- 设置工程主目录的文件夹(可在浏览器的工程主目录项中找到)。路径可以是相对于工程文件的文件夹 (type it in) 或绝对路径。工程主目录可用于存储对工程有用的数据和其他内容。
- 在工程文件路径旁边指定工程的标题
- 选择要素被选中时要使用的颜色
- 选择背景色：用于地图画布的颜色
- 设置工程中图层的路径应保存为绝对(完整)还是相对于工程文件位置。当图层和工程文件都可以移动或共享，或从不同平台的计算机访问工程时，您可能更喜欢相对路径。
- 选择在工程渲染为地图瓦片时避免伪影。请注意，勾选此选项可能会导致性能下降。

计算面积和距离是 GIS 中的常见需求。但是，这些值实际与基础投影设置有关。计量框架允许您控制这些参数，您可以选择：

- 椭球体, 距离和面积的计算完全基于此，可以是：
 - **无/平面**: 这种情况下，返回的值是笛卡尔测量值。
 - **自定义**: 您需要设置长半轴和短半轴的值。
 - 或预定义列表中的现有椭球体 (Clarke 1866, Clarke 1880 IGN, New International 1967, WGS 84...)
- 长度和周长的 **距离计量单位**和 **面积计量单位**。这些设置默认为 QGIS 选项中设置的单位，但随后会重写当前工程中的单位，用于：
 - 属性表字段更新栏
 - 字段计算器的计算
 - 识别工具得到的长度、周长和面积值
 - 测量对话框显示的默认单位

坐标和方位角显示允许您选择和自定义用于在状态栏显示鼠标坐标的方位角格式和单位格式，以及通过识别工具显示的衍生坐标。

最后，可以设置工程预定义比例列表，该列表将重写全局预定义比例。

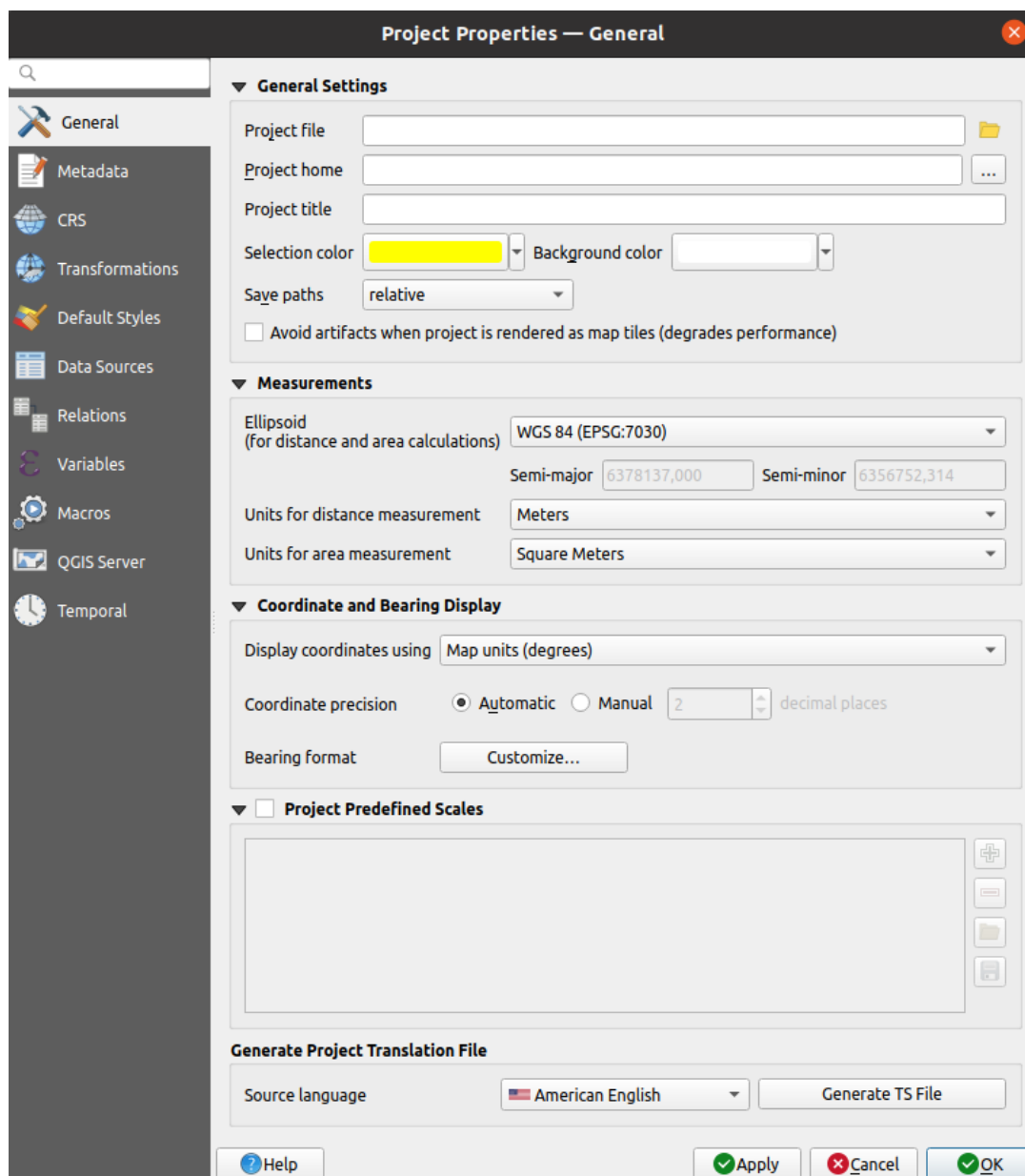


图 9.23: 工程属性对话框的通用选项卡

9.3.2 元数据属性

元数据选项卡允许定义详细的元数据，包括(除其他外)：作者、创建日期、语言、摘要、类别、关键字、联系方式、链接、历史。还有一个验证功能，可以检查特定字段是否已填充，但无论如何，这并不强制执行。有关详细信息，请参阅[矢量图层元数据属性](#)。


9.3.3 CRS 属性

注解: 有关 QGIS 如何处理工程投影的更多信息, 请阅读 *Working with Projections* 专用章节。

 CRS 选项卡可帮助您设置要在此工程中使用的坐标参照系。它可以是:

- 无 CRS (或未知/非地球投影): 图层是根据其原始坐标绘制的
- 或者现有坐标参照系, 可以是地理, 投影或用户自定义。添加到工程中的图层会动态转换到此 CRS, 以便叠加它们, 而不考虑其原始 CRS。

9.3.4 变换属性

 变换选项卡通过配置要应用于当前工程的基准面变换首选项, 帮助您控制图层重投影设置。照例, 这些设置会覆盖任何相应的全局设置。更多信息, 请参阅 *Datum Transformations* 。

9.3.5 默认样式属性

默认样式选项卡, 允许您控制在没有现存 .qml 样式定义时, 如何在工程中绘制新图层。您可以:

- 根据图层几何图形类型以及默认颜色渐变, 设置要应用的默认符号 (标记, 线, 填充)
- 将默认不透明度应用于新建图层
- 给符号指派随机颜色, 修改符号填充颜色, 从而避免对所有图层进行相同的渲染。

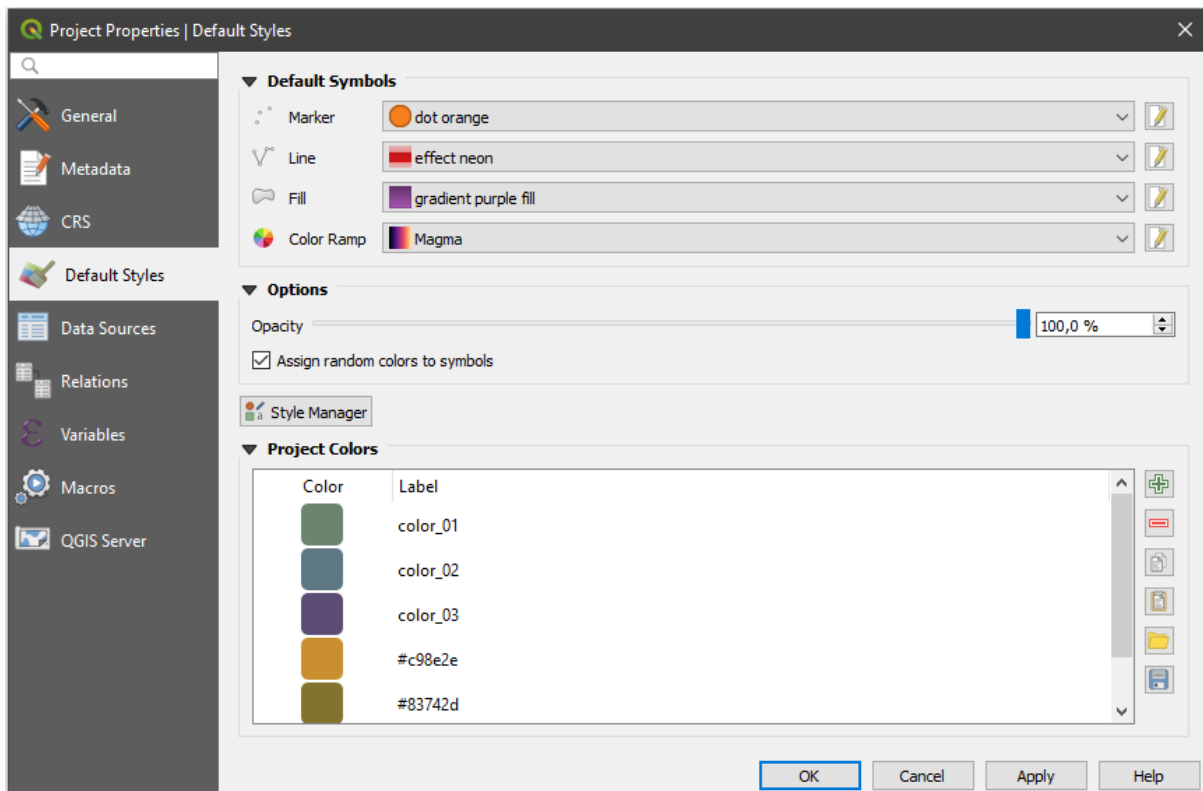








图 9.24: 默认样式选项卡

使用  样式管理器按钮, 还可以快速访问样式管理器对话框, 并配置符号和颜色渐变。

还有附加部件，您可以在其中为正在运行的工程定义指定的颜色。与全局颜色一样，您可以：

-  添加或  移除颜色
-  复制或  粘贴颜色
-  导入或  导出颜色集从/到 .qml 文件。

双击列表中的颜色，在 [选择颜色](#) 对话框中对其进行调整或替换，也可以通过双击 [标注列](#) 对其进行重命名。

这些颜色被认定为 [工程颜色](#)，并作为 [颜色控件](#) 的一部分列出。

小技巧：使用工程颜色快速分配和更新颜色控件

工程颜色可以使用其标注来引用，并且所使用的颜色控件与之绑定。这意味着您不必为许多属性设置相同的颜色而重复替换，为了避免繁琐的更新，您可以：

1. 将颜色定义为工程颜色
2. 单击要设置颜色属性旁边的 [数据定义重写控件](#)
3. 将鼠标悬停在 [颜色](#) 菜单上并选择工程颜色。然后，属性被指定表达式 `project_color('color_label')`，颜色控件反映该颜色。
4. 根据需要重复步骤 2 和 3
5. 每次更新工程颜色，所做的更改就会反映在使用工程颜色的任何地方。

9.3.6 数据源属性

In the *Data Sources* tab, you can:

- *Automatically create transaction groups where possible*: When this mode is turned on, all layers from the same database are synchronised in their edit state, i.e. when one layer is put into edit state, all are, when one layer is committed or one layer is rolled back, so are the others. Also, instead of buffering edit changes locally, they are directly sent to a transaction in the database which gets committed when the user clicks save layer. Note that you can (de)activate this option only if no layer is being edited in the project.
- *Evaluate default values on provider side*: When adding new features in a PostgreSQL table, fields with default value constraint are evaluated and populated at the form opening, and not at the commit moment. This means that instead of an expression like `nextval('serial')`, the field in the *Add Feature* form will display expected value (e.g., 25).
- *Trust project when data source has no metadata*: To speed up project loading by skipping data checks. Useful in QGIS Server context or in projects with huge database views/materialized views. The extent of layers will be read from the QGIS project file (instead of data sources) and when using the PostgreSQL provider the primary key unicity will not be checked for views and materialized views.
- Configure the *Layers Capabilities*, i.e.:
 - Set (or disable) which layers are `identifiable`, i.e. will respond to the *identify tool*. By default, layers are set queryable.
 - Set whether a layer should appear as `read-only`, meaning that it can not be edited by the user, regardless of the data provider's capabilities. Although this is a weak protection, it remains a quick and handy configuration to avoid end-users modifying data when working with file-based layers.
 - Define which layers are `searchable`, i.e. could be queried using the *locator widget*. By default, layers are set searchable.
 - Define which layers are defined as `required`. Checked layers in this list are protected from inadvertent removal from the project.

The *Layers Capabilities* table provides some convenient tools to:

- Select multiple cells and press *Toggle Selection* to have them change their checkbox state;
- *Show spatial layers only*, filtering out non-spatial layers from the layers list;
- *Filter layers...* and quickly find a particular layer to configure.

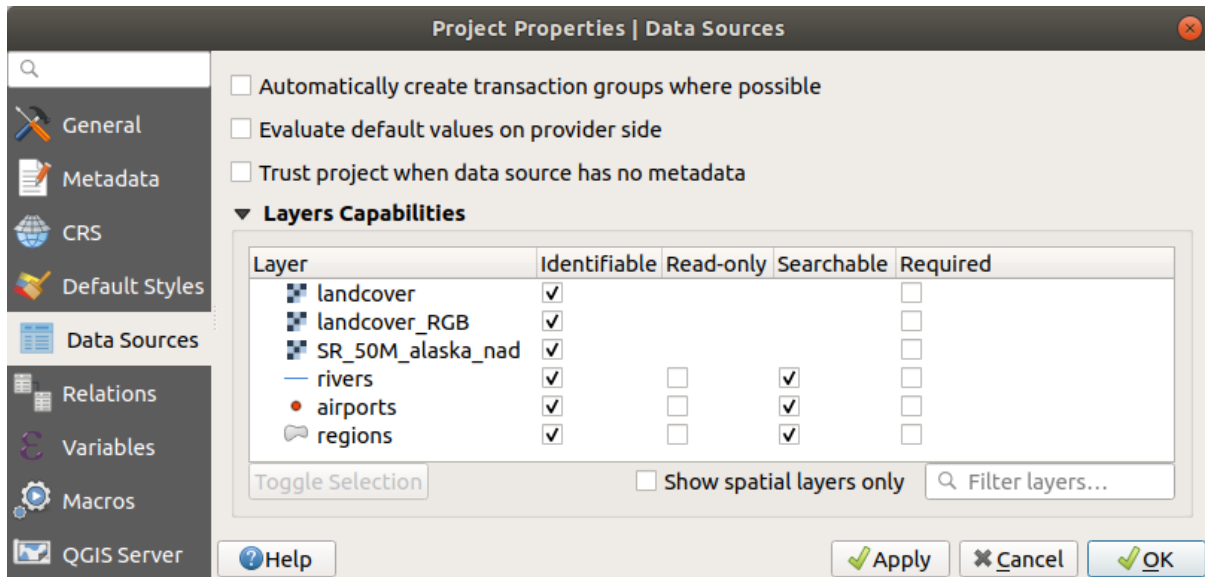


图 9.25: Data Sources tab

9.3.7 Relations Properties

The *Relations* tab is used to define 1:n relations. The relations are defined in the project properties dialog. Once relations exist for a layer, a new user interface element in the form view (e.g. when identifying a feature and opening its form) will list the related entities. This provides a powerful way to express e.g. the inspection history on a length of pipeline or road segment. You can find out more about 1:n relations support in Section *Creating one or many to many relations*.

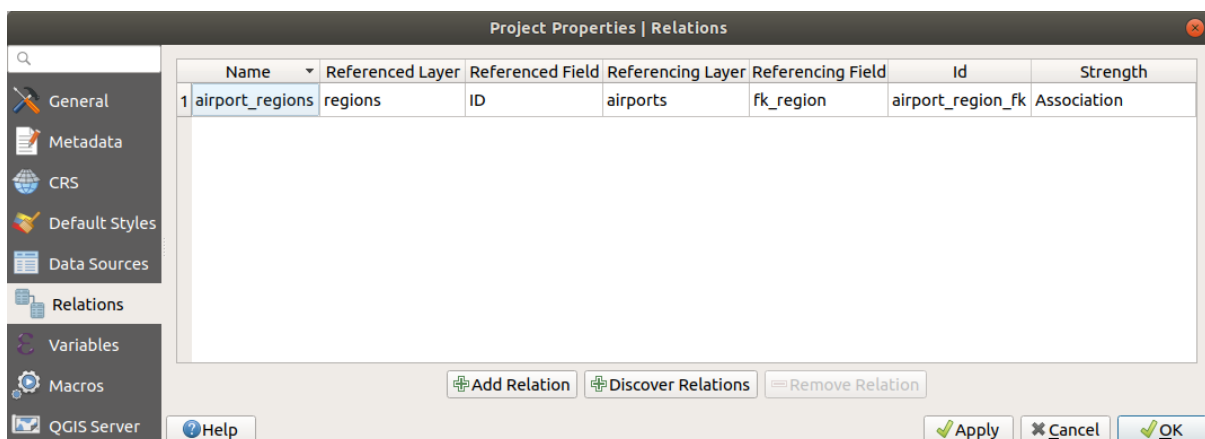




图 9.26: Relations tab

9.3.8 Variables Properties

The *Variables* tab lists all the variables available at the project's level (which includes all global variables). Besides, it also allows the user to manage project-level variables. Click the  button to add a new custom project-level variable. Likewise, select a custom project-level variable from the list and click the  button to remove it. More information on variables usage in the General Tools *Storing values in Variables* section.

9.3.9 Macros Properties

The *Macros* tab is used to edit Python macros for projects. Currently, only three macros are available: `openProject()`, `saveProject()` and `closeProject()`.

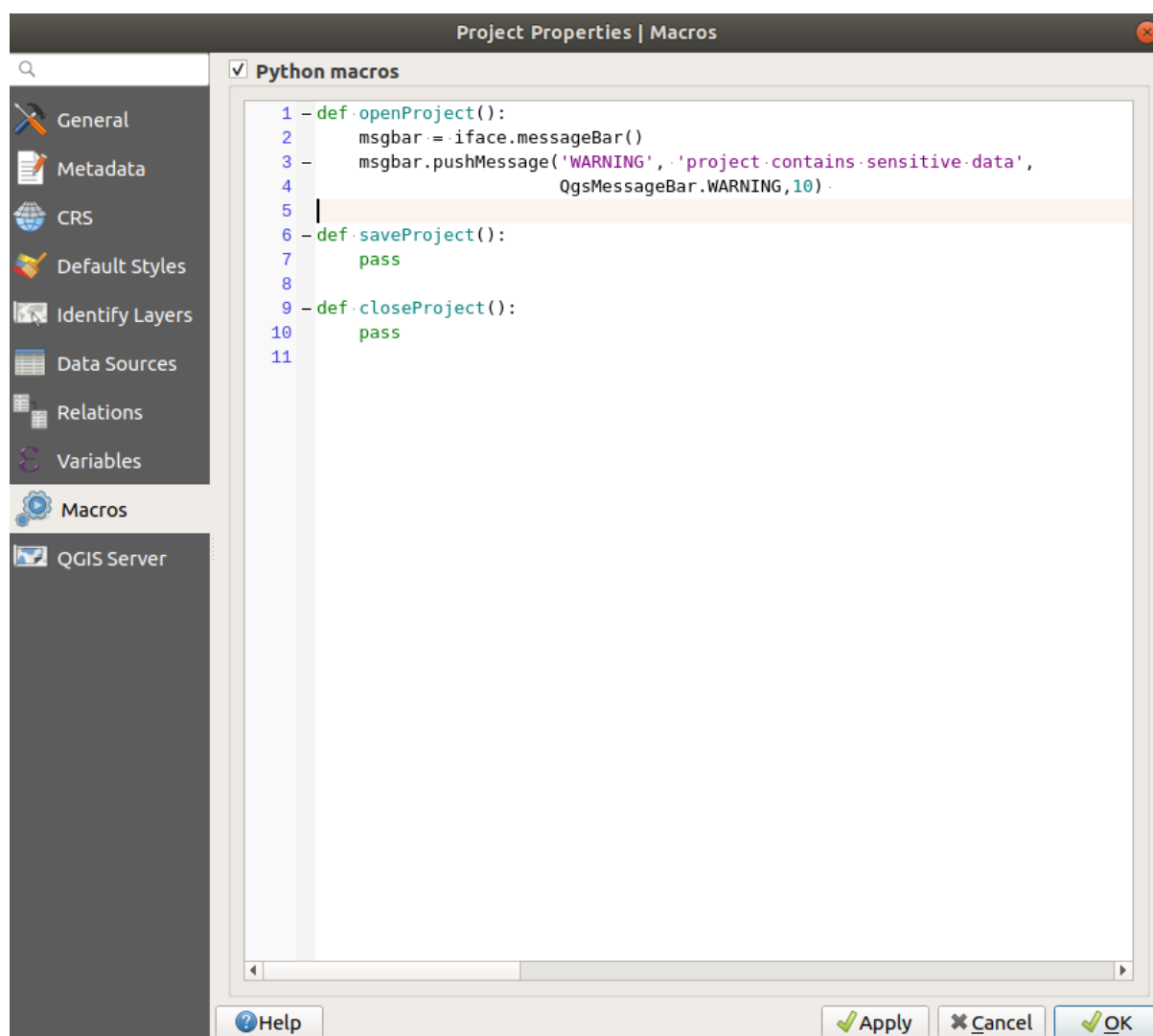


图 9.27: Macro settings in QGIS

9.3.10 QGIS Server Properties

The tab *QGIS Server* allows you to configure your project in order to publish it online. Here you can define information about the QGIS Server WMS and WFS capabilities, extent and CRS restrictions. More information available in section *Creating wms from project* and subsequent.

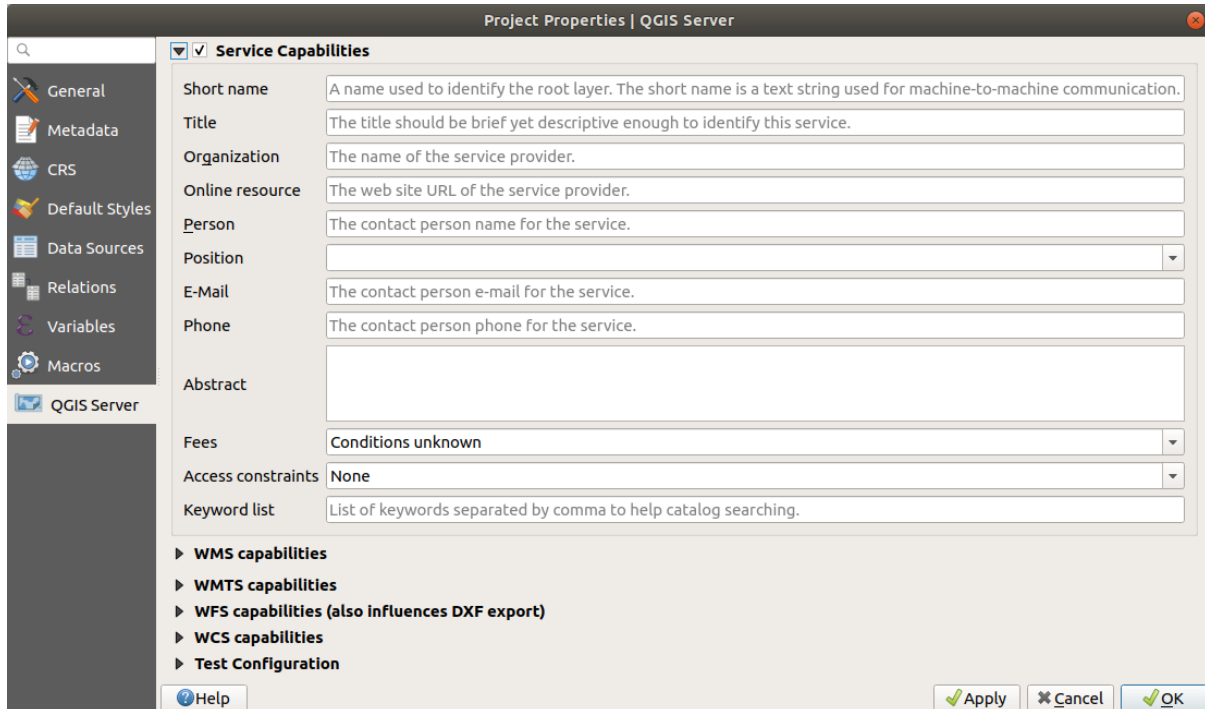


图 9.28: QGIS Server settings tab

9.3.11 Temporal Properties

The tab *Temporal* is used to set the temporal range of your project, either by using manual input or by calculating it from the current project temporal layers.

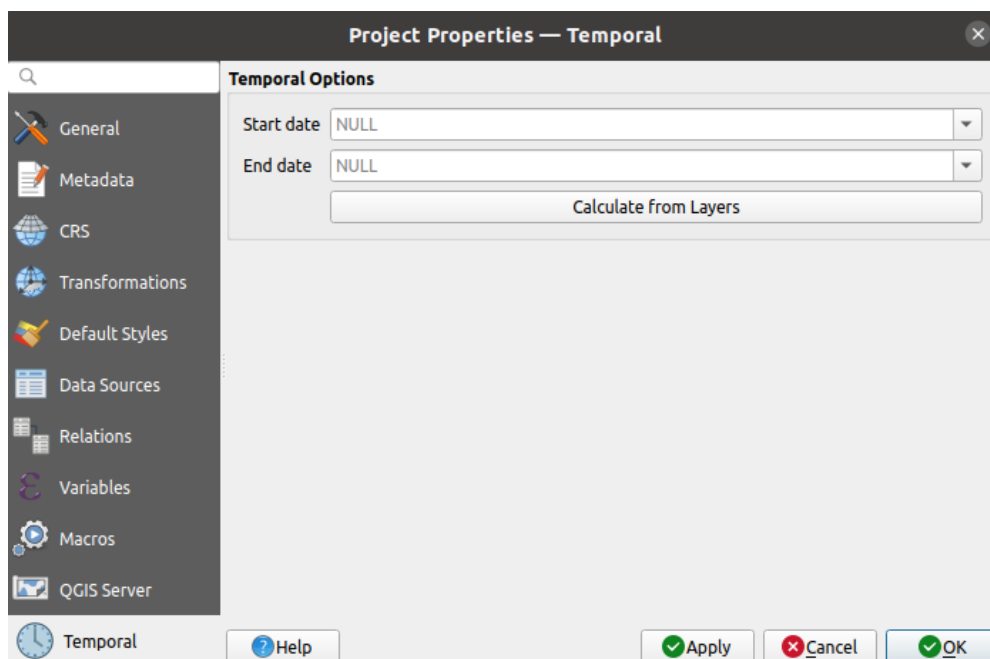


图 9.29: QGIS Temporal tab

9.4 Customization

The customization dialog lets you (de)activate almost every element in the QGIS user interface. This can be very useful if you want to provide your end-users with a 'light' version of QGIS, containing only the icons, menus or panels they need.

注解: Before your changes are applied, you need to restart QGIS.

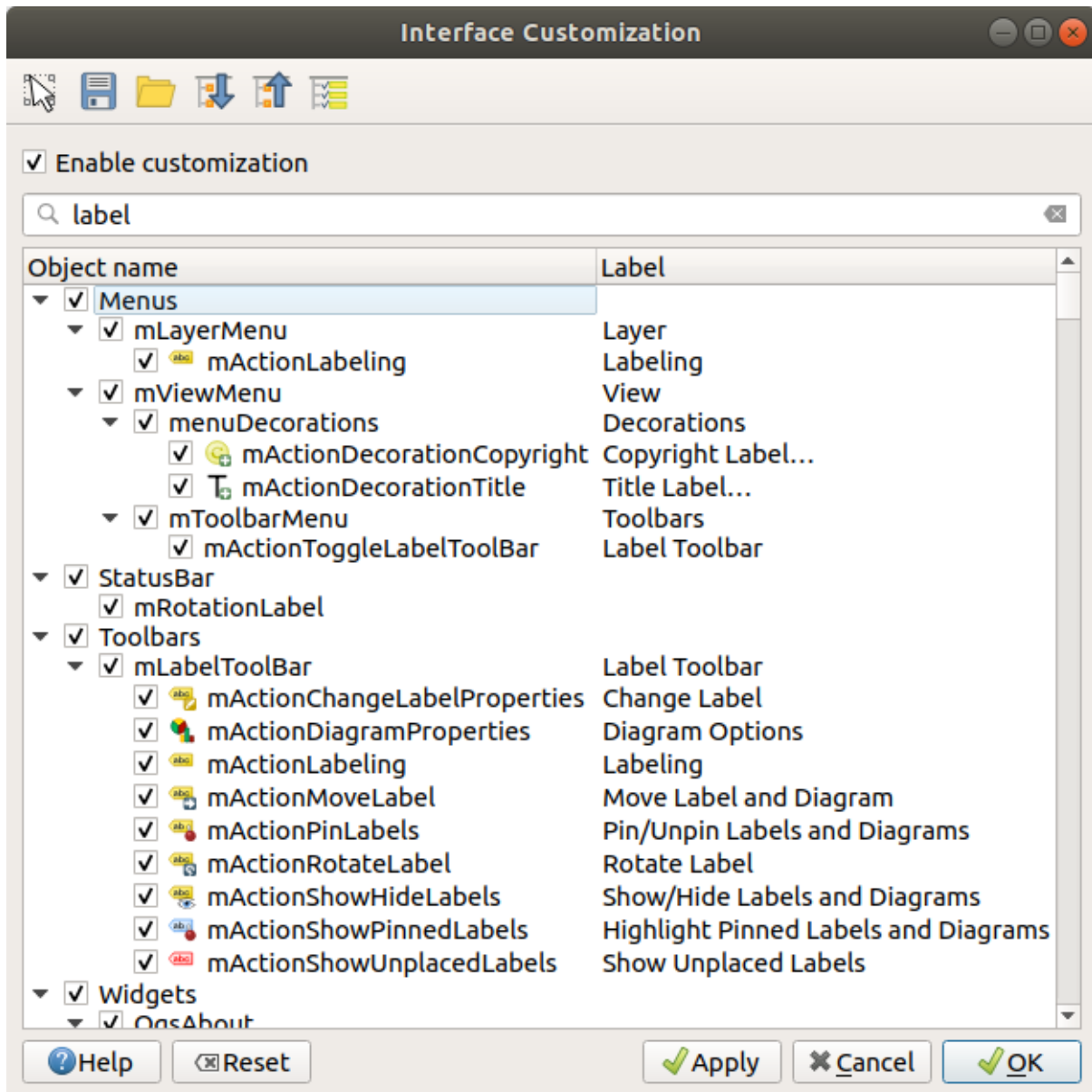



图 9.30: The Customization dialog



Ticking the *Enable customization* checkbox is the first step on the way to QGIS customization. This enables the toolbar and the widget panel from which you can uncheck and thus disable some GUI items.

The configurable item can be:

- a **Menu** or some of its sub-menus from the 菜单栏
- a whole **Panel** (see 面板和工具栏)
- the **Status bar** described in 状态栏 or some of its items
- a **Toolbar**: the whole bar or some of its icons
- or any **widget** from any dialog in QGIS: label, button, combobox...



With  *Switch to catching widgets in main application*, you can click on an item in QGIS interface that you want to be hidden and QGIS automatically unchecks the corresponding entry in the Customization dialog. You can also use the *Search* box to find items by their name or label.

Once you setup your configuration, click *Apply* or *OK* to validate your changes. This configuration becomes the one used by default by QGIS at the next startup.

The modifications can also be saved in a `.ini` file using  Save To File button. This is a handy way to share a common QGIS interface among multiple users. Just click on  Load from File from the destination computer in order to import the `.ini` file. You can also run *command line tools* and save various setups for different use cases as well.


小技巧: Easily restore predefined QGIS

The initial QGIS GUI configuration can be restored by one of the methods below:

- unchecking  *Enable customization* option in the Customization dialog or click the  Check All button
- pressing the *Reset* button in the **Settings** frame under *Settings* ► *Options* menu, *System* tab
- launching QGIS at a command prompt with the following command line `qgis --nocustomization`
- setting to `false` the value of *UI* ► *Customization* ► *Enabled* variable under *Settings* ► *Options* menu, *Advanced* tab (see the *warning*).

In most cases, you need to restart QGIS in order to have the change applied.

9.5 Keyboard shortcuts

QGIS provides default keyboard shortcuts for many features. You can find them in section 菜单栏. Additionally, the menu option *Settings* ►  *Keyboard Shortcuts...* allows you to change the default keyboard shortcuts and add new ones to QGIS features.

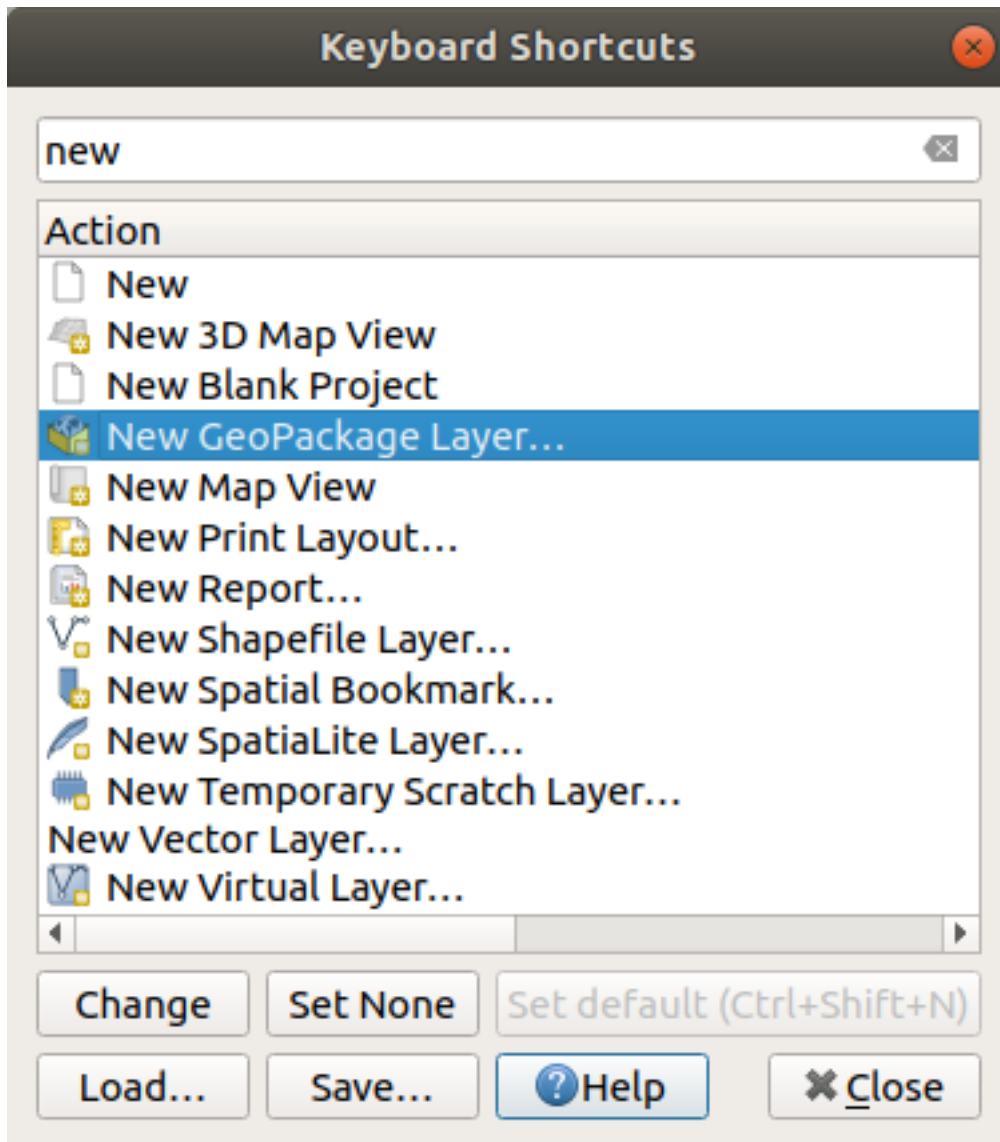


图 9.31: Define shortcut options

Configuration is very simple. Use the search box at the top of the dialog to find a particular action, select it from the list and click on :

- *Change* and press the new combination you want to assign as new shortcut
- *Set None* to clear any assigned shortcut
- or *Set Default* to backup the shortcut to its original and default value.

Proceed as above for any other tools you wish to customize. Once you have finished your configuration, simply *Close* the dialog to have your changes applied. You can also *Save* the changes as an *.XML* file and *Load* them into another QGIS installation.

9.6 Running QGIS with advanced settings

9.6.1 Command line and environment variables

We've seen that *launching QGIS* is done as for any application on your OS. QGIS provides command line options for more advanced use cases (in some cases you can use an environment variable instead of the command line option). To get a list of the options, enter `qgis --help` on the command line, which returns:

```

QGIS is a user friendly Open Source Geographic Information System.
Usage: /usr/bin/qgis.bin [OPTION] [FILE]
  OPTION:
    [--version]          display version information and exit
    [--snapshot filename] emit snapshot of loaded datasets to given file
    [--width width]      width of snapshot to emit
    [--height height]    height of snapshot to emit
    [--lang language]    use language for interface text (changes existing
↳ override)
    [--project projectfile] load the given QGIS project
    [--extent xmin,ymin,xmax,ymax] set initial map extent
    [--nologo]           hide splash screen
    [--noverversioncheck] don't check for new version of QGIS at startup
    [--nopugins]         don't restore plugins on startup
    [--nocustomization]  don't apply GUI customization
    [--customizationfile path] use the given ini file as GUI customization
    [--globalsettingsfile path] use the given ini file as Global Settings
↳ (defaults)
    [--authdbdirectory path] use the given directory for authentication
↳ database
    [--code path]        run the given python file on load
    [--defaulttui]       start by resetting user ui settings to default
    [--hide-browser]     hide the browser widget
    [--dxf-export filename.dxf] emit dxf output of loaded datasets to
↳ given file
    [--dxf-extent xmin,ymin,xmax,ymax] set extent to export to dxf
    [--dxf-symbolology-mode none|symbolayer|feature] symbology mode for dxf
↳ output
    [--dxf-scale-denom scale] scale for dxf output
    [--dxf-encoding encoding] encoding to use for dxf output
    [--dxf-map-theme maptheme] map theme to use for dxf output
    [--take-screenshots output_path] take screen shots for the user
↳ documentation
    [--screenshots-categories categories] specify the categories of
↳ screenshot to be used (see QgsAppScreenShots::Categories).
    [--profile name]     load a named profile from the user's profiles
↳ folder.
    [--profiles-path path] path to store user profile folders. Will create
↳ profiles inside a {path}\profiles folder
    [--version-migration] force the settings migration from older version if
↳ found
    [--openclprogramfolder] path to the folder containing the sources
↳ for OpenCL programs.
    [--help]             this text
    [--]                 treat all following arguments as FILES

FILE:
  Files specified on the command line can include rasters,
  vectors, and QGIS project files (.qgs and .qgz):
  1. Rasters - supported formats include GeoTiff, DEM
     and others supported by GDAL
  2. Vectors - supported formats include ESRI Shapefiles
     and others supported by OGR and PostgreSQL layers using

```

(下页继续)

(续上页)

```
the PostGIS extension
```

小技巧: Example Using command line arguments

You can start QGIS by specifying one or more data files on the command line. For example, assuming you are in the `qgis_sample_data` directory, you could start QGIS with a vector layer and a raster file set to load on startup using the following command: `qgis ./raster/landcover.img ./gml/lakes.gml`

--version

This option returns QGIS version information.

--snapshot

This option allows you to create a snapshot in PNG format from the current view. This comes in handy when you have many projects and want to generate snapshots from your data, or when you need to create snapshots of the same project with updated data.

Currently, it generates a PNG file with 800x600 pixels. The size can be adjusted using the `--width` and `--height` arguments. The filename can be added after `--snapshot`. For example:

```
qgis --snapshot my_image.png --width 1000 --height 600 --project my_project.qgs
```

--width

This option returns the width of the snapshot to be emitted (used with `--snapshot`).

--height

This option returns the height of the snapshot to be emitted (used with `--snapshot`).

--lang

Based on your locale, QGIS selects the correct localization. If you would like to change your language, you can specify a language code. For example, `qgis --lang it` starts QGIS in Italian localization.

--project

Starting QGIS with an existing project file is also possible. Just add the command line option `--project` followed by your project name and QGIS will open with all layers in the given file loaded.

--extent

To start with a specific map extent use this option. You need to add the bounding box of your extent in the following order separated by a comma:

```
--extent xmin,ymin,xmax,ymax
```

This option probably makes more sense when paired with the `--project` option to open a specific project at the desired extent.

--nologo

This option hides the splash screen when you start QGIS.

--noversioncheck

Skip searching for a new version of QGIS at startup.

--noplugins

If you have trouble at start-up with plugins, you can avoid loading them at start-up with this option. They will still be available from the Plugins Manager afterwards.

--nocustomization

Using this option, any existing *GUI customization* will not be applied at startup. This means that any hidden buttons, menu items, toolbars, and so on, will show up on QGIS start up. This is not a permanent change. The customization will be applied again if QGIS is launched without this option.

This option is useful for temporarily allowing access to tools that have been removed by customization.

--customizationfile

Using this option, you can define a UI customization file, that will be used at startup.

--globalsettingsfile

Using this option, you can specify the path for a Global Settings file (`.ini`), also known as the Default Settings. The settings in the specified file replace the original inline default ones, but the user profiles' settings will be set on top of those. The default global settings is located in `your_QGIS_PKG_path/resources/qgis_global_settings.ini`.

Presently, there's no way to specify a file to write settings to; therefore, you can create a copy of an original settings file, rename, and adapt it.

Setting the `qgis_global_setting.ini` file path to a network shared folder, allows a system administrator to change global settings and defaults in several machines by only editing one file.

The equivalent environment variable is `QGIS_GLOBAL_SETTINGS_FILE`.

--authdbdirectory

This option is similar to `--globalsettingsfile`, but defines the path to the directory where the authentication database will be stored and loaded.

--code

This option can be used to run a given python file directly after QGIS has started.

For example, when you have a python file named `load_alaska.py` with following content:

```
from qgis.utils import iface
raster_file = "/home/gisadmin/Documents/qgis_sample_data/raster/landcover.img"
layer_name = "Alaska"
iface.addRasterLayer(raster_file, layer_name)
```

Assuming you are in the directory where the file `load_alaska.py` is located, you can start QGIS, load the raster file `landcover.img` and give the layer the name 'Alaska' using the following command:

```
qgis --code load_alaska.py
```

--defaultui

On load, **permanently resets** the user interface (UI) to the default settings. This option will restore the panels and toolbars visibility, position, and size. Unless it's changed again, the default UI settings will be used in the following sessions.

Notice that this option doesn't have any effect on *GUI customization*. Items hidden by GUI customization (e.g. the status bar) will remain hidden even using the `--defaultui` option. See also the `--nocustomization` option.

--hide-browser

On load, hides the *Browser* panel from the user interface. The panel can be enabled by right-clicking a space in the toolbars or using the *View* ► *Panels* (*Settings* ► *Panels* in  Linux KDE).

Unless it's enabled again, the Browser panel will remain hidden in the following sessions.

--dxf-*

These options can be used to export a QGIS project into a DXF file. Several options are available:

- `--dxf-export`: the DXF filename into which to export the layers;
- `--dxf-extent`: the extent of the final DXF file;
- `--dxf-symbology-mode`: several values can be used here: `none` (no symbology), `symbollayer` (Symbol layer symbology), `feature` (feature symbology);
- `--dxf-scale-denom`: the scale denominator of the symbology;
- `--dxf-encoding`: the file encoding;
- `--dxf-map-theme`: choose a *map theme* from the layer tree configuration.

`--take-screenshots`

Takes screenshots for the user documentation. Can be used together with `--screenshots-categories` to filter which categories/sections of the documentation screenshots should be created (see `QgsAppScreenshots::Categories`).

`--profile`

Loads QGIS using a specific profile from the user's profile folder. Unless changed, the selected profile will be used in the following QGIS sessions.

`--profiles-path`

With this option, you can choose a path to load and save the profiles (user settings). It creates profiles inside a `{path}\profiles` folder, which includes settings, installed plugins, processing models and scripts, and so on.

This option allows you to, for instance, carry all your plugins and settings in a flash drive, or, for example, share the settings between different computers using a file sharing service.

The equivalent environment variable is `QGIS_CUSTOM_CONFIG_PATH`.

`--version-migration`

If settings from an older version are found (e.g., the `.qgis2` folder from QGIS 2.18), this option will import them into the default QGIS profile.

`--openclprogramfolder`

Using this option, you can specify an alternative path for your OpenCL programs. This is useful for developers while testing new versions of the programs without needing to replace the existing ones.

The equivalent environment variable is `QGIS_OPENCL_PROGRAM_FOLDER`.

9.6.2 Deploying QGIS within an organization

If you need to deploy QGIS within an organization with a custom configuration file, first you need to copy/paste the content of the default settings file located in `your_QGIS_PKG_path/resources/qgis_global_settings.ini`. This file already contains some default sections identified by a block starting with `[]`. We recommend that you keep these default values and add your own sections at the bottom of the file. If a section is duplicated in the file, QGIS will take the last one from top to bottom.

You can change `allowVersionCheck=false` to disable the QGIS version check.

If you do not want to display the migration window after a fresh install, you need the following section:

```
[migration]
fileVersion=2
settings=true
```

If you want to add a custom variable in the global scope:

```
[variables]
organisation="Your organization"
```

To discover the possibilities of the settings INI file, we suggest that you set the config you would like in QGIS Desktop and then search for it in your INI file located in your profile using a text editor. A lot of settings can be set using the INI file such as WMS/WMTS, PostGIS connections, proxy settings, maptips...

Finally, you need to set the environment variable `QGIS_GLOBAL_SETTINGS_FILE` to the path of your customized file.

In addition, you can also deploy files such as Python macros, color palettes, layout templates, project templates... either in the QGIS system directory or in the QGIS user profile.

- Layout templates must be deployed in the `composer_templates` directory.
- Project templates must be deployed in the `project_templates` directory.
- Custom Python macros must be deployed in the `python` directory.

A Coordinate Reference System, or CRS, is a method of associating numerical coordinates with a position on the surface of the Earth. QGIS has support for approximately 7,000 standard CRSs, each with different use cases, pros and cons! Choosing an appropriate reference system for your QGIS projects and data can be a complex task, but fortunately QGIS helps guide you through this choice, and makes working with different CRSs as transparent and accurate as possible.

10.1 Overview of Projection Support

QGIS has support for approximately 7,000 known CRSs. These standard CRSs are based on those defined by the European Petroleum Search Group (EPSG) and the Institut Geographique National de France (IGNF), and are made available in QGIS through the underlying "Proj" projection library. Commonly, these standard projections are identified through use of an authority:code combination, where the authority is an organisation name such as "EPSG" or "IGNF", and the code is a unique number associated with a specific CRS. For instance, the common WGS 84 latitude/longitude CRS is known by the identifier `EPSG:4326`, and the web mapping standard CRS is `EPSG:3857`.

Custom, user-created CRSs are stored in a user CRS database. See section *Custom Coordinate Reference System* for information on managing your custom coordinate reference systems.

10.2 Layer Coordinate Reference Systems

In order to correctly project data into a specific target CRS, either your data must contain information about its coordinate reference system or you will need to manually assign the correct CRS to the layer. For PostGIS layers, QGIS uses the spatial reference identifier that was specified when that PostGIS layer was created. For data supported by OGR or GDAL, QGIS relies on the presence of a recognized means of specifying the CRS. For instance, for the Shapefile format this is a file containing an ESRI Well-Known Text (WKT) representation of the layer's CRS. This projection file has the same base name as the `.shp` file and a `.prj` extension. For example, `alaska.shp` would have a corresponding projection file named `alaska.prj`.

Whenever a layer is loaded into QGIS, QGIS attempts to automatically determine the correct CRS for that layer. In some cases this is not possible, e.g. when a layer has been provided without retaining this information. You can configure QGIS behavior whenever it cannot automatically determine the correct CRS for a layer:

1. Open *Settings* ►  *Options...* ► *CRS*

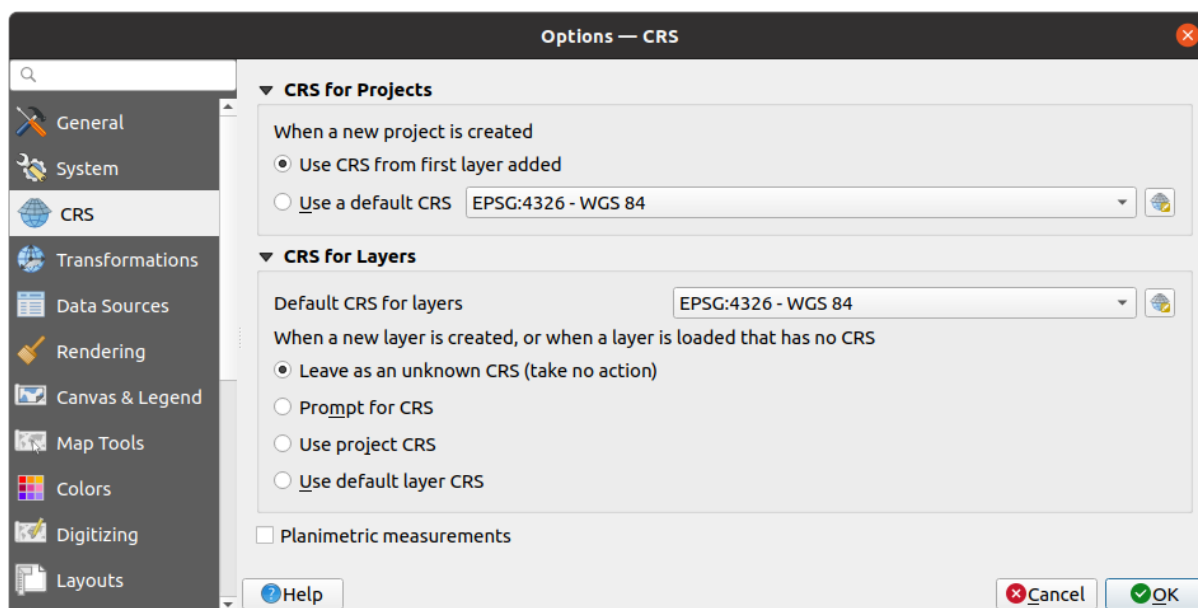



图 10.1: The CRS tab in the QGIS Options Dialog

2. Under the *CRS for layers* group, set the action to do *when a new layer is created, or when a layer is loaded that has no CRS*. One of:

- *Leave as unknown CRS (take no action)*: there will be no prompt to select a CRS when a layer without CRS is loaded, deferring CRS choice to a later time. Convenient when loading a lot of layers at once. Such layers will be identifiable in the *Layers* panel by the  icon next to them. They'll also be un-referenced, with coordinates from the layer treated as purely numerical, non-earth values, i.e. the same behavior as all layers get when *a project is set to have no CRS*.
- *Prompt for CRS*: it will prompt you to manually select the CRS. Selecting the correct choice is crucial, as a wrong choice will place your layer in the wrong position on the Earth's surface! Sometimes, accompanying metadata will describe the correct CRS for a layer, in other cases you will need to contact the original author of the data to determine the correct CRS to use.
- *Use project CRS*
- *Use default layer CRS*, as set in the *Default CRS for layers* combobox above.

小技巧: To assign the same CRS to multiple layers that have no crs or have a wrong one in one operation:

1. Select the layers in the *Layers* panel
2. Press `Ctrl+Shift+C`. You could also right-click over one of the selected layers or go to *Layer ► Set CRS of layer(s)*
3. Find and select the right CRS to use
4. And press *OK*. You can confirm that it has been set correctly in the *Source* tab of the layers' properties dialog.

Note that changing the CRS in this setting does not alter the underlying data source in any way, rather it just changes how QGIS interprets the raw coordinates from the layer in the current QGIS project.

10.3 Project Coordinate Reference Systems

Every project in QGIS also has an associated Coordinate Reference System. The project CRS determines how data is projected from its underlying raw coordinates to the flat map rendered within your QGIS map canvas.

QGIS supports "on the fly" CRS transformation for both raster and vector data. This means that regardless of the underlying CRS of particular map layers in your project, they will always be automatically transformed into the common CRS defined for your project. Behind the scenes, QGIS transparently reprojects all layers contained within your project into the project's CRS, so that they will all be rendered in the correct position with respect to each other!

It is important to make an appropriate choice of CRS for your QGIS projects. Choosing an inappropriate CRS can cause your maps to look distorted, and poorly reflect the real-world relative sizes and positions of features. Usually, while working in smaller geographic areas, there will be a number of standard CRSs used within a particular country or administrative area. It's important to research which CRSs are appropriate or standard choices for the area you are mapping, and ensure that your QGIS project follows these standards.

By default, QGIS starts each new project using a global default projection. This default CRS is EPSG:4326 (also known as "WGS 84"), and it is a global latitude/longitude based reference system. This default CRS can be changed via the *CRS for New Projects* setting in the *CRS* tab under *Settings* ► *Options...* (see 图 10.1). There is an option to automatically set the project's CRS to match the CRS of the first layer loaded into a new project, or alternatively you can select a different default CRS to use for all newly created projects. This choice will be saved for use in subsequent QGIS sessions.

The project CRS can also be set through the *CRS* tab of the *Project* ► *Properties...* dialog. It will also be shown in the lower-right of the QGIS status bar.

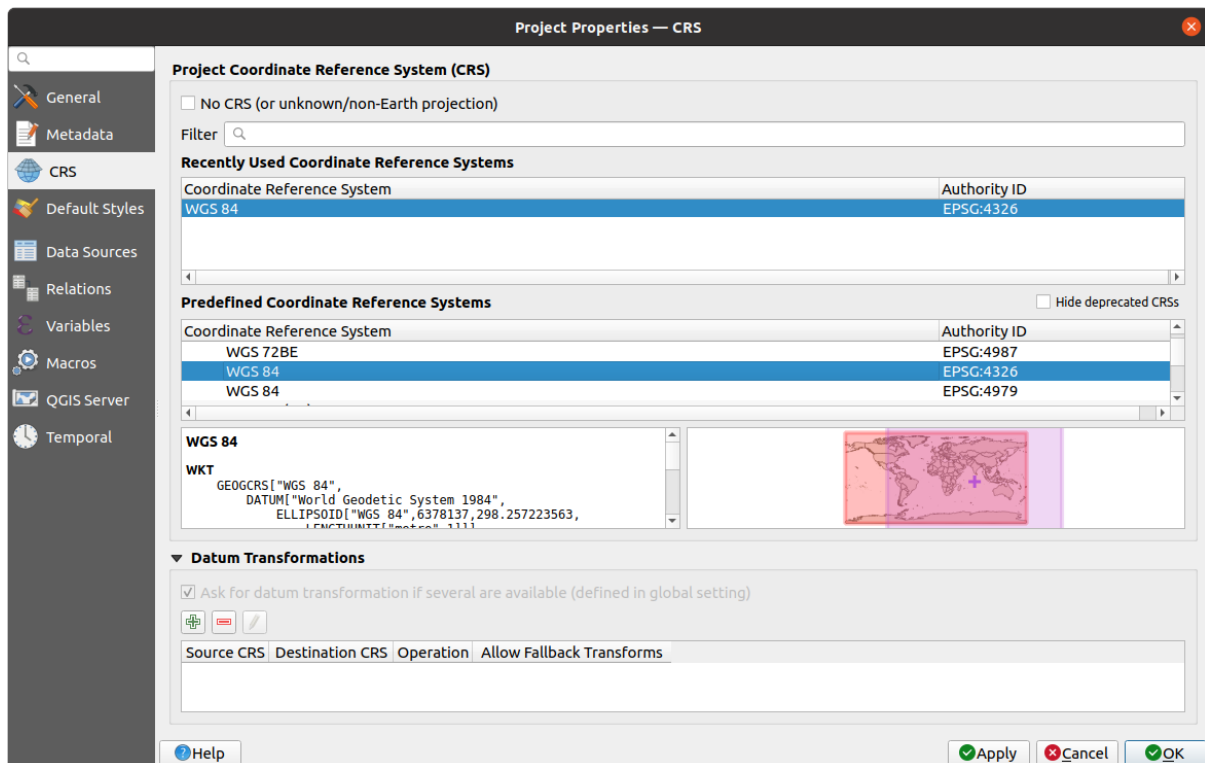


图 10.2: Project Properties Dialog

Available options are:

- No CRS (or unknown/non-Earth projection)*: Checking this setting will disable ALL projection handling within the QGIS project, causing all layers and map coordinates to be treated as simple 2D Cartesian coordinates, with no relation to positions on the Earth's surface. It can be used to guess a layer CRS (based on

its raw coordinates or when using QGIS for non earth uses like role-playing game maps, building mapping or microscopic stuff. In this case:

- No reprojection is done while rendering the layers: features are just drawn using their raw coordinates.
 - The ellipsoid is locked out and forced to `None/Planimetric`.
 - The distance and area units, and the coordinate display are locked out and forced to "unknown units"; all measurements are done in unknown map units, and no conversion is possible.
- or an existing coordinate reference system that can be *geographic*, *projected* or *user-defined*. A preview of the CRS extent on earth is displayed to help you select the appropriate one. Layers added to the project are translated on-the-fly to this CRS in order to overlay them regardless their original CRS. Use of units and ellipsoid setting are available and make sense and you can perform calculations accordingly.

Whenever you select a new CRS for your QGIS project, the measurement units will automatically be changed in the *General* tab of the *Project properties* dialog (*Project ► Properties...*) to match the selected CRS. For instance, some CRSs define their coordinates in feet instead of meters, so setting your QGIS project to one of these CRSs will also set your project to measure using feet by default.

小技巧: Setting the project CRS from a layer

You can assign a CRS to the project using a layer CRS:

1. In the *Layers* panel, right-click on the layer you want to pick the CRS
2. Select *Set project CRS from Layer*.

The project's CRS is redefined using the layer's CRS. Map canvas extent, coordinates display are updated accordingly and all the layers in the project are on-the-fly translated to the new project CRS.


10.4 Coordinate Reference System Selector

This dialog helps you assign a Coordinate Reference System to a project or a layer, provided a set of projection databases. Items in the dialog are:

- **Filter:** If you know the EPSG code, the identifier, or the name for a Coordinate Reference System, you can use the search feature to find it. Enter the EPSG code, the identifier or the name.
- **Recently used coordinate reference systems:** If you have certain CRSs that you frequently use in your everyday GIS work, these will be displayed in this list. Click on one of these items to select the associated CRS.
- **Coordinate reference systems of the world:** This is a list of all CRSs supported by QGIS, including Geographic, Projected and Custom coordinate reference systems. To define a CRS, select it from the list by expanding the appropriate node and selecting the CRS. The active CRS is preselected.
- **PROJ text:** This is the CRS string used by the PROJ projection engine. This text is read-only and provided for informational purposes.

The CRS selector also shows a rough preview of the geographic area for which a selected CRS is valid for use. Many CRSs are designed only for use in small geographic areas, and you should not use these outside of the area they were designed for. The preview map shades an approximate area of use whenever a CRS is selected from the list. In addition, this preview map also shows an indicator of the current main canvas map extent.

10.5 Custom Coordinate Reference System

If QGIS does not provide the coordinate reference system you need, you can define a custom CRS. To define a CRS, select  *Custom CRS...* from the *Settings* menu. Custom CRSs are stored in your QGIS user database. In addition to your custom CRSs, this database also contains your spatial bookmarks and other custom data.


Defining a custom CRS in QGIS requires a good understanding of the PROJ projection library. To begin, refer to "Cartographic Projection Procedures for the UNIX Environment - A User's Manual" by Gerald I. Evenden, U.S. Geological Survey Open-File Report 90-284, 1990 (available at <https://pubs.usgs.gov/of/1990/of90-284/ofr90-284.pdf>).

This manual describes the use of `proj` and related command line utilities. The cartographic parameters used with `proj` are described in the user manual and are the same as those used by QGIS.

The *Custom Coordinate Reference System Definition* dialog requires only two parameters to define a user CRS:

1. A descriptive name
2. The cartographic parameters in PROJ or WKT format

To create a new CRS:

1. Click the  *Add new CRS* button
2. Enter a descriptive name
3. Select the format: it can be *Proj String* or *WKT*
4. Add the *CRS Parameters*.

注解: Prefer storing the CRS definition in WKT format

Although both `Proj String` and `WKT` formats are supported, it's highly recommended to store projection definitions in the `WKT` format. Therefore, if the available definition is in the `proj` format, select that format, enter the parameters and then switch to `WKT` format. QGIS will convert the definition to the `WKT` format that you can later save.

5. Click *Validate* to test whether the CRS definition is an acceptable projection definition.

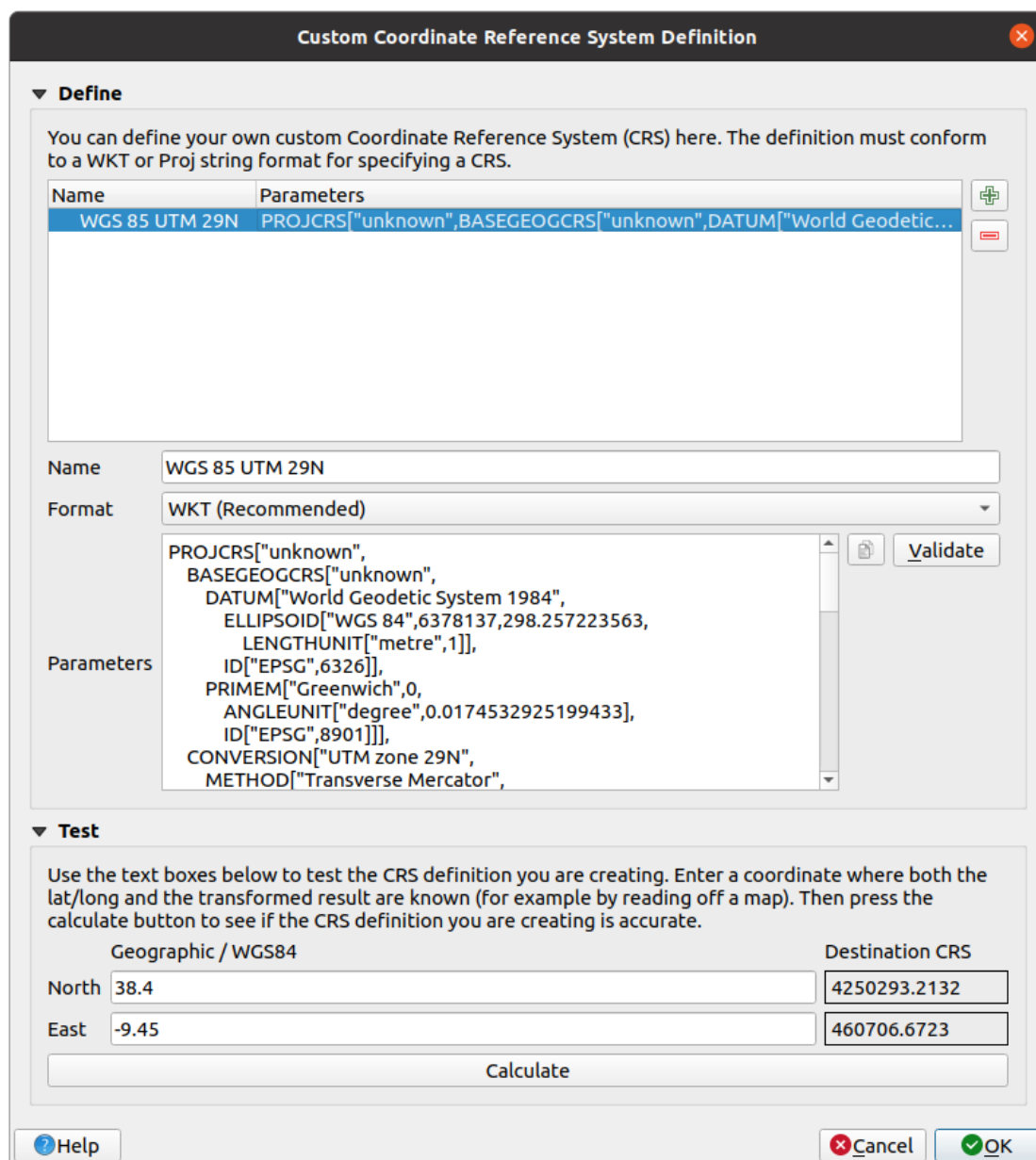


图 10.3: Custom CRS Dialog

You can test your CRS parameters to see if they give sane results. To do this, enter known WGS 84 latitude and longitude values in *North* and *East* fields, respectively. Click on *Calculate*, and compare the results with the known values in your coordinate reference system.

10.5.1 Integrate an NTV2-transformation in QGIS

To integrate an NTV2 transformation file in QGIS you need one more step:

1. Place the NTV2 file (.gsb) in the CRS/Proj folder that QGIS uses (e.g. C:\OSGeo4W64\share\proj for windows users)
2. Add **nadgrids** (+nadgrids=nameofthefile.gsb) to the Proj definition in the *Parameters* field of the *Custom Coordinate Reference System Definition (Settings ► Custom Projections...)*.




图 10.4: Setting an NTV2 transformation

10.6 Datum Transformations

In QGIS, 'on-the-fly' CRS transformation is enabled by default, meaning that whenever you use layers with different coordinate systems QGIS transparently reprojects them to the project CRS. For some CRS, there are a number of possible transforms available to reproject to the project's CRS!


By default, QGIS will attempt to use the most accurate transformation available. However, in some cases this may not be possible, e.g. whenever additional support files are required to use a transformation. Whenever a more accurate transformation is available, but is not currently usable, QGIS will show an informative warning message advising you of the more accurate transformation and how to enable it on your system. Usually, this requires download of an external package of transformation support files, and extracting these to the `proj` folder under your QGIS *user profile* folder.

If desired, QGIS can also prompt you whenever multiple possible transformations can be made between two CRSs, and allow you to make an informed selection of which is the most appropriate transformation to use for your data.

This customization is done in the *Settings* ►  *Options* ► *Transformations* tab menu under the *Default datum transformations* group:

- using *Ask for datum transformation if several are available*: when more than one appropriate datum transformation exist for a source/destination CRS combination, a dialog will automatically be opened prompting users to choose which of these datum transformations to use for the project. If the *Make default* checkbox is ticked when selecting a transformation from this dialog, then the choice is remembered and automatically applied to any newly created QGIS projects.
- or defining a list of appropriate datum transformations to use as defaults when loading a layer to a project or reprojecting a layer.

Use the  button to open the *Select Datum Transformations* dialog. Then:

1. Choose the *Source CRS* of the layer, using the drop-down menu or the  *Select CRS* widget.
2. Provide the *Destination CRS* in the same way.
3. A list of available transformations from source to destination will be shown in the table. Clicking a row shows details on the settings applied and the corresponding accuracy and area of use of the transformation.

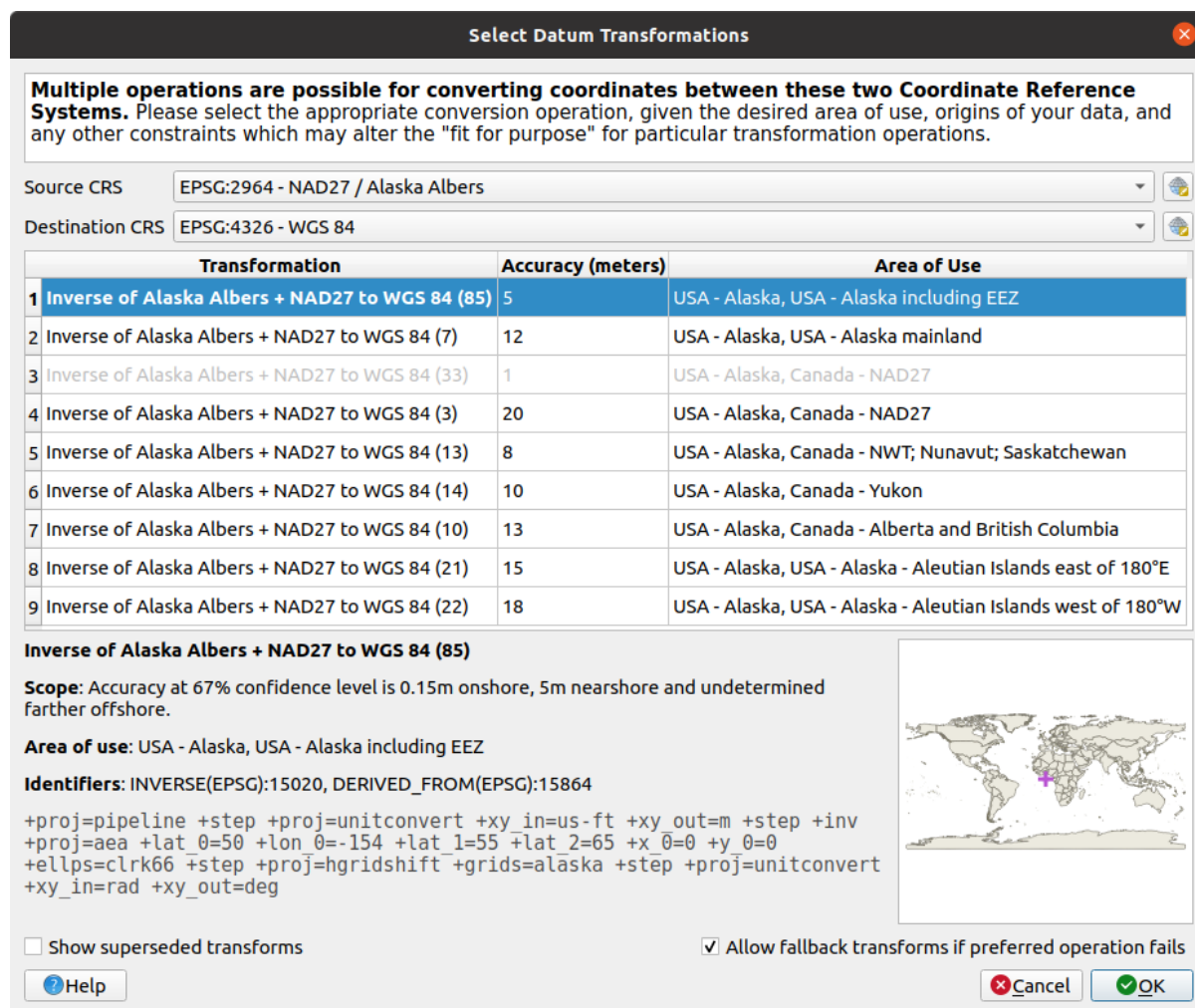


图 10.5: Selecting a preferred default datum transformation

In some cases a transformation may not be available for use on your system. In this case, the transformation will still be shown (greyed) in this list but can not be picked until you install the required package of transformation support. Usually, a button is provided to download and install the corresponding grid, which is then stored under the `proj` folder in the active *user profile* directory.

4. Find your preferred transformation and select it
5. Set whether you *Allow fallback transforms if preferred operation fails*
6. Click *OK*.

A row is added to the table under *Default Datum Transformations* with information about the *Source CRS*, the *Destination CRS*, the *Operation* applied for the transformation and whether *Allow fallback Transforms* is enabled.

From now, QGIS automatically uses the selected datum transformations for further transformation between these two CRSs until you remove it () from the list or change the entry () in the list.

Datum transformations set in the *Settings* ► *Options* ► *Transformations* tab will be inherited by all new QGIS projects created on the system. Additionally, a particular project may have its own specific set of transformations specified via the *CRS* tab of the *Project properties* dialog (*Project* ► *Properties...*). These settings apply to the current project only.

11.1 上下文帮助

每当您需要具体主题帮助时，可以通过多数对话框中的 *Help* 按钮访问当前用户手册中的相应页面 --- 请注意，第三方插件可以指向专用网页。

11.2 面板





By default, QGIS provides many panels to work with. Some of these panels are described below while others may be found in different parts of the document. A complete list of default panels provided by QGIS is available via the *View ► Panels ►* menu and mentioned at 面板.

11.2.1 Layers Panel

The *Layers* panel (also called the map legend) lists all the layers in the project and helps you manage their visibility. You can show or hide it by pressing `Ctrl+1`. A layer can be selected and dragged up or down in the legend to change the Z-ordering. Z-ordering means that layers listed nearer the top of the legend are drawn over layers listed lower down in the legend.

注解: The Z-ordering behavior can be overridden by the *Layer Order* panel.

At the top of the Layers panel, a toolbar allows you to:

-  Open the layer styling dock (F7): toggle the layer styling panel on and off.
-  Add new group
-  Manage Map Themes: control visibility of layers and arrange them in different map themes.
-  Filter Legend by Map Content: only the layers that are set visible and whose features intersect the current map canvas have their style rendered in the layers panel. Otherwise, a generic NULL symbol is applied to the layer. Based on the layer symbology, this is a convenient way to identify which kind of features from which layers cover your area of interest.






-  **Filter Legend by Expression**: apply an expression to remove styles from the selected layer tree that have no feature satisfying the condition. This can be used to highlight features that are within a given area/feature of another layer. From the drop-down list, you can edit and clear the expression currently applied.
-  **Expand All** or  **Collapse All** layers and groups in the layers panel.
-  **Remove Layer/Group** currently selected.










图 11.1: Layer Toolbar in Layers Panel

注解: Tools to manage the layers panel are also available for map and legend items in print layouts

Configuring map themes

The  **Manage Map Themes** drop-down button provides access to convenient shortcuts to manipulate visibility of the layers in the *Layers* panel:

-  *Show All Layers*
-  *Hide All Layers*
-  *Show Selected Layers*
-  *Hide Selected Layers*
-  *Toggle Selected Layers*: changes the visibility of the first selected layer in the panel, and applies that state to the other selected layers. Also accessible through `Space` shortcut.
- *Toggle Selected Layers Independently*: changes the visibility status of each selected layer
-  *Hide Deselected Layers*


Beyond the simple control of layer visibility, the  **Manage Map Themes** menu allows you to configure **Map Themes** in the legend and switch from one map theme to another. A map theme is a **snapshot** of the current map legend that records:


- the layers set as visible in the *Layers* panel
- **and** for each visible layer:
 - the reference to the *style* applied to the layer
 - the visible classes of the style, ie the layer checked node items in the *Layers panel*. This applies to *symbolologies* other than the single symbol rendering
 - the collapsed/expanded state of the layer node(s) and the group(s) it's placed inside

To create a map theme:

1. Check a layer you want to show
2. Configure the layer properties (symbolology, diagram, labels...) as usual
3. Expand the *Style* ► menu at the bottom and click on *Add...* to store the settings as *a new style embedded in the project*

注解: A map theme does not remember the current details of the properties: only a reference to the style name is saved, so whenever you apply modifications to the layer while this style is enabled (eg change the symbology rendering), the map theme is updated with new information.

4. Repeat the previous steps as necessary for the other layers
5. If applicable, expand or collapse groups or visible layer nodes in the *Layers* panel
6. Click on the  Manage Map Themes button on top of the panel, and *Add Theme...*
7. Enter the map theme's name and click *OK*










The new theme is listed in the lower part of the  drop-down menu.

You can create as many map themes as you need: whenever the current combination in the map legend (visible layers, their active style, the map legend nodes) does not match any existing map theme contents as defined above, click on *Add Theme...* to create a new map theme, or use *Replace Theme ▶* to update a map theme. You can rename the active map theme with *Rename Current Theme...* or use the *Remove Current Theme* button to delete it.

Map themes are helpful to switch quickly between different preconfigured combinations: select a map theme in the list to restore its combination. All configured themes are also accessible in the print layout, allowing you to create different map items based on specific themes and independent of the current main canvas rendering (see *Map item layers*).

Overview of the context menu of the Layers panel

At the bottom of the toolbar, the main component of the Layers panel is the frame listing vector or raster layers added to the project, optionally organized in groups. Depending on the item selected in the panel, a right-click shows a dedicated set of options presented below.


Option	Vector Layer	Raster Layer	Group
 <i>Zoom to Layer/Group</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
 <i>Zoom to Selection</i>	<input checked="" type="checkbox"/>		
 <i>Show in Overview</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<i>Show Feature Count</i>	<input checked="" type="checkbox"/>		
<i>Copy Layer/Group</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Rename Layer/Group</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
 <i>Zoom to Native Resolution (100%)</i>		<input checked="" type="checkbox"/>	
<i>Stretch Using Current Extent</i>		<input checked="" type="checkbox"/>	
 <i>Update SQL Layer...</i>	<input checked="" type="checkbox"/>		
 <i>Add Group</i>			<input checked="" type="checkbox"/>
 <i>Duplicate Layer</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
 <i>Remove Layer/Group...</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Move Out of Group</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<i>Move to Top</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Move to Bottom</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Check and all its Parents</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<i>Group Selected</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
 <i>Open Attribute Table</i>	<input checked="" type="checkbox"/>		

下页继续

表 11.1 - 续上页


Option	Vector Layer	Raster Layer	Group
 <i>Toggle Editing</i>	<input checked="" type="checkbox"/>		
 <i>Current Edits</i> ▶	<input checked="" type="checkbox"/>		
<i>Filter...</i>	<input checked="" type="checkbox"/>		
<i>Change Data Source...</i>	<input checked="" type="checkbox"/>		
<i>Repair Data Source...</i>	<input checked="" type="checkbox"/>		
<i>Actions on selections</i> ▶ (in edit mode)	<input checked="" type="checkbox"/>		
▶ <i>Duplicate Feature</i>	<input checked="" type="checkbox"/>		
▶ <i>Duplicate Feature and Digitize</i>	<input checked="" type="checkbox"/>		
<i>Set Layer Scale Visibility...</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<i>Zoom to Visible Scale</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<i>Set CRS</i> ▶	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
▶ <i>Set Layer/Group CRS...</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
▶ <i>Set Project CRS from Layer</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<i>Set Group WMS Data...</i>			<input checked="" type="checkbox"/>
<input type="checkbox"/> <i>Mutually Exclusive Group</i>			<input checked="" type="checkbox"/>
<i>Check and all its children (Ctrl-click)</i>			<input checked="" type="checkbox"/>
<i>Uncheck and all its children (Ctrl-click)</i>			<input checked="" type="checkbox"/>
<i>Make Permanent</i>	<input checked="" type="checkbox"/>		
<i>Export</i> ▶	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
▶ <i>Save As...</i>		<input checked="" type="checkbox"/>	
▶ <i>Save Features As...</i>	<input checked="" type="checkbox"/>		
▶ <i>Save Selected Features As...</i>	<input checked="" type="checkbox"/>		
▶ <i>Save As Layer Definition File...</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
▶ <i>Save As QGIS Layer Style File...</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<i>Styles</i> ▶	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
▶ <i>Copy Style</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
▶ <i>Paste Style</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
▶ <i>Add...</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
▶ <i>Rename Current...</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
▶ <i>Edit symbol...</i>	<input checked="" type="checkbox"/>		
▶ <i>Copy Symbol</i>	<input checked="" type="checkbox"/>		
▶ <i>Paste Symbol</i>	<input checked="" type="checkbox"/>		
<i>Properties...</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	

Table: Context menu from Layers Panel items

For GRASS vector layers,  *Toggle editing* is not available. See section *Digitizing and editing a GRASS vector layer* for information on editing GRASS vector layers.

Interact with groups and layers

Layers in the legend window can be organized into groups. There are two ways to do this:

1. Press the  icon to add a new group. Type in a name for the group and press `Enter`. Now click on an existing layer and drag it onto the group.
2. Select some layers, right-click in the legend window and choose *Group Selected*. The selected layers will automatically be placed in a new group.

To move a layer out of a group, drag it out, or right-click on it and choose *Move Out of Group*: the layer is moved from the group and placed above it. Groups can also be nested inside other groups. If a layer is placed in a nested group, *Move Out of Group* will move the layer out of all nested groups.

To move a group or layer to the top of the layer panel, either drag it to the top, or choose *Move to Top*. If you use this option on a layer nested in a group, the layer is moved to the top in its current group. The *Move to Bottom* option follows the same logic to move layers and groups down.

The checkbox for a group will show or hide the checked layers in the group with one click. With `Ctrl` pressed, the checkbox will also turn on or off all the layers in the group and its sub-groups.

`Ctrl`-click on a checked / unchecked layer will uncheck / check the layer and all its parents.










Enabling the **Mutually Exclusive Group** option means you can make a group have only one layer visible at the same time. Whenever a layer within the group is set visible the others will be toggled not visible.

It is possible to select more than one layer or group at the same time by holding down the `Ctrl` key while clicking additional layers. You can then move all selected layers to a new group at the same time.

You may also delete more than one layer or group at once by selecting several items with the `Ctrl` key and then pressing `Ctrl+D`: all selected layers or groups will be removed from the layers list.

More information on layers and groups using indicator icon

In some circumstances, icons appears next to the layer or group in the *Layers* panel to give more information about the layer/group. These symbols are:

-  to indicate that the layer is in edit mode and you can modify the data
-  to indicate that the layer being edited has some unsaved changes
-  to indicate *a filter* applied to the layer. Hover over the icon to see the filter expression and double-click to update the setting
-  to identify layers that are *required* in the project, hence non removable
-  to identify an *embedded group or layer* and the path to their original project file
-  to identify a layer whose data source was not available at the project file opening (see [处理损坏的文件路径](#)). Click the icon to update the source path or select *Repair Data Source...* entry from the layer contextual menu.
-  to remind you that the layer is a *temporary scratch layer* and its content will be discarded when you close this project. To avoid data loss and make the layer permanent, click the icon to store the layer in any of the OGR vector formats supported by QGIS.
-  to identify a layer that has no/unknown CRS
-  to identify a temporal layer controlled by canvas animation

Editing vector layer style

From the Layers panel, you have shortcuts to change the layer rendering quickly and easily. Right-click on a vector layer and select *Styles* ► in the list in order to:

- see the *styles* currently applied to the layer. If you defined many styles for the layer, you can switch from one to another and your layer rendering will automatically be updated on the map canvas.
- copy part or all of the current style, and when applicable, paste a copied style from another layer




小技巧: Quickly share a layer style

From the context menu, copy the style of a layer and paste it to a group or a selection of layers: the style is applied to all the layers that are of the same type (vector/raster) as the original layer and, for vector layers, have the same geometry type (point, line or polygon).

-
- rename the current style, add a new style (which is actually a copy of the current one) or delete the current style (when multiple styles are available).

注解: The previous options are also available for raster or mesh layers.

- update the *symbol color* using a **Color Wheel**. For convenience, the recently used colors are also available at the bottom of the color wheel.
- *Edit Symbol...*: open the *Symbol Selector* dialog and change feature symbol (symbol, size, color...).

When using a classification symbology type (based on *categorized*, *graduated* or *rule-based*), the aforementioned symbol-level options are available from the class entry context menu. Also provided are the  *Toggle Items*,  *Show All Items* and  *Hide All Items* entries to switch the visibility of all the classes of features. These avoid (un)checking items one by one.








小技巧: Double-clicking a class leaf entry also opens the *Symbol Selector* dialog.



11.2.2 Layer Styling Panel



The *Layer Styling* panel (also enabled with `Ctrl+3`) is a shortcut to some of the functionalities of the *Layer Properties* dialog. It provides a quick and easy way to define the rendering and the behavior of a layer, and to visualize its effects without having to open the layer properties dialog.

In addition to avoiding the blocking (or "modal") layer properties dialog, the layer styling panel also avoids cluttering the screen with dialogs, and contains most style functions (color selector, effects properties, rule edit, label substitution...): e.g., clicking color buttons inside the layer style panel causes the color selector dialog to be opened inside the layer style panel itself rather than as a separate dialog.

From a drop-down list of current layers in the layer panel, select an item and:

- Depending on the layer type, set:
 -  *Symbology*,  *Transparency*, and  *Histogram* properties for raster layer. These options are the same as in the *Raster Properties Dialog*.
 -  *Symbology*,  *Labels*,  *Mask* and  *3D View* properties for vector layer. These options are the same as in the *The Vector Properties Dialog* and can be extended by custom properties introduced by third-party plugins.

–  *Symbology* and  *3D View* properties for mesh layer. These options are the same as in the *Mesh Dataset Properties*.

- Manage the associated style(s) in the  *Style Manager* (more details at *Managing Custom Styles*).
- See the  *History* of changes you applied to the layer style in the current project: you can therefore cancel or restore to any state by selecting it in the list and clicking *Apply*.

For Vector Tile layers there is an option to show *Visible rules only*. This is very useful if you just want to work with rules that fall inside the current map canvas zoom level.

Another powerful feature of this panel is the *Live update* checkbox. Tick it to render your changes immediately on the map canvas: you no longer need to click the *Apply* button.

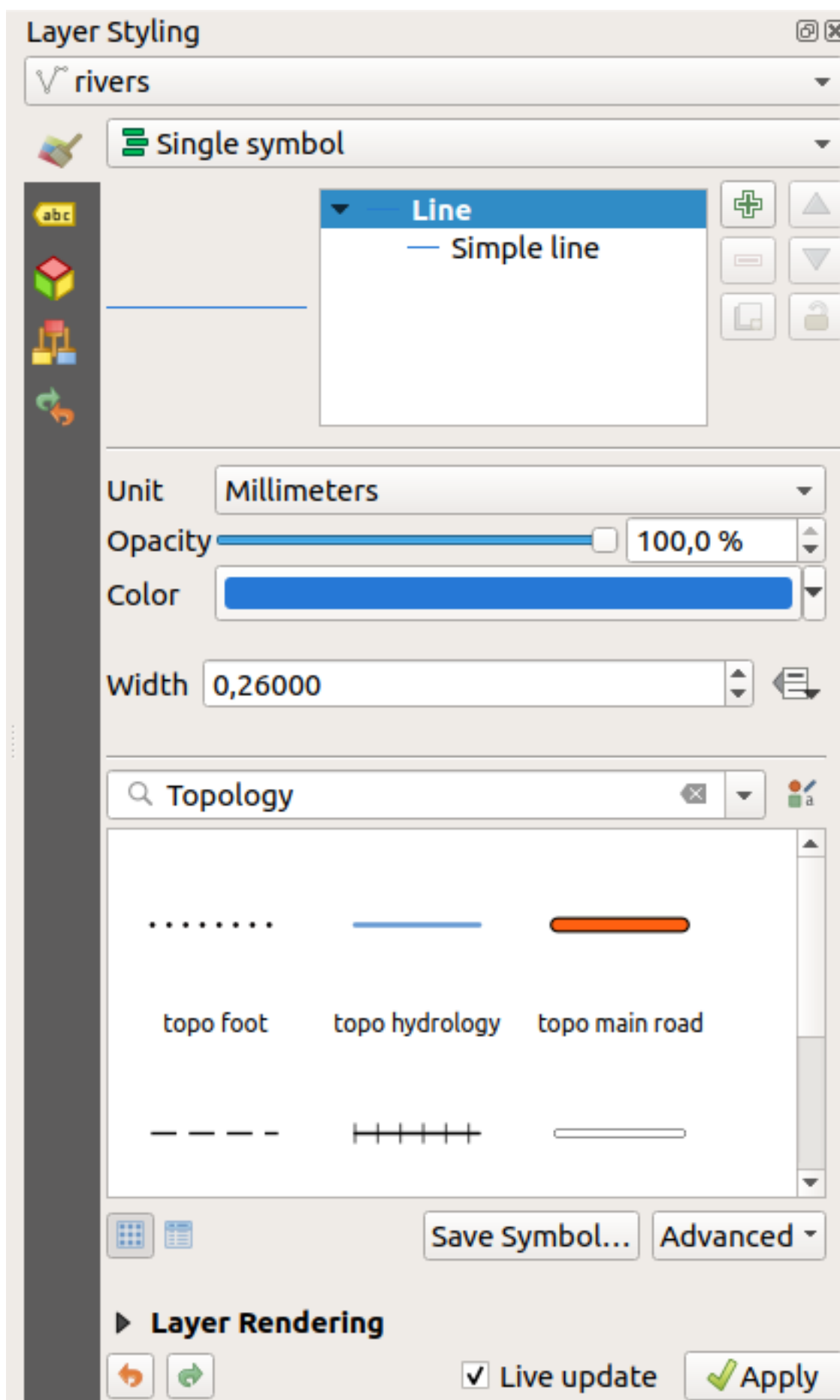


图 11.2: Defining a layer's symbology from the layer styling panel

11.2.3 Layer Order Panel

By default, layers shown on the QGIS map canvas are drawn following their order in the *Layers* panel: the higher a layer is in the panel, the higher (hence, more visible) it'll be in the map view.

You can define a drawing order for the layers independent of the order in the layers panel with the *Layer Order* panel enabled in *View ► Panels ►* menu or with `Ctrl+9`. Check *Control rendering order* underneath the list of layers and reorganize the layers in the panel as you want. This order becomes the one applied to the map canvas. For example, in 图 11.3, you can see that the `airports` features are displayed over the `alaska` polygon despite those layers' respective placement in the Layers panel.

Unchecking *Control rendering order* will revert to default behavior.

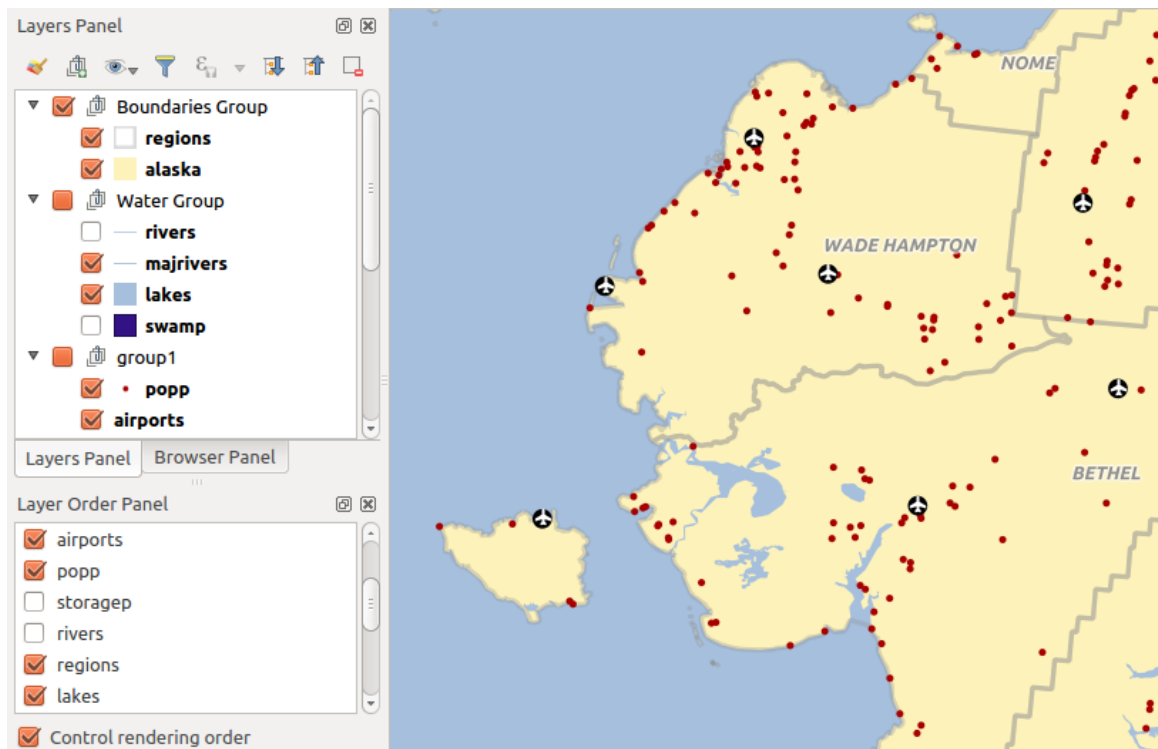



图 11.3: Define a layer order independent of the legend

11.2.4 Overview Panel

The *Overview* panel (`Ctrl+8`) displays a map with a full extent view of some of the layers. The Overview map is filled with layers using the *Show in Overview* option from the *Layer* menu or in the layer contextual menu. Within the view, a red rectangle shows the current map canvas extent, helping you quickly to determine which area of the whole map you are currently viewing. If you click-and-drag the red rectangle in the overview frame, the main map view extent will update accordingly.

Note that labels are not rendered to the map overview even if the layers used in the map overview have been set up for labeling.

11.2.5 Log Messages Panel


When loading or processing some operations, you can track and follow messages that appear in different tabs using the  *Log Messages Panel*. It can be activated using the most right icon in the bottom status bar.

11.2.6 Undo/Redo Panel

For each layer being edited, the *Undo/Redo* (Ctrl+5) panel shows the list of actions carried out, allowing you quickly to undo a set of actions by selecting the action listed above. More details at [Undo and Redo edits](#).

11.2.7 Statistical Summary Panel

The *Statistics* panel (Ctrl+6) provides summarized information on any vector layer. This panel allows you to select:



- the vector layer to compute the statistics on
- the column to use, or an  *expression*
- the statistics to return using the drop-down button at the bottom-right of the dialog. Depending on the field's (or expression's values) type, available statistics are:

Statistics	String	Integer	Float	Date
Count	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Count Distinct Value	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>
Count Missing value	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Sum		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Mean		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Standard Deviation		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Standard Deviation on Sample		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Minimal value	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Maximal value	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
范围 (最大值和最小值之差)		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Minority	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Majority	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Variety		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
First Quartile		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Third Quartile		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Inter Quartile Range		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Minimum Length	<input checked="" type="checkbox"/>			
Maximum Length	<input checked="" type="checkbox"/>			
Mean Length	<input checked="" type="checkbox"/>			

Table: Statistics available for each field type

The statistical summary can be:

- returned for the whole layer or *selected features only*

- recalculated using the  button when the underlying data source changes (eg, new or removed features/fields, attribute modification)
-  copied to the clipboard and pasted as a table in another application

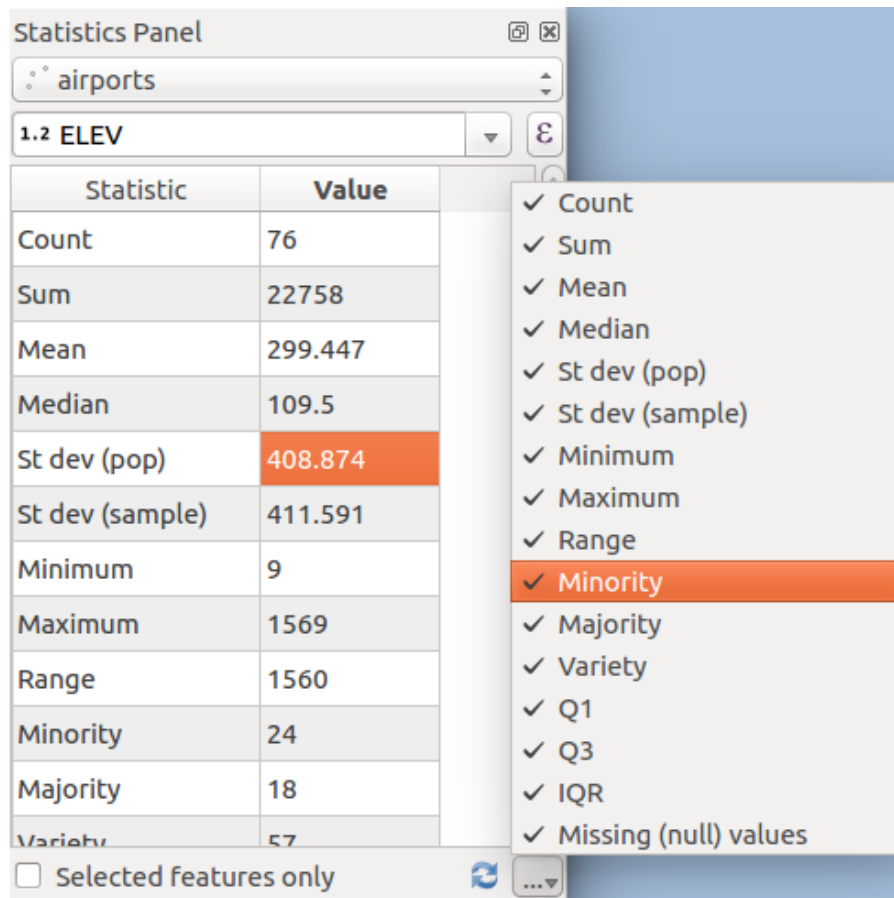


图 11.4: Show statistics on a field

11.3 Nesting Projects

Sometimes, you'd like to keep some layers in different projects, but with the same style. You can either create a *default style* for these layers or embed them from another project to save time and effort.


Embed layers and groups from an existing project has some advantages over styling:

- All types of layers (vector or raster, local or online...) can be added
- Fetching groups and layers, you can keep the same tree structure of the "background" layers in your different projects
- While the embedded layers are editable, you can't change their properties such as symbology, labels, forms, default values and actions, ensuring consistency across projects
- Modify the items in the original project and changes are propagated to all the other projects

If you want to embed content from other project files into your project, select *Layer ► Embed Layers and Groups*:

1. Click the ... button to look for a project: you can see the content of the project (see 图 11.5)
2. Hold down **Ctrl** (or **X Cmd**) and click on the layers and groups you wish to retrieve

3. Click *OK*

The selected layers and groups are embedded in the *Layers* panel and displayed on the map canvas. An  icon is added next to their name for recognition and hovering over displays a tooltip with the original project file path.

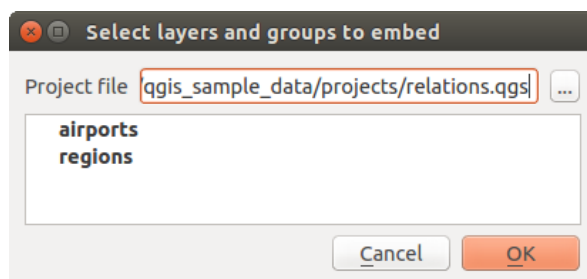


图 11.5: Select layers and groups to embed

Like any other layer, an embedded layer can be removed from the project by right-clicking on the layer and clicking



小技巧: Change rendering of an embedded layer

It's not possible to change the rendering of an embedded layer, unless you make the changes in the original project file. However, right-clicking on a layer and selecting *Duplicate* creates a layer which is fully-featured and not dependent on the original project. You can then safely remove the linked layer.

11.4 Working with the map canvas

11.4.1 Rendering

By default, QGIS renders all visible layers whenever the map canvas is refreshed. The events that trigger a refresh of the map canvas include:


- adding a layer
- panning or zooming
- resizing the QGIS window
- changing the visibility of a layer or layers

QGIS allows you to control the rendering process in a number of ways.

Scale Dependent Rendering

Scale-dependent rendering allows you to specify the minimum and maximum scales at which a layer (raster or vector) will be visible. To set scale-dependent rendering, open the *Properties* dialog by double-clicking on the layer in the legend. On the *Rendering* tab, tick *Scale dependent visibility* and enter the *Minimum (exclusive)* and *Maximum (inclusive)* scale values.

You can also activate scale dependent visibility on a layer from the Layers panel. Right-click on the layer and in the context menu, select *Set Layer Scale Visibility*.

The  *Set to current canvas scale* button allow you to use the current map canvas scale as boundary of the range visibility.

注解: When a layer is not rendered in the map canvas because the map scale is out of its visibility scale range, the layer is greyed in the Layers panel and a new option *Zoom to Visible Scale* appears in the layer context menu. Select it and the map is zoomed to the layer's nearest visibility scale.

Controlling Map Rendering

Map rendering can be controlled in various ways, as described below.

Suspending Rendering

To suspend rendering, click the *Render* checkbox in the bottom-right corner of the status bar. When *Render* is not checked, QGIS does not redraw the canvas in response to any of the events described in the section *Rendering*. Examples of when you might want to suspend rendering include:

- adding many layers and symbolizing them prior to drawing
- adding one or more large layers and setting scale dependency before drawing
- adding one or more large layers and zooming to a specific view before drawing
- any combination of the above

Checking the *Render* checkbox enables rendering and causes an immediate refresh of the map canvas.

Setting Layer Add Option

You can set an option to always load new layers without drawing them. This means the layer will be added to the map, but its visibility checkbox in the legend will be unchecked by default. To set this option, choose menu option *Settings* ► *Options* and click on the *Rendering* tab. Uncheck *By default new layers added to the map should be displayed*. Any layer subsequently added to the map will be off (invisible) by default.

Stopping Rendering

To stop the map drawing, press the `ESC` key. This will halt the refresh of the map canvas and leave the map partially drawn. It may take a bit of time between pressing `ESC` for the map drawing to halt.

Influence Rendering Quality

QGIS has an option to influence the rendering quality of the map. Choose menu option *Settings* ► *Options*, click on the *Rendering* tab and select or deselect *Make lines appear less jagged at the expense of some drawing performance*.











Speed-up rendering


There are some settings that allow you to improve rendering speed. Open the QGIS options dialog using *Settings* ► *Options*, go to the *Rendering* tab and select or deselect the following checkboxes:

- *Use render caching where possible to speed up redraws.*
- *Render layers in parallel using many CPU cores* and then set the *Max cores to use.*
- The map renders in the background onto a separate image and each *Map Update interval*, the content from this (off-screen) image will be taken to update the visible screen representation. However, if rendering finishes faster than this duration, it will be shown instantaneously.
- With *Enable Feature simplification by default for newly added layers*, you simplify features' geometry (fewer nodes) and as a result, they display more quickly. Be aware that this can cause rendering inconsistencies.




11.4.2 Zooming and Panning

There are multiple ways to zoom and pan to an area of interest. You can use the *Map Navigation* toolbar, the mouse and keyboard on the map canvas and also the menu actions from the *View* menu and the layers' contextual menu in the *Layers* panel.


Icon	Label	Usage	View menu	Map Navigation Toolbar	Layer Contextual Menu
	Pan Map	When activated, left click anywhere on the map canvas to pan the map at the cursor position. You can also pan the map by holding down the left mouse button and dragging the map canvas.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
	Zoom In	When activated, left click anywhere on the map canvas to zoom in one level. The mouse cursor position will be the center of the zoomed area of interest. You can also zoom in to an area by dragging a rectangle on the map canvas with the left mouse button.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
	Zoom Out	When activated, left click anywhere on the map canvas to zoom out one level. The mouse cursor position will be the center of the zoomed area of interest. You can also zoom out from an area by dragging a rectangle on the map canvas with the left mouse button.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
	Pan Map to Selection	Pan the map to the active layer's selected features.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
	Zoom To Selection	Zoom to the active layer's selected features.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
	Zoom To Layer	Zoom to the active layer's extent.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
	Zoom Full	Zoom to the extent of all the layers in the project.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
	Zoom Last	Zoom the map to the previous extent in history.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
	Zoom Next	Zoom the map to the next extent in history.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
	Zoom to Native Resolution	Zoom the map to a level where one pixel of the active raster layer covers one screen pixel.	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

A *Zoom factor* can be set under the *Settings* ►  *Options* ► *Map tools* menu to define the scale behavior while zooming. There, you can also set a list of *Predefined Scales* that will be available at the bottom of the map canvas.

With the Mouse on the Map Canvas

In addition to using the  Pan  Zoom In and  Zoom Out tools described above, you can hold the mouse wheel inside of the map canvas and drag the mouse cursor (on macOS, you may need to hold down the `cmd` key). You can also roll the mouse wheel to zoom in and out on the map. The mouse cursor position will be the center of the zoomed area of interest. Holding down `Ctrl` while rolling the mouse wheel results in a finer zoom.

With the Keyboard on the Map Canvas

Holding down `spacebar` on the keyboard and moving the mouse cursor will pan the map the same way dragging the map canvas with  Pan does.

Panning the map is possible with the arrow keys. Place the mouse cursor inside the map area, and press on the arrow keys to pan up, down, left and right.

The `PgUp` and `PgDown` keys on the keyboard will cause the map display to zoom in or out following the zoom factor set. Pressing `Ctrl++` or `Ctrl+-` also performs an immediate zoom in/out on the map canvas.

When certain map tools are active (Identify, Measure...), you can perform a zoom by holding down `Shift` and dragging a rectangle on the map to zoom to that area. This is not enabled for selection tools (since they use `Shift` for adding to selection) or edit tools.

11.4.3 Spatial Bookmarks

Spatial Bookmarks allow you to "bookmark" a geographic location and return to it later. By default, bookmarks are saved in the user's profile (as *User Bookmarks*), meaning that they are available from any project the user opens. They can also be saved for a single project (named *Project Bookmarks*) and stored within the project file, which can be helpful if the project is to be shared with other users.

Creating a Bookmark

To create a bookmark:

1. Zoom and pan to the area of interest.
2. Select the menu option `View` ►  `New Spatial Bookmark...`, press `Ctrl+B` or right-click the  `Spatial Bookmarks` entry in the *Browser* panel and select `New Spatial Bookmark`. The *Bookmark Editor* dialog opens.

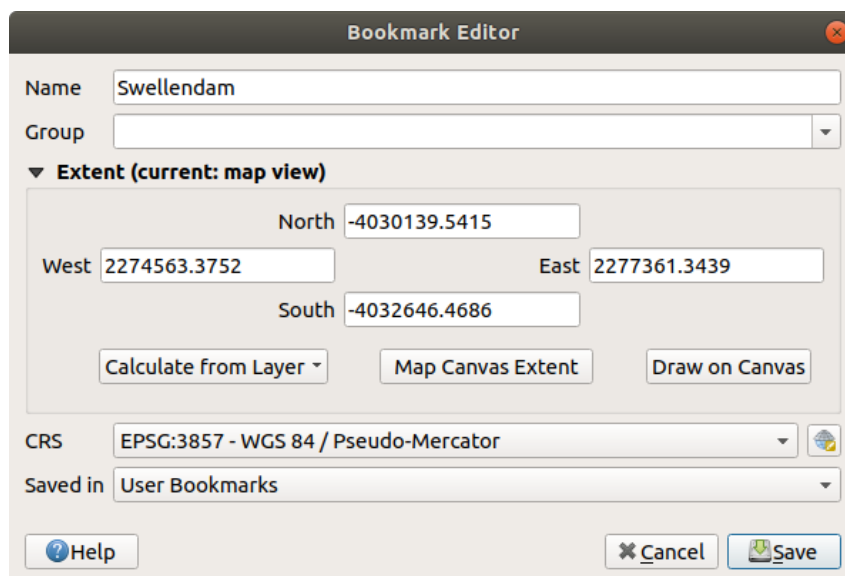


图 11.6: The Bookmark Editor Dialog



3. Enter a descriptive name for the bookmark
4. Enter or select a group name in which to store related bookmarks
5. Select the extent of the area you wish to save, using the extent selector; the extent can be calculated from a loaded layer extent, the current map canvas or drawn over the current map canvas.
6. Indicate the *CRS* to use for the extent
7. Select whether the bookmark will be *Saved in User Bookmarks* or *Project Bookmarks*
8. Press *Save* to add the bookmark to the list

Note that you can have multiple bookmarks with the same name.



Working with Bookmarks

To use and manage bookmarks, you can either use the *Spatial Bookmarks* panel or *Browser*.

Select **View** ►  *Show Spatial Bookmark Manager* or press **Ctrl+7** to open the *Spatial Bookmarks Manager* panel.







Select **View** ►  *Show Bookmarks* or **Ctrl+Shift+B** to show the  *Spatial Bookmarks* entry in the *Browser* panel.

You can perform the following tasks:

Task	Spatial Bookmark Manager	Browser
Zoom to a Bookmark	Double-click on it, or select the bookmark and press the  <i>Zoom to bookmark</i> button.	Double-click on it, drag and drop it to the map canvas, or right-click the bookmark and select <i>Zoom to Bookmark</i> .
Delete a bookmark	Select the bookmark and click the  <i>Delete bookmark</i> button. Confirm your choice.	Right-click the bookmark and select <i>Delete Spatial Bookmark</i> . Confirm your choice.

下页继续

表 11.3 - 续上页

Task	Spatial Bookmark Manager	Browser
Export bookmarks to XML	Click the  <i>Import/Export Bookmarks</i> button and select  <i>Export</i> . All the bookmarks (user or project) are saved in an xml file.	Select one or more folders (user or project) or subfolders (groups), then right-click and select  <i>Export Spatial Bookmarks...</i> The selected bookmark subset is saved.
Import bookmarks from XML	Click the  <i>Import/Export Bookmarks</i> button and select  <i>Import</i> . All bookmarks in the XML file are imported as user bookmarks.	Right-click the <i>Spatial Bookmarks</i> entry or one of its folders (user or project) or subfolders (groups) to determine where to import the bookmarks, then select  <i>Import Spatial Bookmarks</i> . If performed on the <i>Spatial Bookmarks</i> entry, the bookmarks are added to <i>User Bookmarks</i> .
Edit bookmark	You can change a bookmark by changing the values in the table. You can edit the name, the group, the extent and if it is stored in the project or not.	Right-click the desired bookmark and select <i>Edit Spatial Bookmark...</i> The <i>Bookmark Editor</i> will open, allowing you to re-define every aspect of the bookmark as if you were creating it for the first time. You can also drag and drop the bookmark between folders (user and project) and subfolders (groups).

You can also zoom to bookmarks by typing the bookmark name in the *locator*.

11.4.4 Decorations

Decorations include Grid, Title Label, Copyright Label, Image, North Arrow, Scale Bar and Layout Extents. They are used to 'decorate' the map by adding cartographic elements.

Grid



Grid allows you to add a coordinate grid and coordinate annotations to the map canvas.

1. Select menu option *View ► Decorations ► Grid...* to open the dialog.

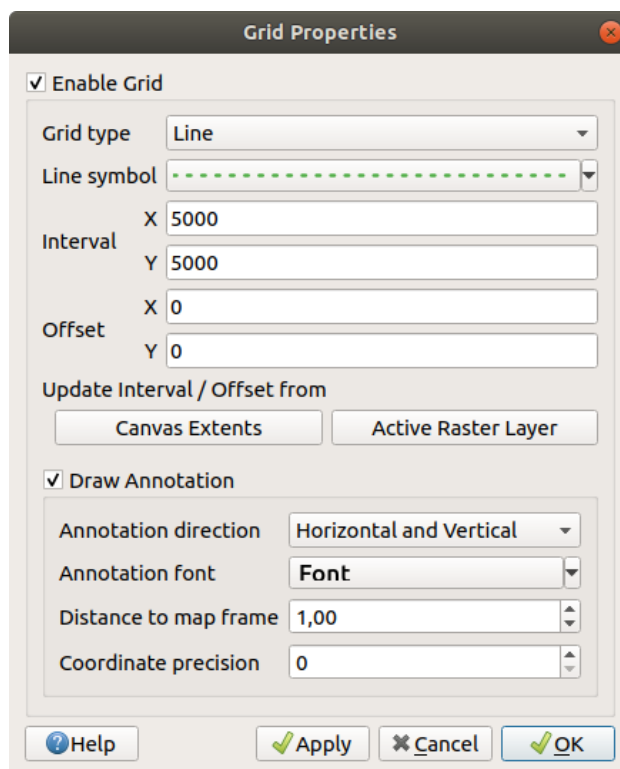



图 11.7: The Grid Dialog

2. Tick *Enable grid* and set grid definitions according to the layers loaded in the map canvas:
 - The *Grid type*: it can be *Line* or *Marker*
 - The associated *Line symbol* or *marker symbol* used to represent the grid marks
 - The *Interval X* and *Interval Y* between the grid marks, in map units
 - An *Offset X* and *Offset Y* distance of the grid marks from the bottom left corner of the map canvas, in map units
 - The interval and offset parameters can be set based on the:
 - *Canvas Extents*: generates a grid with an interval that is approximately 1/5 of the canvas width
 - *Active Raster Layer* resolution
3. Tick *Draw annotations* to display the coordinates of the grid marks and set:
 - The *Annotation direction*, ie how the labels would be placed relative to their grid line. It can be:
 - *Horizontal* or *Vertical* for all the labels
 - *Horizontal and Vertical*, ie each label is parallel to the grid mark it refers to
 - *Boundary direction*, ie each label follows the canvas boundary, and is perpendicular to the grid mark it refers to
 - The *Annotation font* (text formatting, buffer, shadow...) using the *font selector widget*
 - The *Distance to map frame*, margin between annotations and map canvas limits. Convenient when *exporting the map canvas* eg to an image format or PDF, and avoid annotations to be on the "paper" limits.
 - The *Coordinate precision*
4. Click *Apply* to verify that it looks as expected or *OK* if you're satisfied.

Title Label

 **Title Label** allows you to decorate your map with a **Title**.

To add a Title Label decoration:

1. Select menu option *View ► Decorations ► Title Label...* to open the dialog.

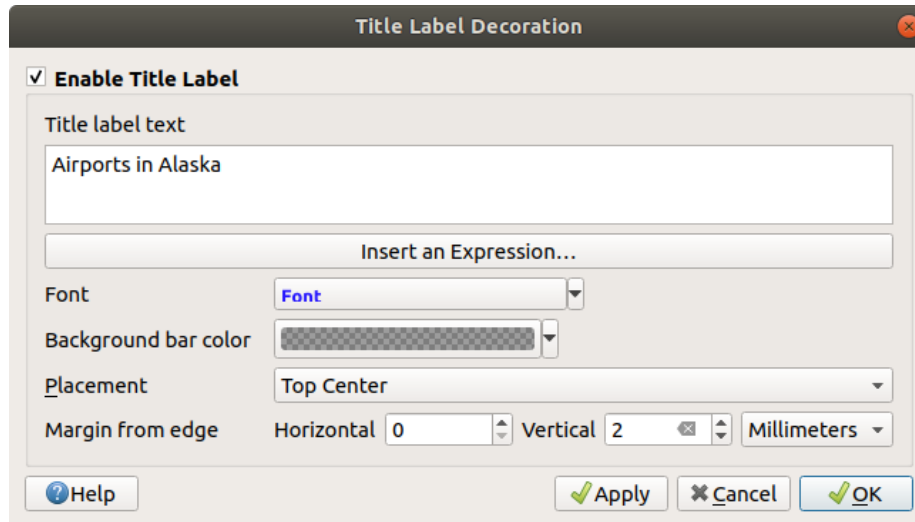



图 11.8: The Title Decoration Dialog

2. Make sure  *Enable Title Label* is checked
3. Enter the title text you want to place on the map. You can make it dynamic using the *Insert or Edit an Expression...* button.
4. Choose the *Font* for the label using the *font selector widget* with full access to QGIS *text formatting* options. Quickly set the font color and opacity by clicking the black arrow to the right of the font combo box.
5. Select the *color* to apply to the title's *Background bar color*.
6. Choose the *Placement* of the label in the canvas: options are *Top left*, *Top Center* (default), *Top Right*, *Bottom left*, *Bottom Center* and *Bottom Right*.
7. Refine the placement of the item by setting a horizontal and/or vertical *Margin from Edge*. These values can be in **Millimeters** or **Pixels** or set as a **Percentage** of the width or height of the map canvas.
8. Click *Apply* to verify that it looks as expected or *OK* if you're satisfied.

Copyright Label

 **Copyright Label** can be used to decorate your map with a **Copyright** label.

To add this decoration:

1. Select menu option *View ► Decorations ► Copyright Label...* to open the dialog.

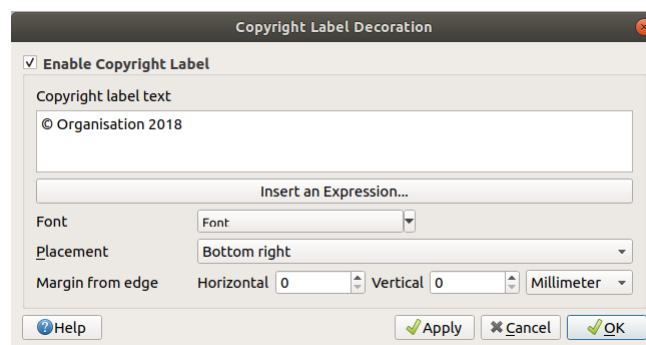



图 11.9: The Copyright Decoration Dialog

2. Make sure *Enable Copyright Label* is checked
3. Enter the copyright text you want to place on the map. You can make it dynamic using the *Insert or Edit an Expression...* button.
4. Choose the *Font* for the label using the *font selector widget* with full access to QGIS *text formatting* options. Quickly set the font color and opacity by clicking the black arrow to the right of the font combo box.
5. Choose the *Placement* of the label in the canvas: options are *Top left*, *Top Center*, *Top Right*, *Bottom left*, *Bottom Center*, and *Bottom Right* (default for Copyright decoration)
6. Refine the placement of the item by setting a horizontal and/or vertical *Margin from Edge*. These values can be in **Millimeters** or **Pixels** or set as a **Percentage** of the width or height of the map canvas.
7. Click *Apply* to verify that it looks as expected or *OK* if you're satisfied.

Image Decoration

 *Image* allows you to add an image (logo, legend, ..) on the map canvas.

To add an image:

1. Select menu option *View ► Decorations ► Image...* to open the dialog.

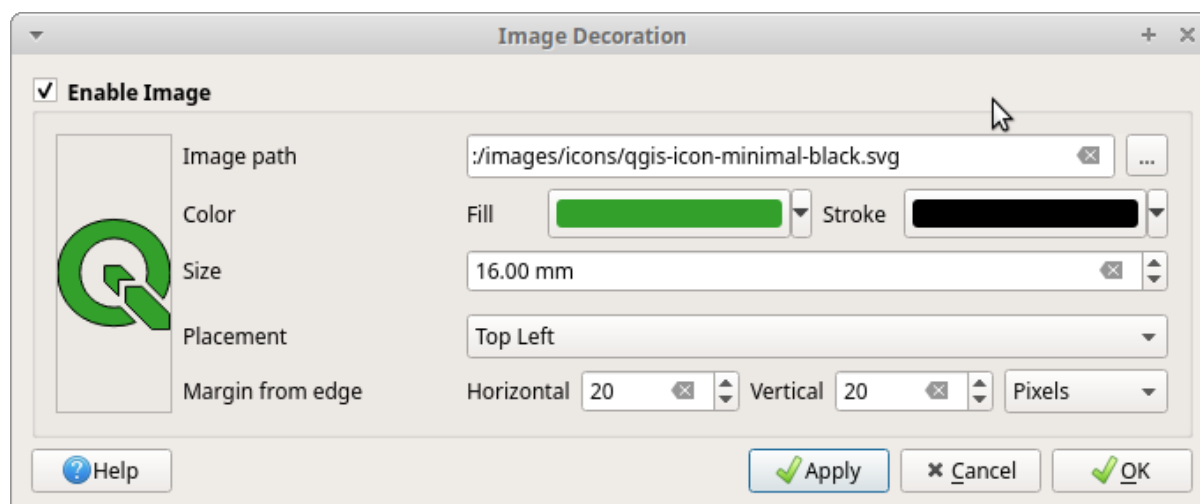



图 11.10: The Image Decoration Dialog

2. Make sure *Enable Image* is checked
3. Select a bitmap (e.g. png or jpg) or SVG image using the ... ^{Browse} button

4. If you have chosen a parameter enabled SVG then you can also set a *Fill* or *Stroke* (outline) color. For bitmap images, the color settings are disabled.
5. Set a *Size* of the image in mm. The width of selected image is used to resize it to given *Size*.
6. Choose where you want to place the image on the map canvas with the *Placement* combo box. The default position is *Top Left*.
7. Set the *Horizontal* and *Vertical Margin from (Canvas) Edge*. These values can be set in **Millimeters**, **Pixels** or as a **Percentage** of the width or height of the map canvas.
8. Click *Apply* to verify that it looks as expected and *OK* if you're satisfied.

North Arrow

 *North Arrow* allows you to add a north arrow on the map canvas.

To add a north arrow:

1. Select menu option *View* ► *Decorations* ► *North Arrow...* to open the dialog.

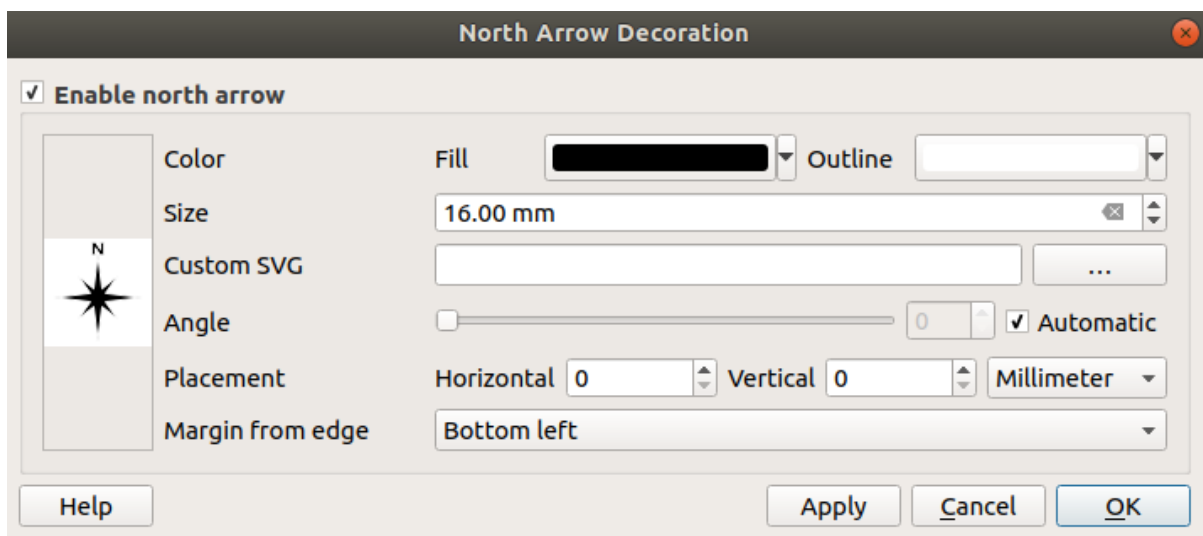



图 11.11: The North Arrow Dialog

2. Make sure *Enable north arrow* is checked
3. Optionally change the color and size, or choose a custom SVG
4. Optionally change the angle or choose **Automatic** to let QGIS determine the direction
5. Optionally choose the placement from the Placement combo box
6. Optionally refine the placement of the arrow by setting a horizontal and/or vertical *Margin from (Canvas) Edge*. These values can be in **Millimeters** or **Pixels** or set as a **Percentage** of the width or height of the map canvas.
7. Click *Apply* to verify that it looks as expected and *OK* if you're satisfied.

Scale Bar

 *Scale Bar* adds a simple scale bar to the map canvas. You can control the style and placement, as well as the labelling of the bar.

QGIS only supports displaying the scale in the same units as your map frame. So, if the units of your project's CRS are meters, you can't create a scale bar in feet. Likewise, if you are using decimal degrees, you can't create a scale bar to display distance in meters.

To add a scale bar:

1. Select menu option *View ► Decorations ► Scale Bar...* to open the dialog

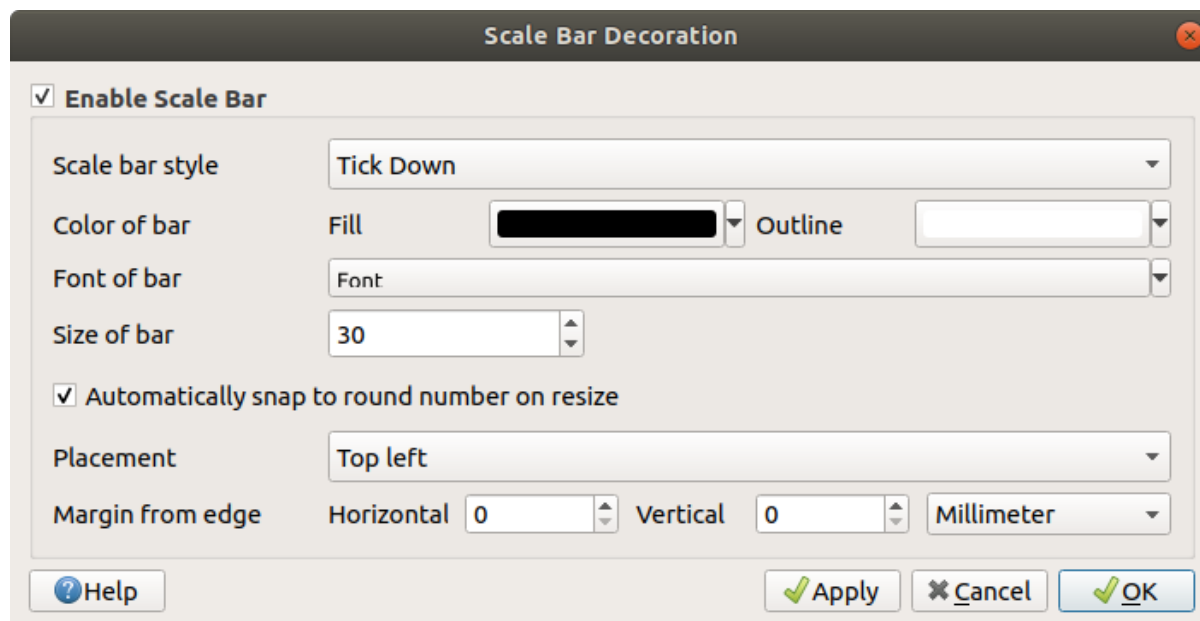







图 11.12: The Scale Bar Dialog

2. Make sure *Enable scale bar* is checked
3. Choose a style from the *Scale bar style*  combo box
4. Select the *Color of bar*  by choosing a fill color (default: black) and an outline color (default: white). The scale bar fill and outline can be made opaque by clicking on the down arrow to the right of the color input.
5. Select the font for the scale bar from the *Font of bar*  combo box
6. Set the *Size of bar* 
7. Optionally check *Automatically snap to round number on resize* to display easy-to-read values
8. Choose the placement from the *Placement*  combo box
9. You can refine the placement of the item by setting a horizontal and/or vertical *Margin from (Canvas) Edge*. These values can be in **Millimeters** or **Pixels** or set as a **Percentage** of the width or height of the map canvas.
10. Click *Apply* to verify that it looks as expected or *OK* if you're satisfied.

Layout Extents


 *Layout Extents* adds the extents of *map item(s)* in print layout(s) to the canvas. When enabled, the extents of all map items within all print layouts are shown using a lightly dotted border labeled with the name of the print layout and map item. You can control the style and labeling of the displayed layout extents. This decoration is useful when you are tweaking the positioning of map elements such as labels, and need to know the actual visible region of print layouts.



图 11.13: Example of layout extents displayed in a QGIS project with two print layouts. The print layout named 'Sights' contains two map items, while the other print layout contains one map item.

To add layout extent(s):

1. Select *View* ► *Decorations* ► *Layout Extents* to open the dialog

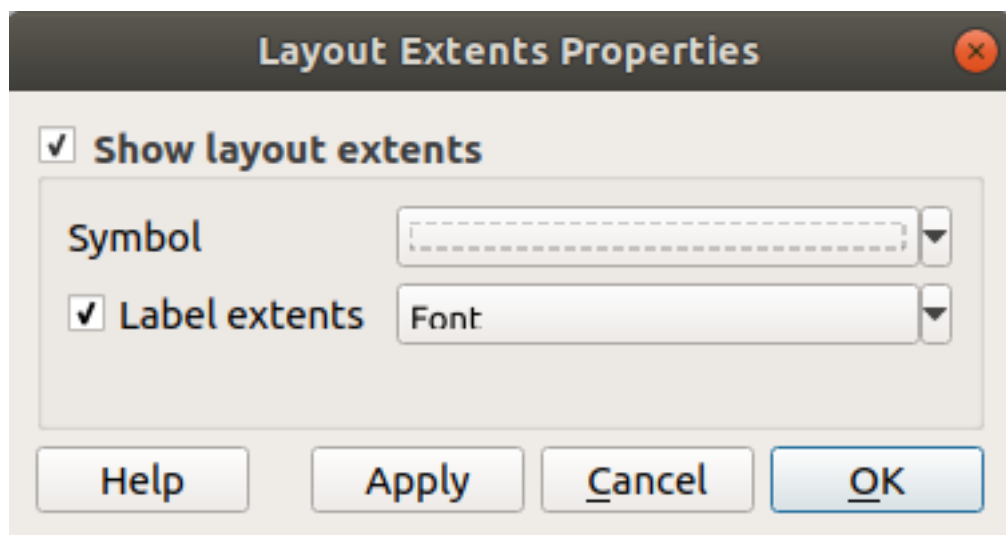


图 11.14: The Layout Extents Dialog





2. Make sure *Show layout extents* is checked.
3. Optionally change the symbol and labeling of the extents.
4. Click *Apply* to verify that it looks as expected and *OK* if you're satisfied.

小技巧: Decorations Settings

When you save a QGIS project file, any changes you have made to Grid, North Arrow, Scale Bar, Copyright and Layout Extents will be saved in the project and restored the next time you load the project.

11.4.5 Annotation Tools

Annotations are information added to the map canvas and shown within a balloon. This information can be of different types and annotations are added using the corresponding tools in the *Attributes Toolbar*:

-  Text Annotation for custom formatted text
-  HTML Annotation to place the content of an `html` file
-  SVG Annotation to add an SVG symbol
-  Form Annotation: useful to display attributes of a vector layer in a customized `ui` file (see 图 11.15). This is similar to the *custom attribute forms*, but displayed in an annotation item. Also see this video <https://www.youtube.com/watch?v=0pDBuSbQ02o&feature=youtu.be&t=2m25s> from Tim Sutton for more information.

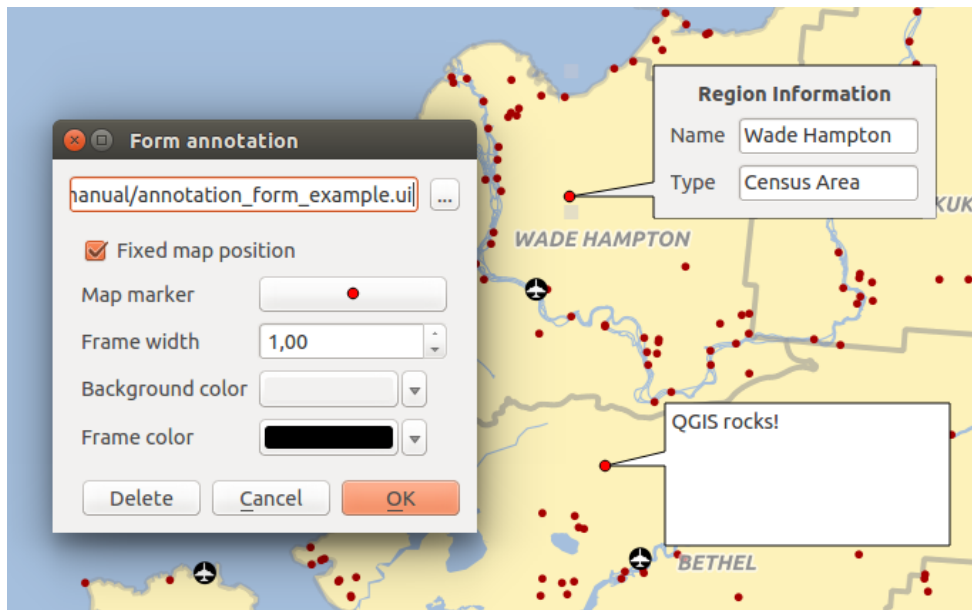


图 11.15: Customized QT Designer annotation form

To add an annotation, select the corresponding tool and click on the map canvas. An empty balloon is added. Double-click on it and a dialog opens with various options. This dialog is almost the same for all the annotation types:

- At the top, a file selector to fill with the path to an `html`, `svg` or `ui` file depending on the type of annotation. For text annotation, you can enter your message in a text box and set its rendering with the normal font tools.
- *Fixed map position*: when unchecked, the balloon placement is based on a screen position (instead of the map), meaning that it's always shown regardless the map canvas extent.
- *Linked layer*: associates the annotation with a map layer, making it visible only when that layer is visible.
- *Map marker*: using *QGIS symbols*, sets the symbol to display at the balloon anchor position (shown only when *Fixed map position* is checked).
- *Frame style*: sets the frame background color, transparency, stroke color or width of the balloon using QGIS symbols.
- *Contents margins*: sets interior margins of the annotation frame.

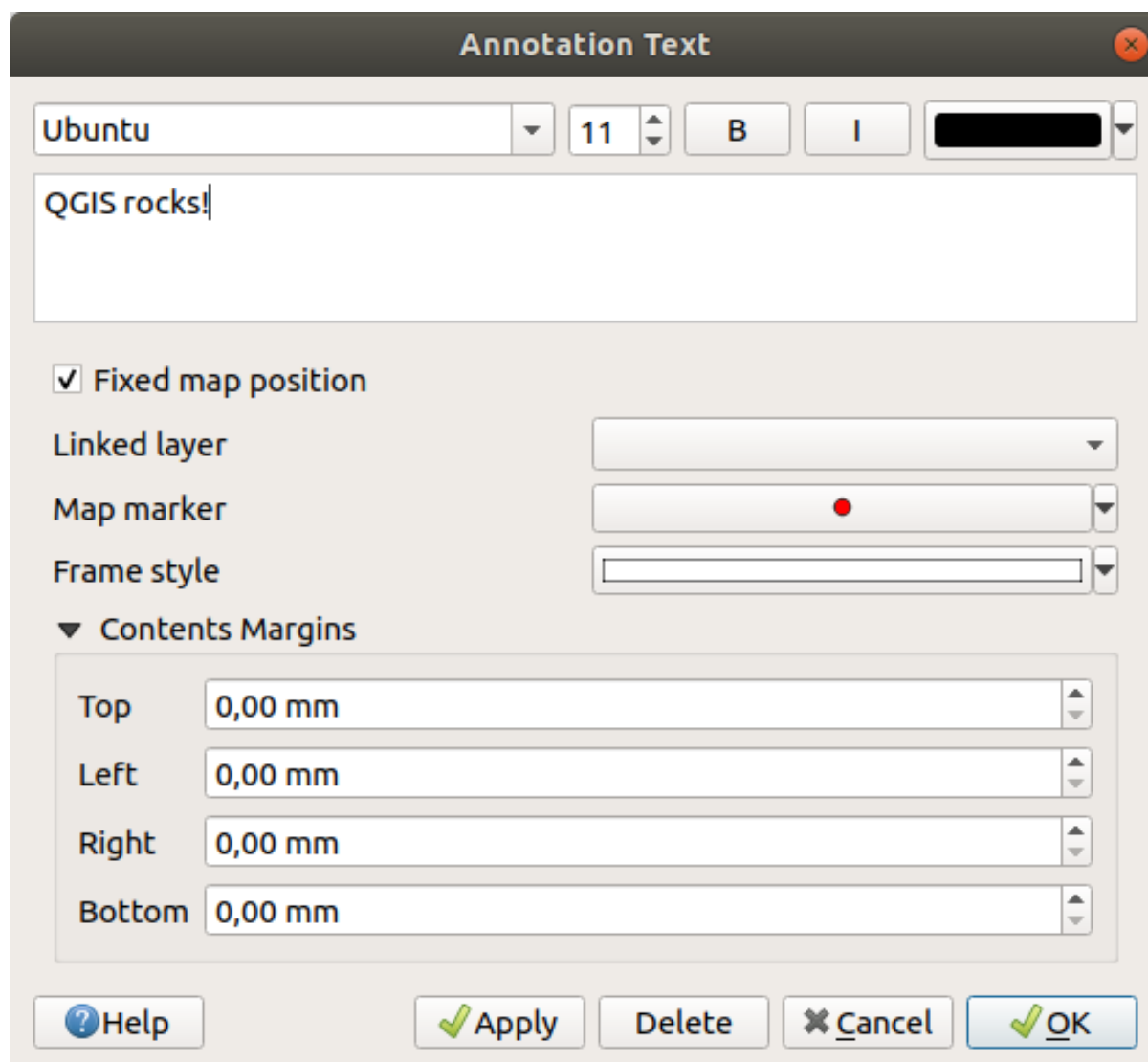



图 11.16: Annotation text dialog

Annotations can be selected when an annotation tool is enabled. They can then be moved by map position (by dragging the map marker) or by moving only the balloon. The  Move Annotation tool also allows you to move the balloon on the map canvas.

To delete an annotation, select it and either press the `Del` or `Backspace` button, or double-click it and press the *Delete* button in the properties dialog.

注解: If you press `Ctrl+T` while an *Annotation* tool (move annotation, text annotation, form annotation) is active, the visibility states of the items are inverted.

小技巧: Layout the map with annotations


You can print or export annotations with your map to various formats using:

- map canvas export tools available in the *Project* menu
 - *print layout*, in which case you need to check *Draw map canvas items* in the corresponding map item properties
-

11.4.6 Measuring

General information

QGIS provides four means of measuring geometries:





- interactive measurement tools 
- measuring in the  Field Calculator
- derived measurements in the *Identifying Features* tool
- the vector analysis tool: *Vector* ► *Geometry Tools* ► *Export/Add Geometry Columns*

Measuring works within projected coordinate systems (e.g., UTM) and unprojected data. The first three measuring tools behave equally to global project settings:

- Unlike most other GIS, the default measurement metric is ellipsoidal, using the ellipsoid defined in *Project* ► *Properties...* ► *General*. This is true both when geographic and projected coordinate systems are defined for the project.
- If you want to calculate the projected/planimetric area or distance using cartesian maths, the measurement ellipsoid has to be set to "None/Planimetric" (*Project* ► *Properties...* ► *General*). However, with a geographic (ie unprojected) CRS defined for the data and project, area and distance measurement will be ellipsoidal.

However, neither the identify tool nor the field calculator will transform your data to the project CRS before measuring. If you want to achieve this, you have to use the vector analysis tool: *Vector* ► *Geometry Tools* ► *Add Geometry Attributes...* Here, measurement is planimetric, unless you choose the ellipsoidal measurement.


Measure length, areas and angles interactively

Click the  icon in the Attribute toolbar to begin measurements. The down arrow near the icon switches between  length,  area or  angle. The default unit used in the dialog is the one set in *Project* ► *Properties...* ► *General* menu.

注解: Configuring the measure tool

While measuring length or area, clicking the *Configuration* button at the bottom of the widget opens the *Settings* ► *Options* ► *Map Tools* menu, where you can select the rubberband color, the precision of the measurements and the unit behavior. You can also choose your preferred measurement or angle units, but keep in mind that those values are overridden in the current project by the selection made in the *Project* ► *Properties...* ► *General* menu, and by the selection made in the measurement widget.

All measuring modules use the snapping settings from the digitizing module (see section *Setting the Snapping Tolerance and Search Radius*). So, if you want to measure exactly along a line feature, or around a polygon feature, first set its layer snapping tolerance. Now, when using the measuring tools, each mouse click (within the tolerance setting) will snap to that layer.

By default,  Measure Line measures real distances between given points according to a defined ellipsoid. The tool then allows you to click points on the map. Each segment length, as well as the total, shows up in the measure window. To stop measuring, click the right mouse button. Now it is possible to copy all your line measurements at once to the clipboard using the *Copy All* button.

Note that you can use the drop-down list near the total to change the measurement units interactively while working with the measure tool ('Meters', 'Kilometers', 'Feet', 'Yards', 'Miles', 'Nautical miles', 'Centimeters', 'Millimeters', 'Degrees', 'Map units'). This unit is retained for the widget until a new project is created or another project is opened.

The *Info* section in the dialog explains how calculations are made according to the CRS settings available.

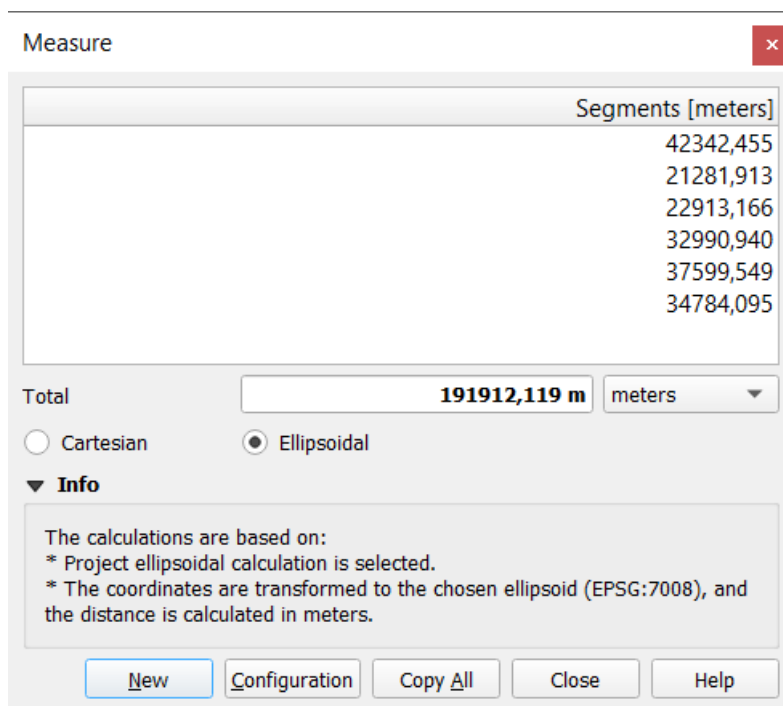



图 11.17: Measure Distance

 **Measure Area**: Areas can also be measured. In the measure window, the accumulated area size appears. Right-click to stop drawing. The Info section is also available as well as the ability to switch between different area units ('Square meters', 'Square kilometers', 'Square feet', 'Square yards', 'Square miles', 'Hectares', 'Acres', 'Square centimeters', 'Square millimeters', 'Square nautical miles', 'Square degrees', 'Map units').

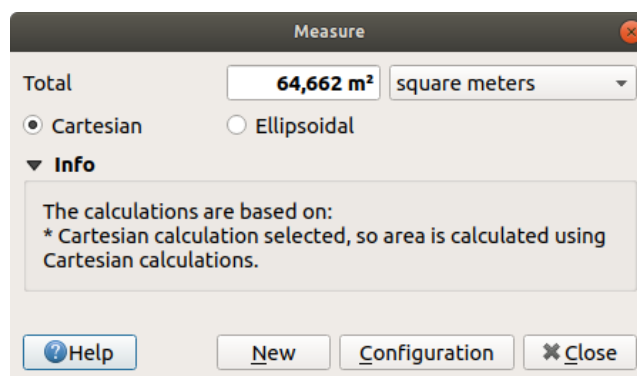



图 11.18: Measure Area

 **Measure Angle**: You can also measure angles. The cursor becomes cross-shaped. Click to draw the first segment of the angle you wish to measure, then move the cursor to draw the desired angle. The measurement is displayed in a pop-up dialog.

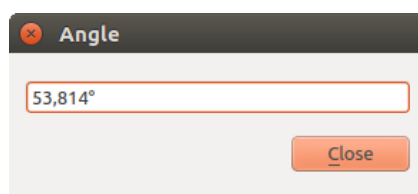


图 11.19: Measure Angle

11.5 Interacting with features





11.5.1 Selecting features


QGIS provides several tools to select features on the map canvas. Selection tools are available in the *Edit ► Select* menu or in the *Selection Toolbar*.


注解: Selection tools work with the currently active layer.

Selecting manually on the map canvas


To select one or more features with the mouse, you can use one of the following tools:

-  Select Features by area or single click
-  Select Features by Polygon
-  Select Features by Freehand
-  Select Features by Radius

注解: Other than  *Select Features by Polygon*, these manual selection tools allow you to select feature(s) on the map canvas with a single click.

注解: Use the  *Select Features by Polygon* tool to use an existing polygon feature (from any layer) to select overlapping features in the active layer. Right-click in the polygon and choose it from the context menu that shows a list of all the polygons that contain the clicked point. All the overlapping features from the active layer are selected.

小技巧: Use the *Edit ► Select ► Reselect Features* tool to redo your latest selection. Very useful when you have painstakingly made a selection, and then click somewhere else accidentally and clear your selection.








While using the  *Select Feature(s)* tool, holding *Shift* or *Ctrl* toggles whether a feature is selected (ie either adds to the current selection or remove from it).

For the other tools, different behaviors can be performed by holding down:


- *Shift*: add features to the current selection
- *Ctrl*: subtract features from the current selection
- *Ctrl+Shift*: intersect with current selection, ie only keep overlapping features from the current selection
- *Alt*: select features that are totally within the selection shape. Combined with *Shift* or *Ctrl* keys, you can add or subtract features to/from the current selection.

Automatic selection

The other selection tools, most of them available from the *Attribute table*, perform a selection based on a feature's attribute or its selection state (note that attribute table and map canvas show the same information, so if you select one feature in the attribute table, it will be selected on the map canvas too):

-  Select By Expression... select features using expression dialog
-  Select Features By Value... or press F3
-  Deselect Features from All Layers or press `Ctrl+Alt+A` to deselect all selected features in all layers
-  Deselect Features from the Current Active Layer or press `Ctrl+Shift+A`
-  Select All Features or press `Ctrl+A` to select all features in the current layer
-  Invert Feature Selection to invert the selection in the current layer
-  Select by Location to select the features based on their spatial relationship with other features (in the same or another layer - see *Select by location*)

For example, if you want to find regions that are boroughs from `regions.shp` of the QGIS sample data, you can:

1. Use the  Select features using an Expression icon
2. Expand the *Fields and Values* group
3. Double-click the field that you want to query ("TYPE_2")
4. Click *All Unique* in the panel that shows up on the right
5. From the list, double-click 'Borough'. In the *Expression* editor field, write the following query:

```
"TYPE_2" = 'Borough'
```

6. Click *Select Features*

From the expression builder dialog, you can also use *Function list* ► *Recent (Selection)* to make a selection that you have used before. The dialog remembers the last 20 expressions used. See *Expressions* for more information and examples.

小技巧: Save your selection into a new file

Users can save selected features into a **New Temporary Scratch Layer** or a **New Vector Layer** using *Edit* ► *Copy Features* and *Edit* ► *Paste Features as* in the desired format.

Select Features By Value

This selection tool opens the layer's feature form allowing the user to choose which value to look for for each field, whether the search should be case-sensitive, and the operation that should be used. The tool has also autocompletes, automatically filling the search box with existing values.

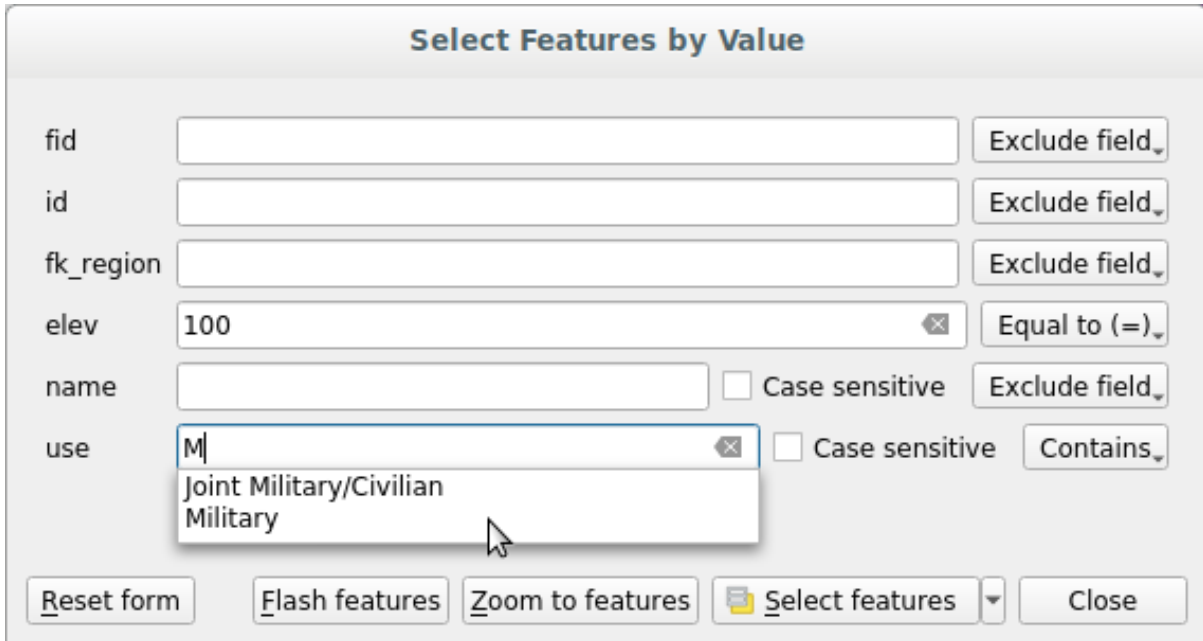


图 11.20: Filter/Select features using form dialog

Alongside each field, there is a drop-down list with options to control the search behaviour:

Field search option	String	Numeric	Date
<i>Exclude Field</i> from the search	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Equal to (=)</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Not equal to (≠)</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Greater than (>)</i>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Less than (<)</i>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Greater than or equal to (≥)</i>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Less than or equal to (≤)</i>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Between (inclusive)</i>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Not between (inclusive)</i>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Contains</i>	<input checked="" type="checkbox"/>		
<i>Does not contain</i>	<input checked="" type="checkbox"/>		
<i>Is missing (null)</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Is not missing (not null)</i>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<i>Starts with</i>	<input checked="" type="checkbox"/>		
<i>Ends with</i>	<input checked="" type="checkbox"/>		

For string comparisons, it is also possible to use the *Case sensitive* option.

After setting all search options, click *Select features* to select the matching features. The drop-down options are:

- *Select features*
- *Add to current selection*
- *Remove from current selection*
- *Filter current selection*


You can also clear all search options using the *Reset form* button.

Once the conditions are set, you can also either:


- *Zoom to features* on the map canvas without the need of a preselection
- *Flash features*, highlighting the matching features. This is a handy way to identify a feature without selection or using the Identify tool. Note that the flash does not alter the map canvas extent and would be visible only if the feature is within the bounds of the current map canvas.

11.5.2 Identifying Features

The Identify tool allows you to interact with the map canvas and get information on features in a pop-up window. To identify features, use:


- *View ► Identify Features*
- `Ctrl+Shift+I` (or **X** `Cmd+Shift+I`),
-  Identify Features icon on the Attributes toolbar

Using the Identify Features tool

QGIS offers several ways to identify features with the  Identify Features tool:

- **left click** identifies features according to the *selection mode* and the *selection mask* set in the *Identify Results* panel
- **right click** with *Identify Feature(s)* as *selection mode* set in the *Identify Results* panel fetches all snapped features from all visible layers. This opens a context menu, allowing the user to choose more precisely the features to identify or the action to execute on them.
- **right click** with *Identify Features by Polygon* as *selection mode* in the *Identify Results* panel identifies the features that overlap with the chosen existing polygon, according to the *selection mask* set in the *Identify Results* panel

小技巧: Filter the layers to query with the Identify Features tool

Under *Layer Capabilities* in *Project ► Properties...* ► *Data Sources*, uncheck the *Identifiable* column next to a layer to avoid it being queried when using the  Identify Features tool in a mode other than **Current Layer**. This is a handy way to return features from only layers that are of interest for you.

If you click on feature(s), the *Identify Results* dialog will list information about the feature(s) clicked. The default view is a tree view in which the first item is the name of the layer and its children are its identified feature(s). Each feature is described by the name of a field along with its value. This field is the one set in *Layer Properties ► Display*. All the other information about the feature follows.

Feature information

The Identify Results dialog can be customized to display custom fields, but by default it will display the following information:

- The feature *display name*;
- **Actions:** Actions can be added to the identify feature windows. The action is run by clicking on the action label. By default, only one action is added, namely *View feature form* for editing. You can define more actions in the layer's properties dialog (see *Actions Properties*).
- **Derived:** This information is calculated or derived from other information. It includes:
 - general information about the feature's geometry:

- * depending on the geometry type, the cartesian measurements of length, perimeter or area in the layer's CRS units. For 3D line vectors the cartesian line length is available.
- * depending on the geometry type and if an ellipsoid is set in the project properties dialog for *Measurements*, the ellipsoidal values of length, perimeter or area using the specified units
- * the count of geometry parts in the feature and the number of the part clicked
- * the count of vertices in the feature
- coordinate information, using the project properties *Coordinates display* settings:
 - * X and Y coordinate values of the point clicked
 - * the number of the closest vertex to the point clicked
 - * X and Y coordinate values of the closest vertex (and Z/M if applicable)
 - * if you click on a curved segment, the radius of that section is also displayed.
- **Data attributes:** This is the list of attribute fields and values for the feature that has been clicked.
- information about the related child feature if you defined a *relation*:
 - the name of the relation
 - the entry in reference field, e.g. the name of the related child feature
 - **Actions:** lists actions defined in the layer's properties dialog (see *Actions Properties*) and the default action is `View feature form`.
 - **Data attributes:** This is the list of attributes fields and values of the related child feature.

注解: Links in the feature's attributes are clickable from the *Identify Results* panel and will open in your default web browser.

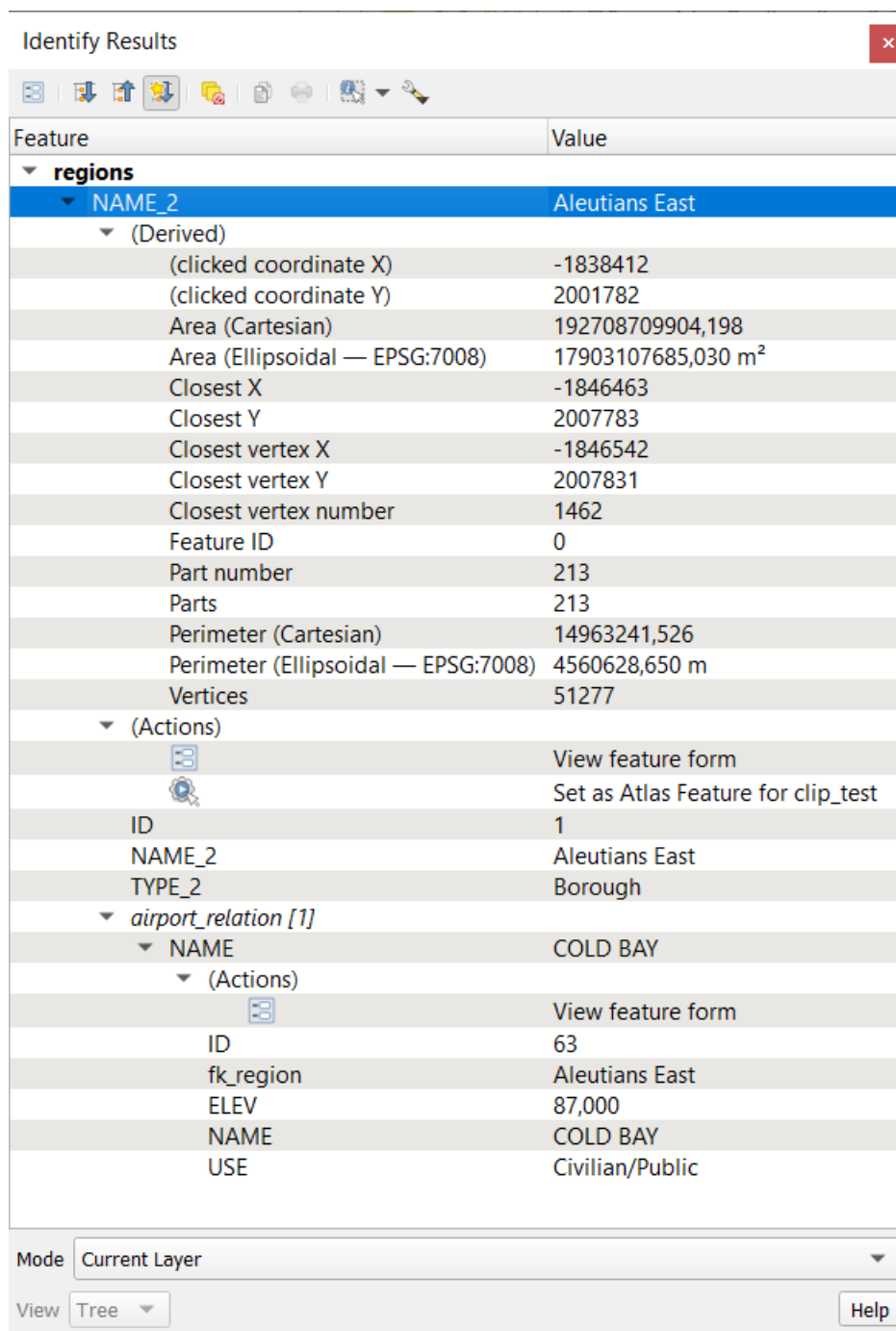





图 11.21: Identify Results dialog

The Identify Results dialog

At the top of the window, you have a handful of tools:

- Open Form of the current feature
- Expand tree
- Collapse tree
- Expand New Results by Default to define whether the next identified feature's information should be collapsed or expanded



-  Clear Results
-  Copy selected feature to clipboard
-  Print selected HTML response
- selection mode to use to fetch features to identify:
 -  Identify Features by area or single click
 -  Identify Features by Polygon
 -  Identify Features by Freehand
 -  Identify Features by Radius

注解: When using  Identify Features by Polygon, you can right-click any existing polygon and use it to identify overlapping features in another layer.

At the bottom of the window are the *Mode* and *View* combo boxes. *Mode* defines from which layers features should be identified:

- **Current layer:** only features from the selected layer are identified. The layer need not be visible in the canvas.
- **Top down, stop at first:** only features from the upper visible layer.
- **Top down:** all features from the visible layers. The results are shown in the panel.
- **Layer selection:** opens a context menu where the user selects the layer to identify features from, similar to a right-click. Only the chosen features will be shown in the result panel.

The *View* can be set as **Tree**, **Table** or **Graph**. 'Table' and 'Graph' views can only be set for raster layers.

The identify tool allows you to  *Auto open form for single feature results*, found under  Identify Settings. If checked, each time a single feature is identified, a form opens showing its attributes. This is a handy way to quickly edit a feature's attributes.

Other functions can be found in the context menu of the identified item. For example, from the context menu you can:

- View the feature form
- Zoom to feature
- Copy feature: Copy all feature geometry and attributes
- Toggle feature selection: Add identified feature to selection
- Copy attribute value: Copy only the value of the attribute that you click on
- Copy feature attributes: Copy the attributes of the feature
- Clear result: Remove results in the window
- Clear highlights: Remove features highlighted on the map
- Highlight all
- Highlight layer
- Activate layer: Choose a layer to be activated
- Layer properties: Open layer properties window
- Expand all
- Collapse all

11.6 Save and Share Layer Properties

11.6.1 Managing Custom Styles

When a vector layer is added to the map canvas, QGIS by default uses a random symbol/color to render its features. However, you can set a default symbol in *Project ► Properties... ► Default styles* that will be applied to each newly added layer according to its geometry type.

Most of the time, though, you'd rather have a custom and more complex style that can be applied automatically or manually to the layers (with less effort). You can achieve this by using the *Style* menu at the bottom of the Layer Properties dialog. This menu provides you with functions to create, load and manage styles.

A style stores any information set in the layer properties dialog to render or interact with the layer (including symbology, labeling, fields and form definitions, actions, diagrams...) for vector layers, or the pixels (band or color rendering, transparency, pyramids, histogram ...) for raster.

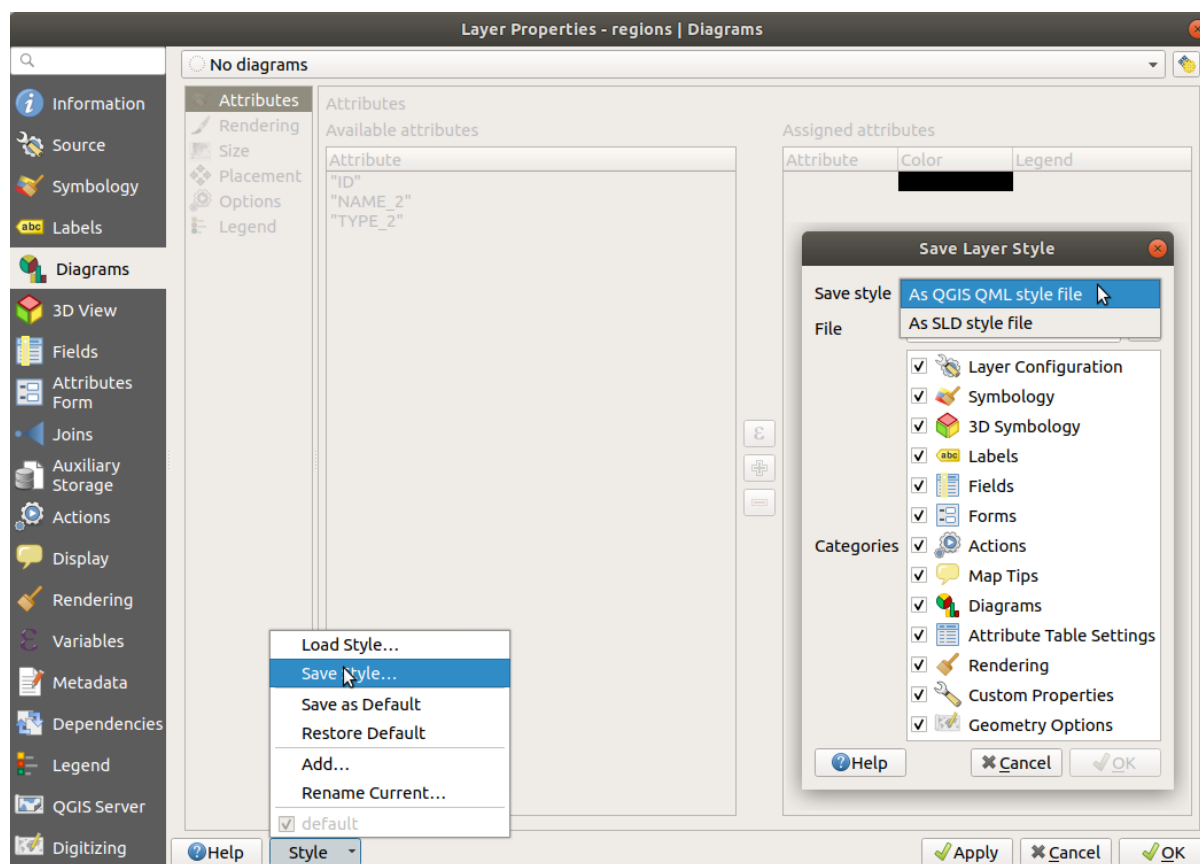



图 11.22: Vector layer style combo box options

By default, the style applied to a loaded layer is named `default`. Once you have got the ideal and appropriate rendering for your layer, you can save it by clicking the  *Style* combo box and choosing:

- **Rename Current:** The active style is renamed and updated with the current options
- **Add:** A new style is created using the current options. By default, it will be saved in the QGIS project file. See below to save the style in another file or a database
- **Remove:** Delete unwanted style, in case you have more than one style defined for the layer.

At the bottom of the Style drop-down list, you can see the styles set for the layer with the active one checked.

Note that each time you validate the layer properties dialog, the active style is updated with the changes you've made.

You can create as many styles as you wish for a layer but only one can be active at a time. In combination with *Map Themes*, this offers a quick and powerful way to manage complex projects without the need to duplicate any layer in the map legend.

注解: Given that whenever you apply modifications to the layer properties, changes are stored in the active style, always ensure you are editing the right style to avoid mistakenly altering a style used in a *map theme*.

小技巧: Manage styles from layer context menu

Right-click on the layer in the *Layers* panel to copy, paste, add or rename layer styles.

11.6.2 Storing Styles in a File or a Database

While styles created from the *Style* combo box are by default saved inside the project and can be copied and pasted from layer to layer in the project, it's also possible to save them outside the project so that they can be loaded in another project.

Save as text file

Clicking the  *Style* ► *Save Style*, you can save the style as a:

- QGIS layer style file (.qml)
- SLD file (.sld), only available for vector layers

Used on file-based format layers (.shp, .tab...), *Save as Default* generates a .qml file for the layer (with the same name). SLDs can be exported from any type of renderer -- single symbol, categorized, graduated or rule-based -- but when importing an SLD, either a single symbol or rule-based renderer is created. This means that categorized or graduated styles are converted to rule-based. If you want to preserve those renderers, you have to use the QML format. On the other hand, it can be very handy sometimes to have this easy way of converting styles to rule-based.

Save in database

Vector layer styles can also be stored in a database if the layer datasource is a database provider. Supported formats are PostGIS, GeoPackage, SpatiaLite, MSSQL and Oracle. The layer style is saved inside a table (named *layer_styles*) in the database. Click on *Save Style...* ► *Save in database* then fill in the dialog to define a style name, add a description, a .ui file if applicable and to check if the style should be the default style.

You can save several styles for a single table in the database. However, each table can have only one default style. Default styles can be saved in the layer database or in *qgis.db*, a local SQLite database in the active *user profile* directory.

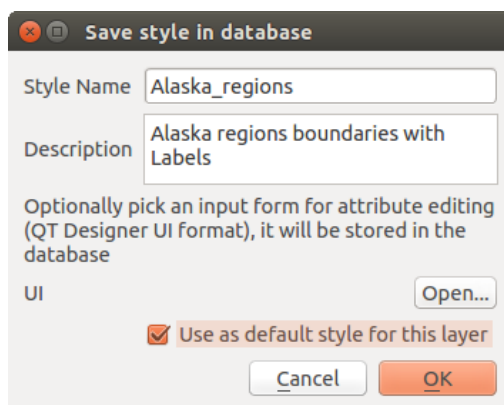


图 11.23: Save Style in database Dialog

小技巧: Sharing style files between databases

You can only save your style in a database if the layer comes from such a database. You can't mix databases (layer in Oracle and style in MSSQL for instance). Use instead a plain text file if you want the style to be shared among databases.

注解: You may encounter issues restoring the `layer_styles` table from a PostgreSQL database backup. Follow [QGIS layer_style table and database backup](#) to fix that.

Load style

When loading a layer in QGIS, if a default style already exists for this layer, QGIS loads the layer with this style. Also *Style ► Restore Default* looks for and loads that file, replacing the layer's current style.

Style ► Load Style helps you apply any saved style to a layer. While text-file styles (`.sld` or `.qml`) can be applied to any layer whatever its format, loading styles stored in a database is only possible if the layer is from the same database or the style is stored in the QGIS local database.

The *Database Styles Manager* dialog displays a list of styles related to the layer found in the database and all the other styles saved in it, with name and description.

小技巧: Quickly share a layer style within the project

You can also share layer styles within a project without importing a file or database style: right-click on the layer in the *Layers Panel* and, from the *Styles* combo box, copy the style of a layer and paste it to a group or a selection of layers: the style is applied to all the layers that are of the same type (vector vs raster) as the original layer and, in the case of vector layers, have the same geometry type (point, line or polygon).

11.6.3 Layer definition file

Layer definitions can be saved as a Layer Definition File (.qlr) using *Export ► Save As Layer Definition File...* in the active layers' context menu. A layer definition file (.qlr) includes references to the data source of the layers and their styles. .qlr files are shown in the Browser Panel and can be used to add the layers (with the saved style) to the Layers Panel. You can also drag and drop .qlr files from the system file manager into the map canvas.

11.7 Storing values in Variables

In QGIS, you can use variables to store useful recurrent values (e.g. the project's title, or the user's full name) that can be used in expressions. Variables can be defined at the application's global level, project level, layer level, layout level, and layout item's level. Just like CSS cascading rules, variables can be overwritten - e.g., a project level variable will overwrite any application global level variables set with the same name. You can use these variables to build text strings or other custom expressions using the @ character before the variable name. For example in print layout creating a label with this content:

```
This map was made using QGIS [% @qgis_version %]. The project file for this
map is: [% @project_path %]
```

Will render the label like this:

```
This map was made using QGIS 3.4.4-Madeira. The project file for this map is:
/gis/qgis-user-conference-2019.qgs
```

Besides the *preset read-only variables*, you can define your own custom variables for any of the levels mentioned above. You can manage:

- **global variables** from the *Settings ► Options* menu
- **project variables** from the *Project Properties* dialog (see 工程屬性)
- **vector layer variables** from the *Layer Properties* dialog (see *The Vector Properties Dialog*);
- **layout variables** from the *Layout* panel in the Print layout (see *The Layout Panel*);
- and **layout item variables** from the *Item Properties* panel in the Print layout (see *Layout Items Common Options*).

To differentiate from editable variables, read-only variable names and values are displayed in italic. On the other hand, higher level variables overwritten by lower level ones are strike through.

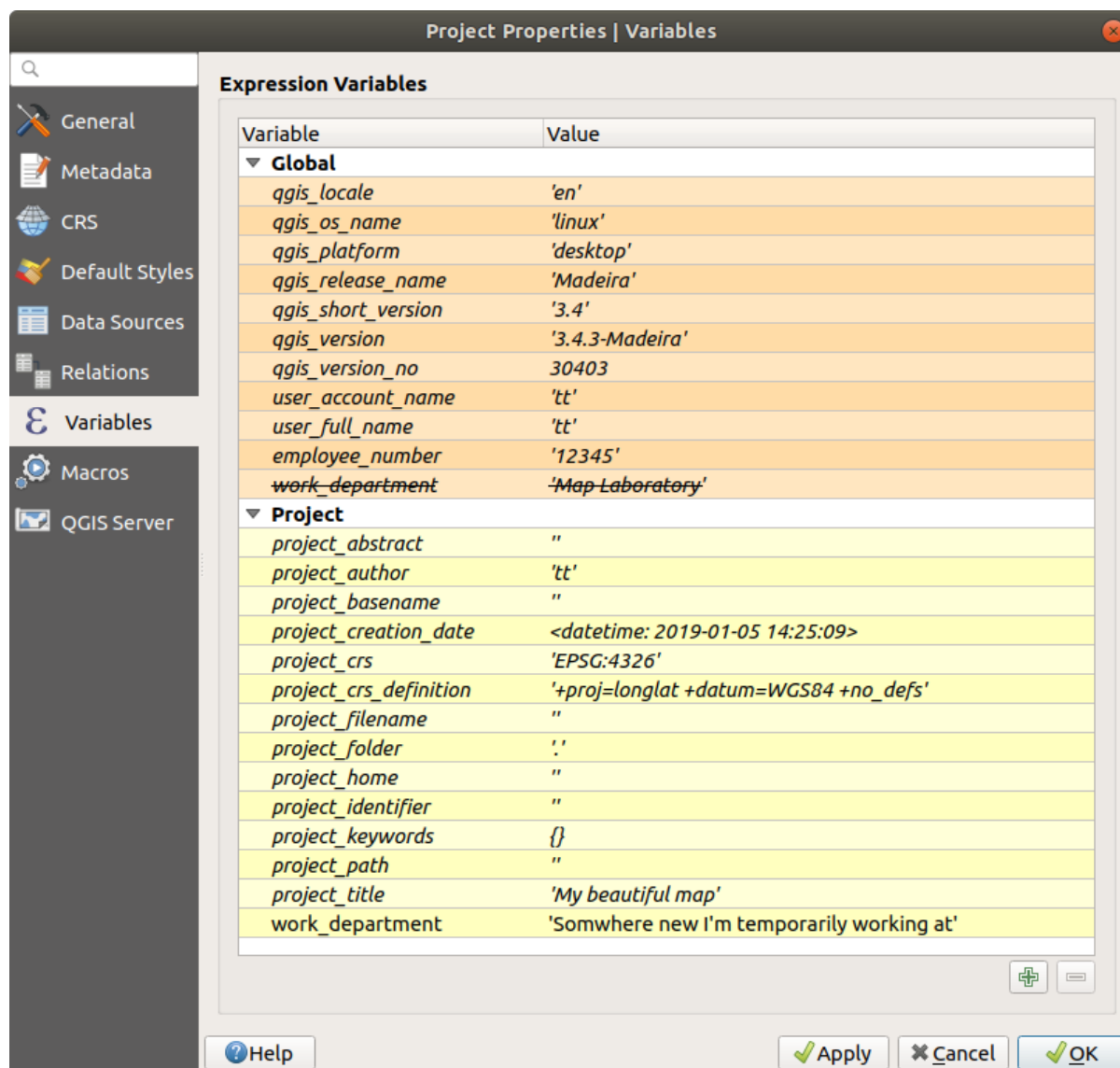


图 11.24: Variables editor at the project level

注解: You can read more about variables and find some examples in Nyal Dawson's [Exploring variables in QGIS 2.12, part 1](#), [part 2](#) and [part 3](#) blog posts.

11.8 Authentication

QGIS has the facility to store/retrieve authentication credentials in a secure manner. Users can securely save credentials into authentication configurations, which are stored in a portable database, can be applied to server or database connections, and are safely referenced by their ID tokens in project or settings files. For more information see [认证系统](#).


A master password needs to be set up when initializing the authentication system and its portable database.





11.9 Common widgets

In QGIS, there are some options you'll often have to work with. For convenience, QGIS provides you with special widgets that are presented below.

11.9.1 Color Selector

The color dialog

The *Select Color* dialog will appear whenever you click the  icon to choose a color. The features of this dialog depend on the state of the *Use native color chooser dialogs* parameter checkbox in *Settings* ► *Options...* ► *General*. When checked, the color dialog used is the native one of the OS on which QGIS is running. Otherwise, the QGIS custom color chooser is used.

The custom color chooser dialog has four different tabs which allow you to select colors by  Color ramp,  Color wheel,  Color swatches or  Color picker. With the first two tabs, you can browse to all possible color combinations and apply your choice to the item.

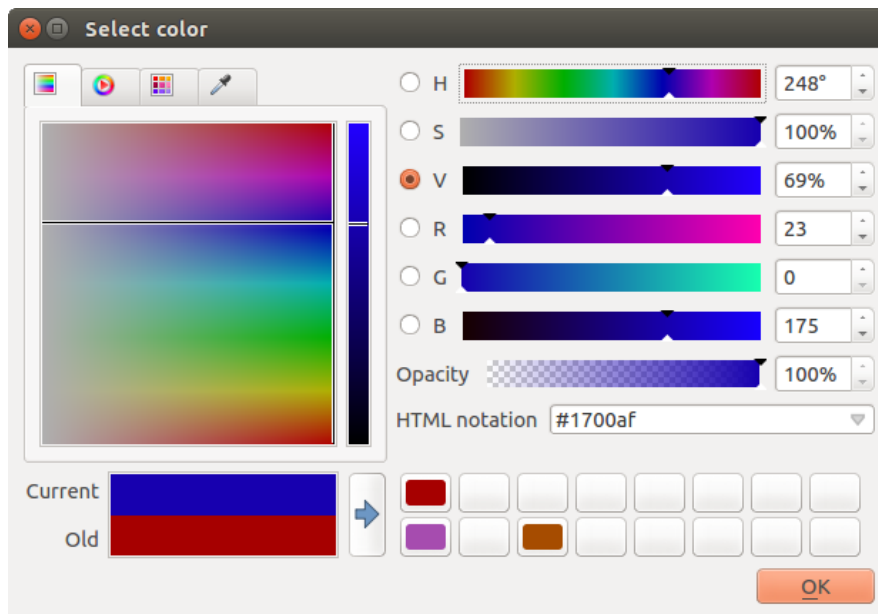





图 11.25: Color selector ramp tab

In the  Color swatches tab, you can choose from a list of color palettes (see 颜色设置 for details). All but the *Recent colors* palette can be modified with the  Add current color and  Remove selected color buttons at the bottom of the frame.

The ... button next to the palette combo box also offers several options to:

- copy, paste, import or export colors
- create, import or remove color palettes
- add the custom palette to the color selector widget with the *Show in Color Buttons* item (see 图 11.27)

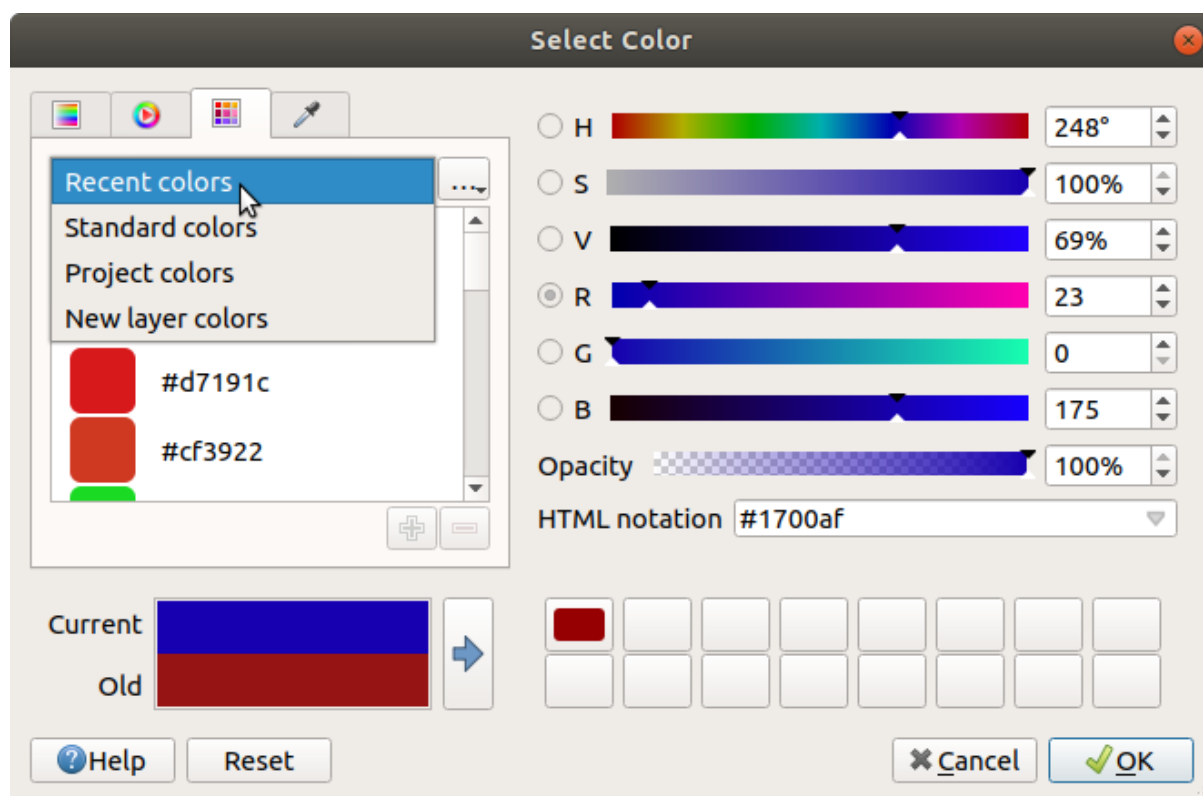




图 11.26: Color selector swatches tab

Another option is to use the  *Color picker* which allows you to sample a color from under your mouse cursor at any part of the QGIS UI or even from another application: press the space bar while the tab is active, move the mouse over the desired color and click on it or press the space bar again. You can also click the *Sample Color* button to activate the picker.

Whatever method you use, the selected color is always described through color sliders for HSV (Hue, Saturation, Value) and RGB (Red, Green, Blue) values. The color is also identifiable in *HTML notation*.

Modifying a color is as simple as clicking on the color wheel or ramp or on any of the color parameters sliders. You can adjust such parameters with the spinbox beside or by scrolling the mouse wheel over the corresponding slider. You can also type the color in HTML notation. Finally, there is an *Opacity* slider to set transparency level.

The dialog also provides a visual comparison between the *Old* color (applied to object) and the *Current* one (being selected). Using drag-and-drop or pressing the  *Add color to swatch* button, any of these colors can be saved in a slot for easy access.

小技巧: Quick color modification

Drag-and-drop a color selector widget onto another one to apply its color.

The color drop-down shortcut

Click the drop-down arrow to the right of the  color button to display a widget for quick color selection. This shortcut provides access to:

- a color wheel to pick a color from
- an alpha slider to change color opacity
- the color palettes previously set to *Show in Color Buttons*
- copy the current color and paste it into another widget
- pick a color from anywhere on your computer display
- choose a color from the color selector dialog
- drag-and-drop the color from one widget to another for quick modification

注解: When the color widget is set to a *project color* through the data-defined override properties, the above functions for changing the color are unavailable. You'd first need to *Unlink color* or *Clear* the definition.

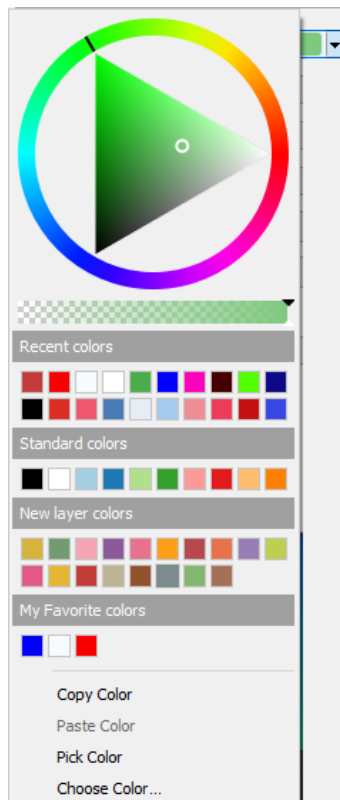



图 11.27: Quick color selector menu

The color ramp drop-down shortcut

Color ramps are a practical way to apply a set of colors to one or many features. Their creation is described in the *Setting a Color Ramp* section. As for the colors, pressing the  color ramp button opens the corresponding color ramp type dialog allowing you to change its properties.

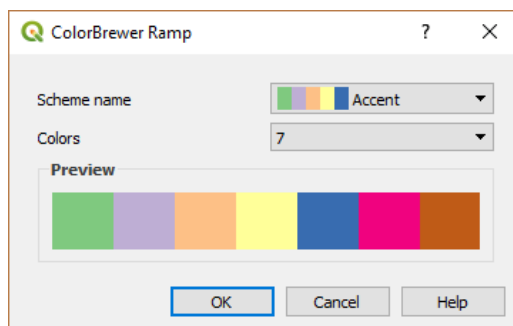


图 11.28: Customizing a colorbrewer ramp

The drop-down menu to the right of the button gives quick access to a wider set of color ramps and options:

- *Invert Color Ramp*
- a preview of the gradient or catalog: `cpt-city` color ramps flagged as **Favorites** in the *Style Manager* dialog
- *All Color Ramps* to access the compatible color ramps database
- *Create New Color Ramp...* of any supported type that could be used in the current widget (note that this color ramp will not be available elsewhere unless you save it in the library)
- *Edit Color Ramp...*, the same as clicking the whole color ramp button
- *Save Color Ramp...*, to save the current color ramp with its customizations in the style library

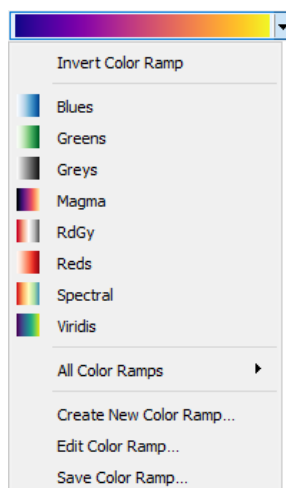


图 11.29: Quick color ramp selection widget

11.9.2 Symbol Widget

The *Symbol* selector widget is a convenient shortcut when you want to set symbol properties of a feature. Clicking the drop-down arrow shows the following symbol options, together with the features of the *color drop-down widget*:

- *Configure Symbol...*: the same as pressing the symbol selector widget. It opens a dialog to set the *symbol parameters*.
- *Copy Symbol* from the current item
- *Paste Symbol* to the current item, speeding configuration

11.9.3 Font Selector

The *Font* selector widget is a convenient shortcut when you want to set font properties for textual information (feature labels, decoration labels, map legend text, ...). Clicking the drop-down arrow shows some or all of the following options:




图 11.30: Font selector drop-down menu

- *Font Size* in the associated unit
- *Recent Fonts* ► menu with the active font checked (at the top)
- *Configure Format...*: same as pressing the font selector widget. It opens a dialog to set text format parameters. Depending on the context, it can be the OS default *Text format* dialog or the QGIS custom dialog with advanced formatting options (opacity, orientation, buffer, background, shadow, ...) as described in section *Formatting the label text*.
- *Copy Format* of the text
- *Paste Format* to the text, speeding configuration
- the *color widget* for quick color setting

11.9.4 Unit Selector

Size properties of the items (labels, symbols, layout elements, ...) in QGIS are not necessarily bound to either the project units or the units of a particular layer. For a large set of properties, the *Unit* selector drop-down menu allows you to tweak their values according to the rendering you want (based on screen resolution, paper size, or the terrain). Available units are:

- *Millimeters*
- *Points*
- *Pixels*
- *Inches*
- *Meters at Scale*: This allows you to always set the size in meters, regardless of what the underlying map units are (e.g. they can be in inches, feet, geographic degrees, ...). The size in meters is calculated based on the current project ellipsoid setting and a projection of the distances in meters at the center of the current map extent.
- and *Map Units*: The size is scaled according to the map view scale. Because this can lead to too big or too small values, use the  button next to the entry to constrain the size to a range of values based on:
 - The *Minimum scale* and the *Maximum scale*: The value is scaled based on the map view scale until you reach any of these scale limits. Out of the range of scale, the value at the nearest scale limit is kept.
 - and/or The *Minimum size* and the *Maximum size* in mm: The value is scaled based on the map view scale until it reaches any of these limits; Then the limit size is kept.

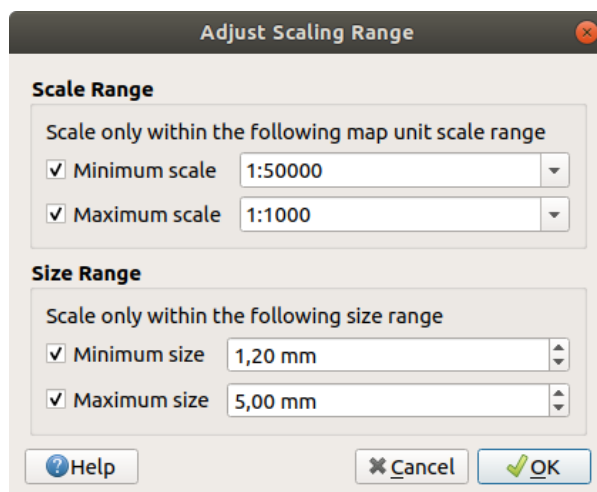


图 11.31: Adjust scaling range dialog

11.9.5 Number Formatting

Numeric formatters allow formatting of numeric values for display, using a variety of different formatting techniques (for instance scientific notation, currency values, percentage values, etc). One use of this is to set text in a layout scale bar or fixed table.

Different categories of formats are supported. For most of them, you can set part or all of the following numeric options:

- *Show thousands separator*
- *Show plus sign*
- *Show trailing zeros*

But they can also have their custom settings. Provided categories are:

- *General*, the default category: has no setting and displays values as set in the parent widget properties or using the global settings.
- *Number*
 - The value can be *Round to* a self defined number of *Decimal places* or their *Significant figures*
 - customize the *Thousands separator* and *Decimal separator*
- *Bearing* for a text representation of a direction/bearing using:
 - *Format*: possible ranges of values are 0 to 180°, with E/W suffix, -180 to +180° and 0 to 360°
 - number of *Decimal places*
- *Currency* for a text representation of a currency value.
 - *Prefix*
 - *Suffix*
 - number of *Decimal places*
- *Fraction* for a vulgar fractional representation of a decimal value (e.g. 1/2 instead of 0.5)
 - Use unicode super/subscript to show. For example 1/2 instead of 1/2
 - Use dedicated Unicode characters
 - customize the *Thousands separator*
- *Percentage* - appends % to the values, with setting of:
 - number of *Decimal places*
 - *Scaling* to indicate whether the actual values already represent percentages (then they will be kept as is) or fractions (then they are converted)
- *Scientific* notation in the form 2.56e+03. The number of *Decimal places* can be set.

A live preview of the settings is displayed under the *Sample* section.


11.9.6 Blending Modes

QGIS offers different options for special rendering effects with these tools that you may previously only know from graphics programs. Blending modes can be applied on layers and features, and also on print layout items:


- **Normal**: This is the standard blend mode, which uses the alpha channel of the top pixel to blend with the pixel beneath it. The colors aren't mixed.
- **Lighten**: This selects the maximum of each component from the foreground and background pixels. Be aware that the results tend to be jagged and harsh.
- **Screen**: Light pixels from the source are painted over the destination, while dark pixels are not. This mode is most useful for mixing the texture of one item with another item (such as using a hillshade to texture another layer).
- **Dodge**: Brighten and saturate underlying pixels based on the lightness of the top pixel. Brighter top pixels cause the saturation and brightness of the underlying pixels to increase. This works best if the top pixels aren't too bright. Otherwise the effect is too extreme.
- **Addition**: Adds pixel values of one item to the other. In case of values above the maximum value (in the case of RGB), white is displayed. This mode is suitable for highlighting features.
- **Darken**: Retains the lowest values of each component of the foreground and background pixels. Like lighten, the results tend to be jagged and harsh.

- **Multiply:** Pixel values of the top item are multiplied with the corresponding values for the bottom item. The results are darker.
- **Burn:** Darker colors in the top item cause the underlying items to darken. Burn can be used to tweak and colorize underlying layers.
- **Overlay:** Combines multiply and screen blending modes. Light parts become lighter and dark parts become darker.
- **Soft light:** Very similar to overlay, but instead of using multiply/screen it uses color burn/dodge. This is supposed to emulate shining a soft light onto an image.
- **Hard light:** Hard light is also very similar to the overlay mode. It's supposed to emulate projecting a very intense light onto an image.
- **Difference:** Subtracts the top pixel from the bottom pixel, or the other way around, in order always to get a positive value. Blending with black produces no change, as the difference with all colors is zero.
- **Subtract:** Subtracts pixel values of one item from the other. In the case of negative values, black is displayed.

11.9.7 Data defined override setup





Next to many options in the vector layer properties dialog or settings in the print layout, you will find a  Data defined override icon. Using *expressions* based on layer attributes or item settings, prebuilt or custom functions and *variables*, this tool allows you to set dynamic values for parameters. When enabled, the value returned by this widget is applied to the parameter regardless of its normal value (checkbox, textbox, slider...).


The data defined override widget

Clicking the  Data defined override icon shows the following entries:


- *Description...* that indicates if the option is enabled, which input is expected, the valid input type and the current definition. Hovering over the widget also pops up this information.
- *Store data in the project:* a button allowing the property to be stored using to the *Auxiliary Storage Properties* mechanism.
- *Field type:* an entry to select from the layer's fields that match the valid input type.
- *Color:* when the widget is linked to a color property, this menu gives access to the colors defined as part of the current *project's colors* scheme.
- *Variable:* a menu to access the available user-defined *variables*
- *Edit...* button to create or edit the expression to apply, using the *Expression String Builder* dialog. To help you correctly fill in the expression, a reminder of the expected output's format is provided in the dialog.
- *Paste* and *Copy* buttons.
- *Clear* button to remove the setup.
- For numeric and color properties, *Assistant...* to rescale how the feature data is applied to the property (more details *below*)



小技巧: Use right-click to (de)activate the data override

When the data-defined override option is set up correctly the icon is yellow  or . If it is broken, the icon is red  or .

You can enable or disable a configured  data-defined override button by simply clicking the widget with the right mouse button.

Using the data-defined assistant interface

When the  Data-defined override button is associated with a numeric or color parameter, it has an *Assistant...* option that allows you to change how the data is applied to the parameter for each feature. The assistant allows you to:

- Define the *Input* data, ie:
 - the attribute to represent, using the Field listbox or the  Set column expression function (see *Expressions*)
 - the range of values to represent: you can manually enter the values or use the  Fetch value range from layer button to fill these fields automatically with the minimum and maximum values returned by the chosen attribute or the expression applied to your data
- *Apply transform curve*: by default, output values (see below for setting) are applied to input features following a linear scale. You can override this logic: enable the transform option, click on the graphic to add break point(s) and drag the point(s) to apply a custom distribution.
- Define the *Output* values: the options vary according to the parameter to define. You can globally set:
 - the minimum and maximum values to apply to the selected property (in case of a color setting, you'll need to provide a *color ramp*)
 - the *Scale method* of representation which can be **Flannery**, **Exponential**, **Surface** or **Radius**
 - the *Exponent* to use for data scaling
 - the output value or *color* to represent features with NULL values

When compatible with the property, a live-update preview is displayed in the right-hand side of the dialog to help you control the value scaling.

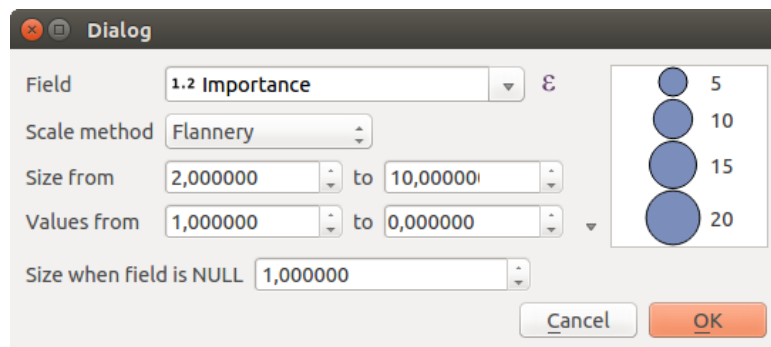


图 11.32: The data-defined size assistant

The values presented in the varying size assistant above will set the size 'Data-defined override' with:




```
coalesce(scale_exp(Importance, 1, 20, 2, 10, 0.57), 1)
```


12.1 The Style Manager

12.1.1 The Style Manager dialog

The *Style Manager* is the place where you can manage and create generic style items. These are symbols, color ramps, text formats or label settings that can be used to symbolize features, layers or print layouts. They are stored in the `symbology-style.db` database under the active *user profile* and shared with all the project files opened with that profile. Style items can also be shared with others thanks to the export/import capabilities of the *Style Manager* dialog.

You can open that modeless dialog either:

- from the *Settings* ►  *Style Manager...* menu
- with the  *Style Manager* button from the Project toolbar
- or with the  *Style Manager* button from a vector *Layer Properties* ► menu (while *configuring a symbol* or *formatting a text*).

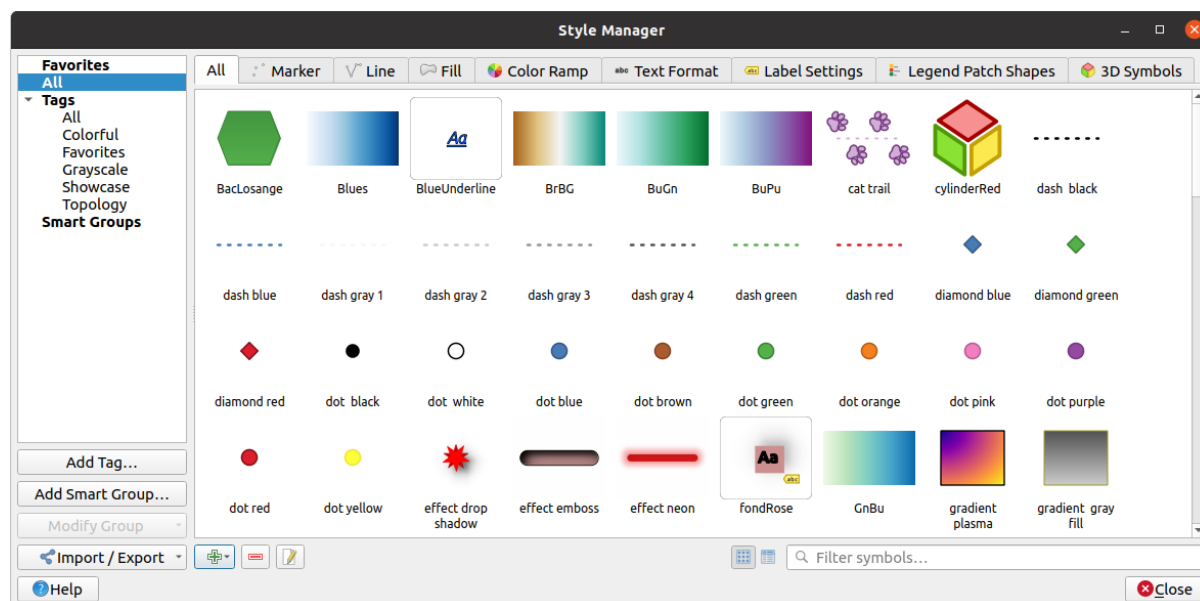










图 12.1: The Style Manager

Organizing style items

The *Style Manager* dialog displays in its center a frame with previewed items organized into tabs:

- *All* for a complete collection of point, linear and surface symbols and label settings as well as predefined color ramps and text formats;
-  *Marker* for point symbols only;
-  *Line* for linear symbols only;
-  *Fill* for surface symbols only;
-  *Color ramp*;
-  *Text format* to manage *text formats*, which store the font, color, buffers, shadows, and backgrounds of texts (i.e. all the formatting parts of the label settings, which for instance can be used in layouts);
-  *Label settings* to manage *label settings*, which include the text formats and some layer-type specific settings such as label placement, priority, callouts, rendering...
-  *3D Symbols* to configure symbols with *3D properties* (extrusion, shading, altitude, ...) for the features to render in a *3D Map view*

For each family of items, you can organize the elements into different categories, listed in the panel on the left:

- **Favorites:** displayed by default when configuring an item, it shows an extensible set of items;
- **All:** lists all the available items for the active type;
- **Tags:** shows a list of labels you can use to identify the items. An item can be tagged more than once. Select a tag in the list and the tabs are updated to show only their items that belong to it. To create a new tag you could later attach to a set of items, use the *Add Tag...* button or select the  *Add Tag...* from any tag contextual menu;
- **Smart Group:** a smart group dynamically fetches its symbols according to conditions set (see eg, 图 12.2). Click the *Add Smart Group...* button to create smart groups. The dialog box allows you to enter an expression

to filter the items to select (has a particular tag, have a string in its name, etc.). Any symbol, color ramp, text format or label setting that satisfies the entered condition(s) is automatically added to the smart group.

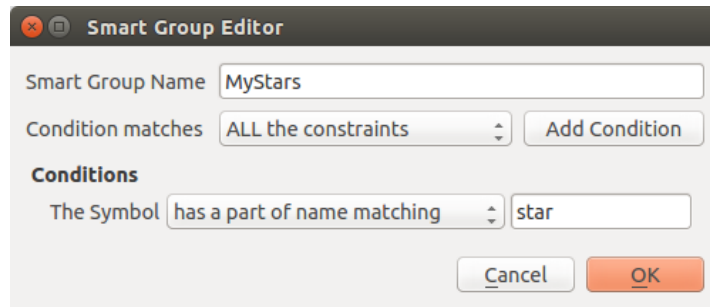



图 12.2: Creating a Smart Group




Tags and smart groups are not mutually exclusive: they are simply two different ways to organize your style elements. Unlike the smart groups that automatically fetch their belonged items based on the input constraints, tags are filled by the user. To edit any of those categories, you can either:

- select the items, right-click and choose *Add to Tag* ► and then select the tag name or create a new tag;
- select the tag and press *Modify group...* ► *Attach Selected Tag to Symbols*. A checkbox appears next to each item to help you select or deselect it. When selection is finished, press *Modify group...* ► *Finish Tagging*.
- select the smart group, press *Modify group...* ► *Edit smart group...* and configure a new set of constraints in the *Smart Group Editor* dialog. This option is also available in the contextual menu of the smart group.

To remove a tag or a smart group, right-click on it and select the  *Remove* button. Note that this does not delete the items grouped in the category.

Adding, editing or removing an item

As seen earlier, style elements are listed under different tabs whose contents depend on the active category (tag, smart group, favorites...). When a tab is enabled, you can:

- Add new items: press the  *Add item* button and configure the item following *symbols*, *color ramps* or *text format and label* builder description.
- Modify an existing item: select an item and press  *Edit item* button and configure as mentioned above.
- Delete existing items: to delete an element you no longer need, select it and click  *Remove item* (also available through right-click). The item will be deleted from the local database.


Note that the *All* tab provides access to these options for every type of item.

Right-clicking over a selection of items also allows you to:

- *Add to Favorites*;
- *Remove from Favorites*;
- *Add to Tag* ► and select the appropriate tag or create a new one to use; the currently assigned tags are checked;
- *Clear Tags*: detaching the symbols from any tag;
- *Remove Item(s)*;
- *Edit Item*: applies to the item you right-click over;
- *Copy Item*;
- *Paste Item ...*: pasting to one of the categories of the style manager or elsewhere in QGIS (symbol or color buttons)



- *Export Selected Symbol(s) as PNG...* (only available with symbols);
- *Export Selected Symbol(s) as SVG...* (only available with symbols);

Sharing style items

The  *Import/Export* tool, at the left bottom of the Style Manager dialog, offers options to easily share symbols, color ramps, text formats and label settings with others. These options are also available through right-click over the items.

Exporting items

You can export a set of items to an .XML file:

1. Expand the  *Import/Export* drop-down menu and select  *Export Item(s)...*
2. Choose the items you'd like to integrate. Selection can be done with the mouse or using a tag or a group previously set.
3. Press *Export* when ready. You'll be prompted to indicate the destination of the saved file. The XML format generates a single file containing all the selected items. This file can then be imported in another user's style library.

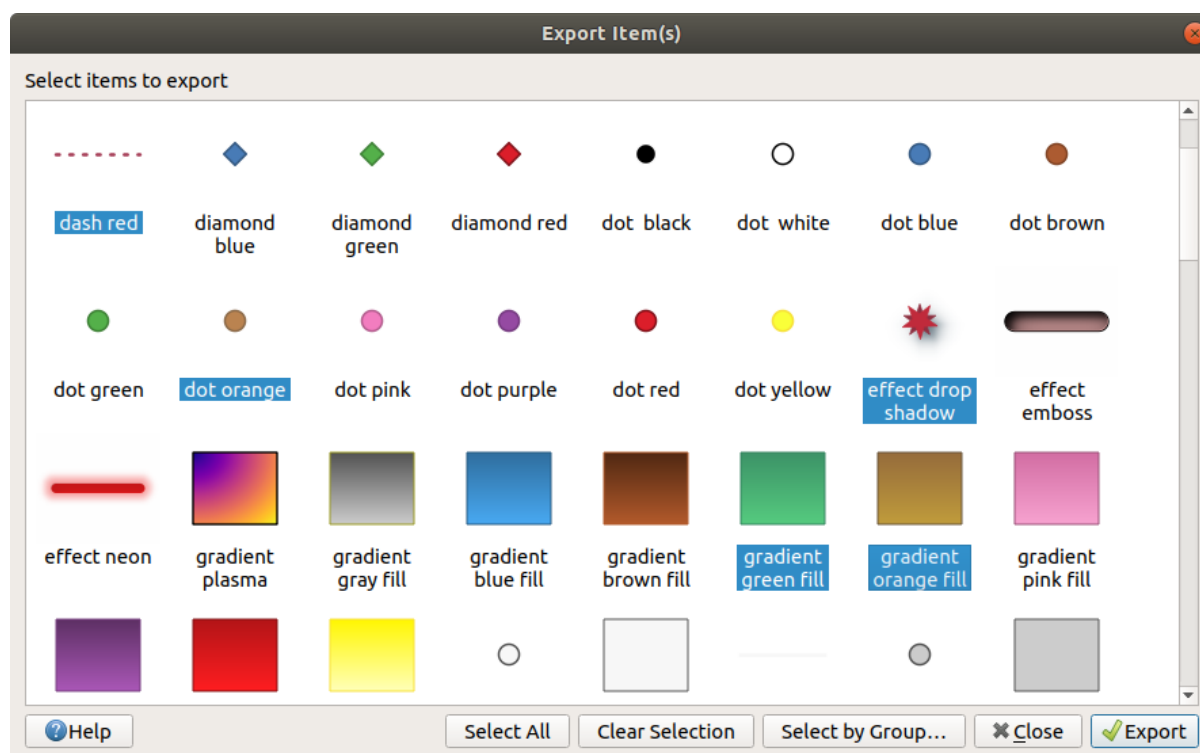




图 12.3: Exporting style items

When symbols are selected, you can also export them to .PNG or .SVG. Exporting to .PNG or .SVG (both not available for other style item types) creates a file for each selected symbol in a given folder. The SVG folder can be added to the *SVG paths* in *Settings* ► *Options* ► *System* menu of another user, allowing him direct access to all these symbols.

Importing items

You can extend your style library by importing new items:

1. Expand the  *Import/Export* drop-down menu and select  *Import Item(s)* at the left bottom of the dialog.
2. In the new dialog, indicate the source of the style items (it can be an `.xml` file on the disk or a url).
3. Set whether to *Add to favorites* the items to import.
4. Check *Do not import embedded tags* to avoid the import of tags associated to the items being imported.
5. Give the name of any *Additional tag(s)* to apply to the new items.
6. Select from the preview the symbols you want to add to your library.
7. And press *Import*.

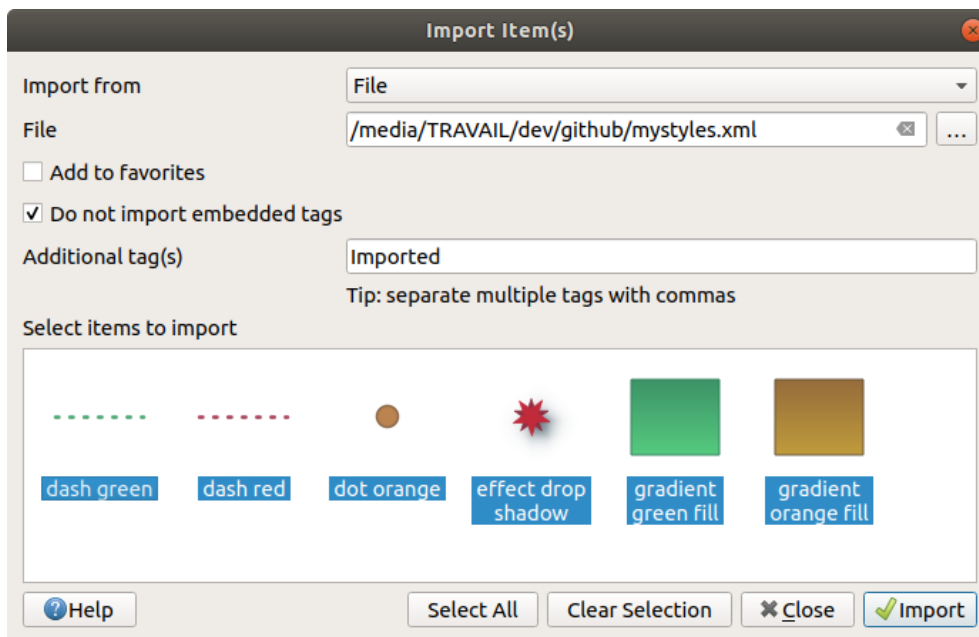


图 12.4: Importing style items

Using the Browser panel

It's also possible to import style items into the active user profile style database directly from the *Browser* panel:

1. Select the style `.xml` file in the browser
2. Drag-and-drop it over the map canvas or right-click and select *Import Style...*
3. Fill the *Import Items* dialog following *Importing items*
4. Press *Import* and the selected style items are added to the style database

Double-clicking the style file in the browser opens the *Style Manager* dialog showing the items in the file. You can select them and press *Copy to Default Style...* to import them into the active style database. Tags can be assigned to items. Also available through right-click, *Open Style...* command.

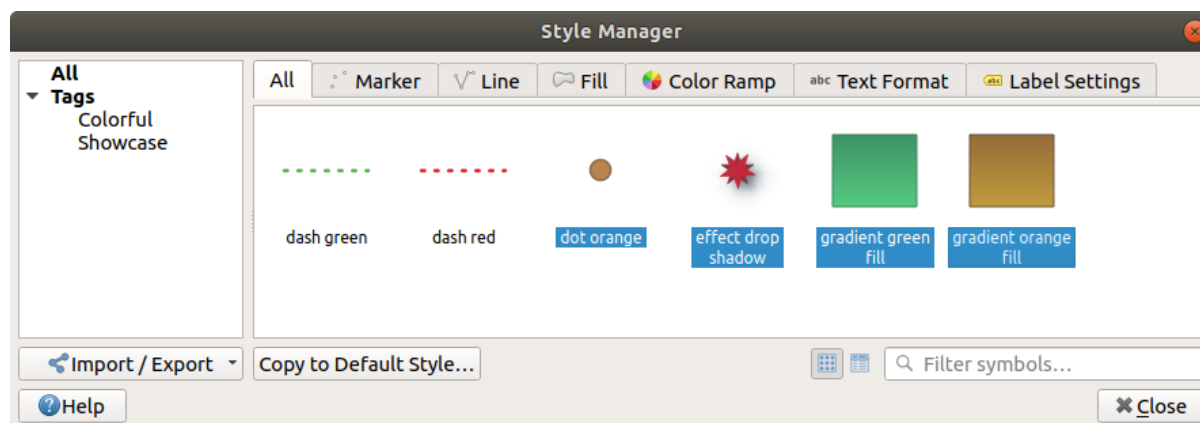



图 12.5: Opening a style items file

The dialog also allows to export single symbols as .PNG or .SVG files.

12.1.2 Setting a Color Ramp

The Color ramp tab in the *Style Manager* dialog helps you preview different color ramps based on the category selected in the left panel.

To create a custom color ramp, activate the Color ramp tab and click the  Add item button. The button reveals a drop-down list to choose the ramp type:

- *Gradient*: given a start and end colors, generate a color ramp which can be **continuous** or **discrete**. With double-clicking the ramp preview, you can add as many intermediate color stops as you want.

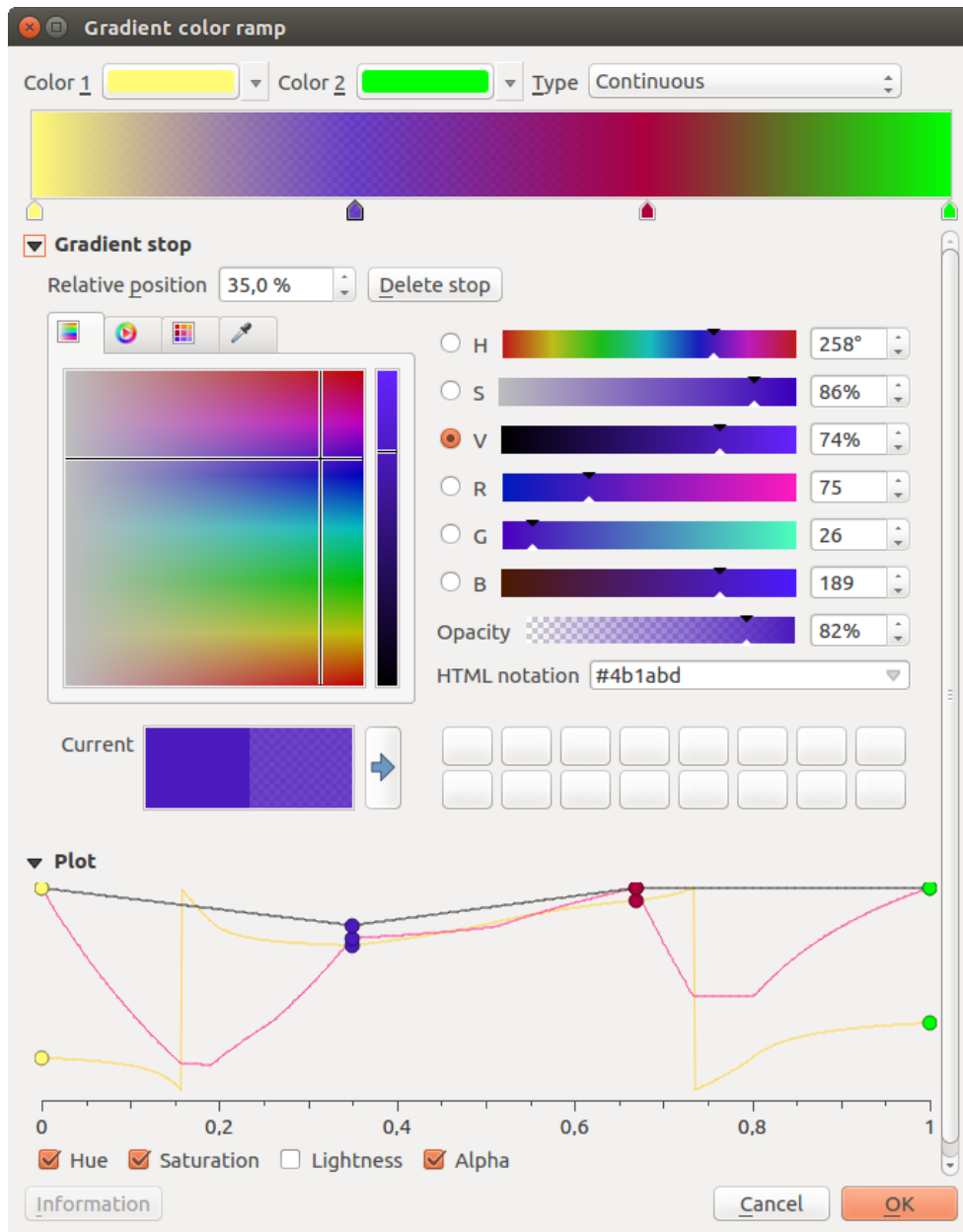


图 12.6: Example of custom gradient color ramp with multiple stops

- *Color presets*: allows to create a color ramp consisting of a list of colors selected by the user;
- *Random*: creates a random set of colors based on range of values for *Hue*, *Saturation*, *Value* and *Opacity* and a number of colors (*Classes*);
- *Catalog: ColorBrewer*: a set of predefined discrete color gradients you can customize the number of colors in the ramp;
- or *Catalog: cpt-city*: an access to a whole catalog of color gradients to locally *save as standard gradient*. The *cpt-city* option opens a new dialog with hundreds of themes included 'out of the box'.

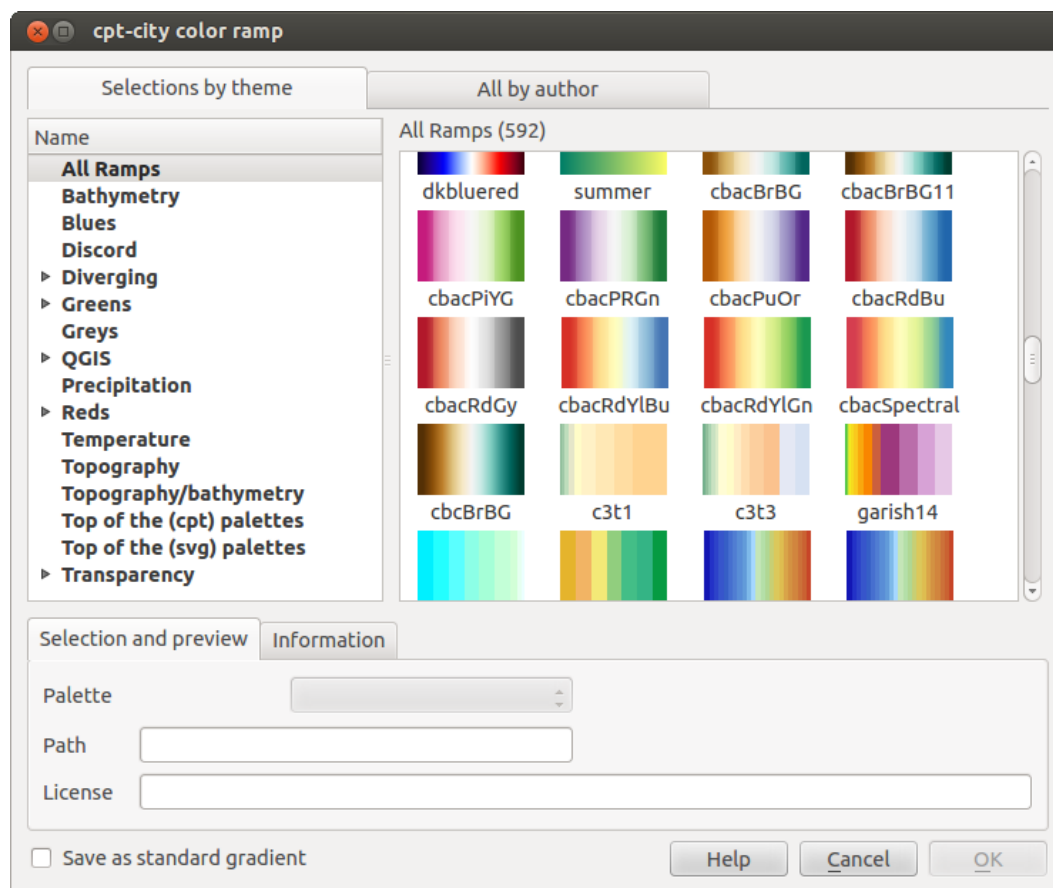


图 12.7: cpt-city dialog with hundreds of color ramps

小技巧: Easily adjust the color stops of the gradient color ramp

Double-clicking the ramp preview or drag-and-drop a color from the color spot onto the ramp preview adds a new color stop. Each color stop can be tweaked using the *Color Selector* widgets or by plotting each of its parameters. You can also reposition it using the mouse, the arrow keys (combine with *Shift* key for a larger move) or the *Relative position* spinbox. Pressing *Delete stop* as well as *DEL* key removes the selected color stop.

12.2 The Symbol Selector

The Symbol selector is the main dialog to design a symbol. You can create or edit Marker, Line or Fill Symbols.

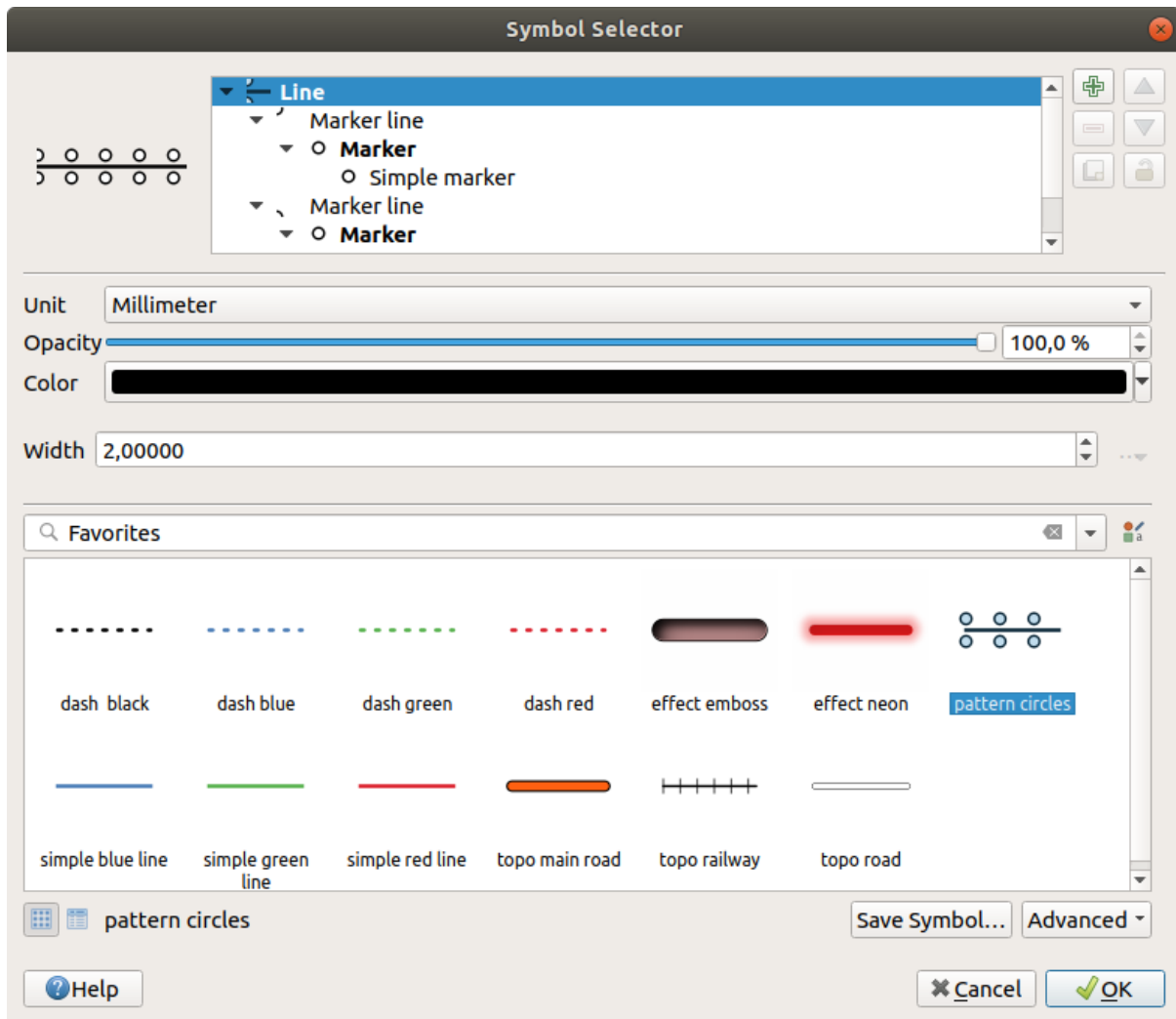


图 12.8: Designing a Line symbol





Two main components structure the symbol selector dialog:

- the symbol tree, showing symbol layers that are combined afterwards to shape a new global symbol
- and settings to configure the selected symbol layer in the tree.

12.2.1 The symbol layer tree

A symbol can consist of several *Symbol layers*. The symbol tree shows the overlay of these symbol layers that are combined afterwards to shape a new global symbol. Besides, a dynamic symbol representation is updated as soon as symbol properties change.

Depending on the level selected in the symbol tree items, various tools are made available to help you manage the tree:

-  add new symbol layer: you can stack as many symbols as you want
-  remove the selected symbol layer
- lock colors of symbol layer: a  locked color stays unchanged when user changes the color at the global (or upper) symbol level
-  duplicate a (group of) symbol layer(s)

- move up or down the symbol layer

12.2.2 Configuring a symbol

In QGIS, configuring a symbol is done in two steps: the symbol and then the symbol layer.

The symbol


At the top level of the tree, it depends on the layer geometry and can be of **Marker**, **Line** or **Fill** type. Each symbol can embed one or more symbols (including, of any other type) or symbol layers.

You can setup some parameters that apply to the global symbol:



- *Unit*: it can be **Millimeters**, **Points**, **Pixels**, **Meters at Scale**, **Map units** or **Inches** (see *Unit Selector* for more details)
- *Opacity*
- *Color*: when this parameter is changed by the user, its value is echoed to all unlocked sub-symbols color
- *Size* and *Rotation* for marker symbols
- *Width* for line symbols

小技巧: Use the *Size* (for marker symbols) or the *Width* (for line symbols) properties at the symbol level to proportionally resize all of its embedded *symbol layers* dimensions.

注解: The *Data-defined override* button next to the width, size or rotation parameters is inactive when setting the symbol from the Style manager dialog. When the symbol is connected to a map layer, this button helps you create *proportional or multivariate analysis* rendering.

- A preview of the *symbols library*: Symbols of the same type are shown and, through the editable drop-down list just above, can be filtered by free-form text or by *categories*. You can also update the list of symbols using the  Style Manager button and open the eponym dialog. There, you can use any capabilities as exposed in *The Style Manager* section.

The symbols are displayed either:

- in an icon list (with thumbnail, name and associated tags) using the  List View button below the frame;
- or as icon preview using the  Icon View button.

- Press the *Save Symbol* button to add the symbol being edited to the symbols library.

- With the *Advanced*  option, you can:

- for line and fill symbols, *Clip features to canvas extent*.
- for fill symbols, *Force right-hand rule orientation*: allows forcing rendered fill symbols to follow the standard "right hand rule" for ring orientation (i.e, polygons where the exterior ring is clockwise, and the interior rings are all counter-clockwise).

The orientation fix is applied while rendering only, and the original feature geometry is unchanged. This allows for creation of fill symbols with consistent appearance, regardless of the dataset being rendered and the ring orientation of individual features.

- Depending on the *symbolology* of the layer a symbol is being applied to, additional settings are available in the *Advanced* menu:

- * *Symbol levels...* to define the order of symbols rendering

- * *Data-defined Size Legend*
- * *Match to Saved Symbols...* and *Match to Symbols from File...* to automatically *assign symbols to classes*




The symbol layer

At a lower level of the tree, you can customize the symbol layers. The available symbol layer types depend on the upper symbol type. You can apply on the symbol layer  *paint effects* to enhance its rendering.

Because describing all the options of all the symbol layer types would not be possible, only particular and significant ones are mentioned below.

Common parameters

Some common options and widgets are available to build a symbol layer, regardless it's of marker, line or fill sub-type:

- the *color selector* widget to ease color manipulation
- *Units*: it can be **Millimeters**, **Points**, **Pixels**, **Meters at Scale**, **Map units** or **Inches** (see *Unit Selector* for more details)
- the  *data-defined override* widget near almost all options, extending capabilities of customizing each symbol (see *Data defined override setup* for more information)
- the  *Enable symbol layer* option controls the symbol layer's visibility. Disabled symbol layers are not drawn when rendering the symbol but are saved in the symbol. Being able to hide symbol layers is convenient when looking for the best design of your symbol as you don't need to remove any for the testing. The data-defined override then makes it possible to hide or display different symbol layers based on expressions (using, for instance, feature attributes).
- the  *Draw effects* button for *effects rendering*.

注解: While the description below assumes that the symbol layer type is bound to the feature geometry, keep in mind that you can embed symbol layers in each others. In that case, the lower level symbol layer parameter (placement, offset...) might be bound to the upper-level symbol, and not to the feature geometry itself.

Marker Symbols

Appropriate for point geometry features, marker symbols have several *Symbol layer types*:

- **Simple marker** (default)

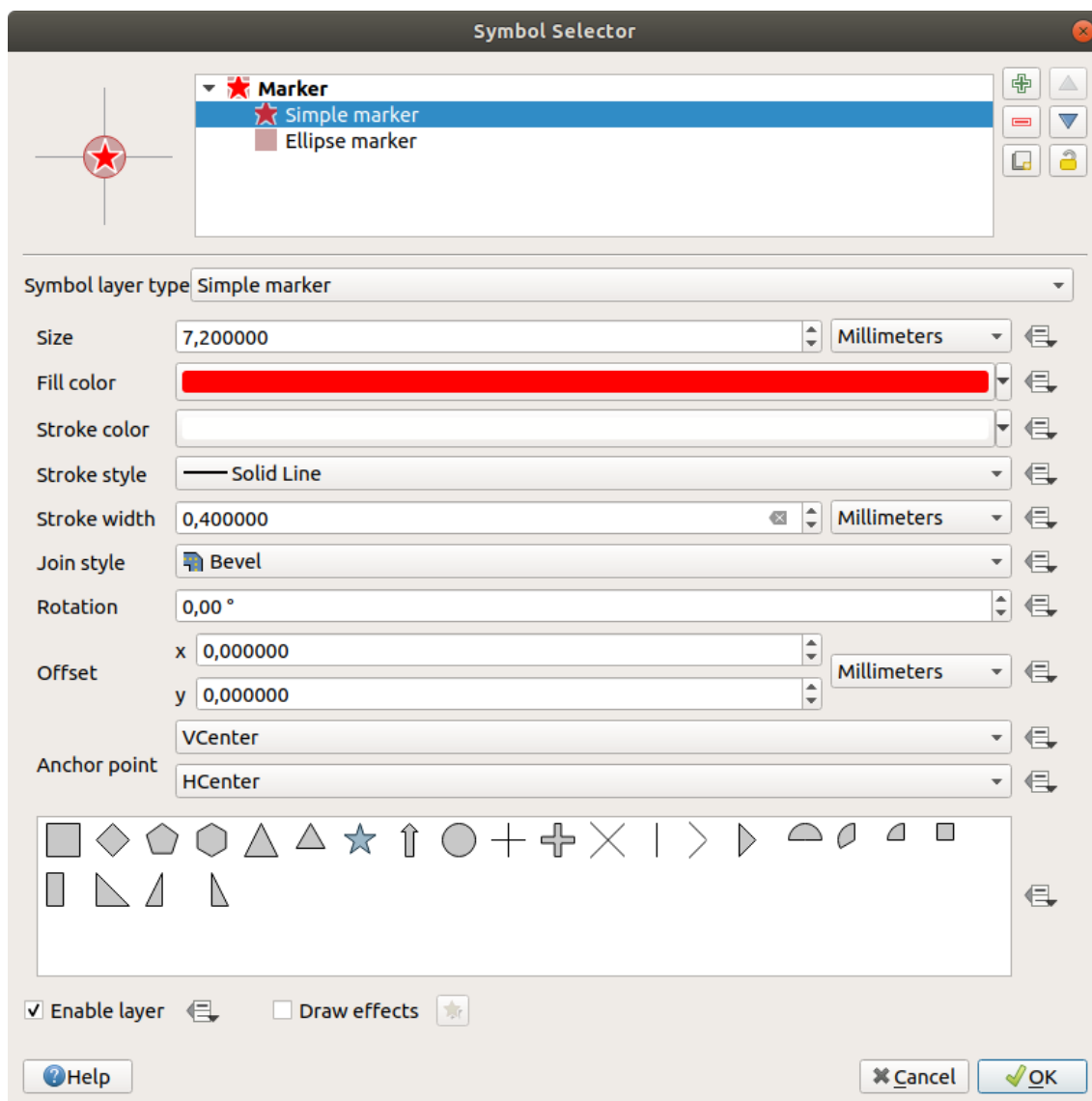




图 12.9: Designing a Simple Marker Symbol

- **Ellipse marker:** a simple marker symbol layer, with customizable width and height
- **Filled marker:** similar to the simple marker symbol layer, except that it uses a *fill sub symbol* to render the marker. This allows use of all the existing QGIS fill (and stroke) styles for rendering markers, e.g. gradient or shapeburst fills.
- **Font marker:** similar to the simple marker symbol layer, except that it uses installed fonts to render the marker. Its additional properties are:
 - *Font family*
 - *Font style*
 - *Character(s)*, representing the text to display as symbol. They can be typed in or selected from the font characters collection widget and you can live *Preview* them with the selected settings.
- **Geometry generator** (see *The Geometry Generator*)
- **Mask:** its sub-symbol defines a mask shape whose color property will be ignored and only the opacity will be used. This is convenient when the marker symbol overlaps with labels or other symbols whose colors are close, making it hard to decipher. More details at *Masks Properties*.

- **Raster image marker:** use an image (PNG, JPG, BMP ...) as marker symbol. The image can be a file on the disk, a remote URL or embedded in the style database (*more details*). Width and height of the image can be set independently or using the  Lock aspect ratio. The size can be set using any of the *common units* or as a percentage of the image's original size (scaled by the width).
- **Vector Field marker** (see *The Vector Field Marker*)
- **SVG marker:** provides you with images from your SVG paths (set in *Settings ► Options... ► System* menu) to render as marker symbol. Width and height of the symbol can be set independently or using the  Lock aspect ratio. Each SVG file colors and stroke can also be adapted. The image can be a file on the disk, a remote URL or embedded in the style database (*more details*).

注解: SVG version requirements

QGIS renders SVG files that follow the [SVG Tiny 1.2 profile](#), intended for implementation on a range of devices, from cellphones and PDAs to laptop and desktop computers, and thus includes a subset of the features included in SVG 1.1 Full, along with new features to extend the capabilities of SVG.

Some features not included in these specifications might not be rendered correctly in QGIS.

小技巧: Enable SVG marker symbol customization

To have the possibility to change the colors of a *SVG marker*, you have to add the placeholders `param(fill)` for fill color, `param(outline)` for stroke color and `param(outline-width)` for stroke width. These placeholders can optionally be followed by a default value, e.g.:

```
<svg width="100%" height="100%">
<rect fill="param(fill) #ff0000" stroke="param(outline) #00ff00" stroke-width=
→"param(outline-width) 10" width="100" height="100">
</rect>
</svg>
```

Line Symbols

Appropriate for line geometry features, line symbols have the following symbol layer types:

- **Simple line** (default): available settings are:

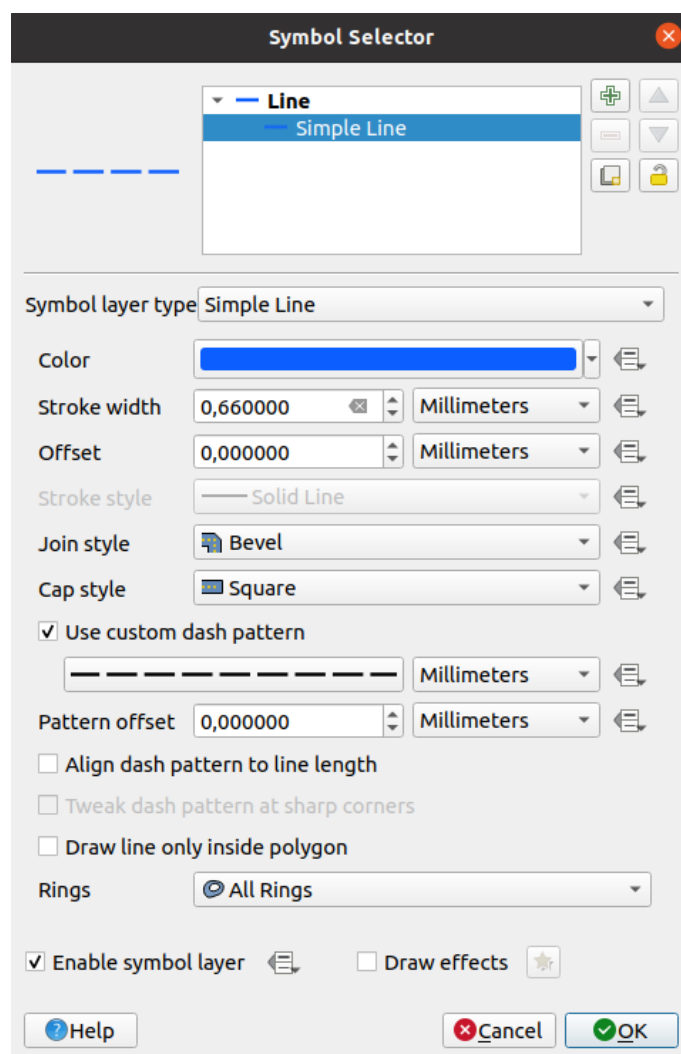


图 12.10: Designing a Simple Line Symbol

The simple line symbol layer type has many of the same properties as the *simple marker symbol*, and in addition:

- *Cap style*
- *Use custom dash pattern*: overrides the *Stroke style* setting with a custom dash.
- *Align dash pattern to line length*: the dash pattern length will be adjusted so that the line will end with a complete dash element, instead of a gap.
- *Tweak dash pattern at sharp corners*: dynamically adjusts the dash pattern placement so that sharp corners are represented by a full dash element coming into and out of the sharp corner. Dependent on *Align dash pattern to line length*.
- *Draw line only inside polygon*
- **Arrow**: draws lines as curved (or not) arrows with a single or a double head with configurable (and data-defined):
 - *Head type*
 - *Arrow type*
 - *Arrow width*
 - *Arrow width at start*

- Head length
- Head thickness
- Offset

It is possible to create *Curved arrows* (the line feature must have at least three vertices) and *Repeat arrow on each segment*. It also uses a *fill symbol* such as gradients or shapeburst to render the arrow body. Combined with the geometry generator, this type of layer symbol helps you representing flow maps.

- **Geometry generator** (see *The Geometry Generator*)

- **Marker line:** repeats a *marker symbol* over the length of a line.

- The markers placement can be at a regular distance or based on the line geometry: first, last or each vertex, on the central point of the line or of each segment, or on every curve point.
- The markers placement can also be given an offset along the line
- The *Rotate marker to follow line direction* option sets whether each marker symbol should be oriented relative to the line direction or not.

Because a line is often a succession of segments of different directions, the rotation of the marker is calculated by averaging over a specified distance along the line. For example, setting the *Average angle over* property to 4mm means that the two points along the line that are 2mm before and after the symbol placement are used to calculate the line angle for that marker symbol. This has the effect of smoothing (or removing) any tiny local deviations from the overall line direction, resulting in much nicer visual orientations of the marker line symbols.

- The marker line can also be offset from the line itself.

- **Hashed line:** repeats a line segment (a hash) over the length of a line symbol, with a line sub-symbol used to render each individual segment. In other words, a hashed line is like a marker line in which marker symbols are replaced with segments. As such, the hashed lines have the *same properties* as marker line symbols, along with:

- Hash length
- Hash rotation
- *Rotate hash to follow line direction*

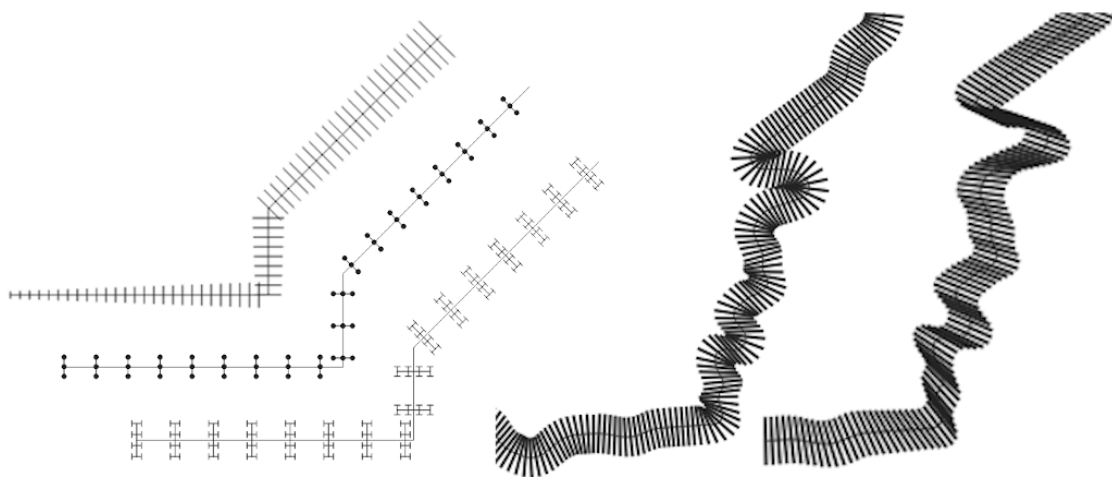


图 12.11: Examples of hashed lines

Fill Symbols

Appropriate for polygon geometry features, fill symbols have also several symbol layer types:

- **Simple fill** (default): fills a polygon with a uniform color

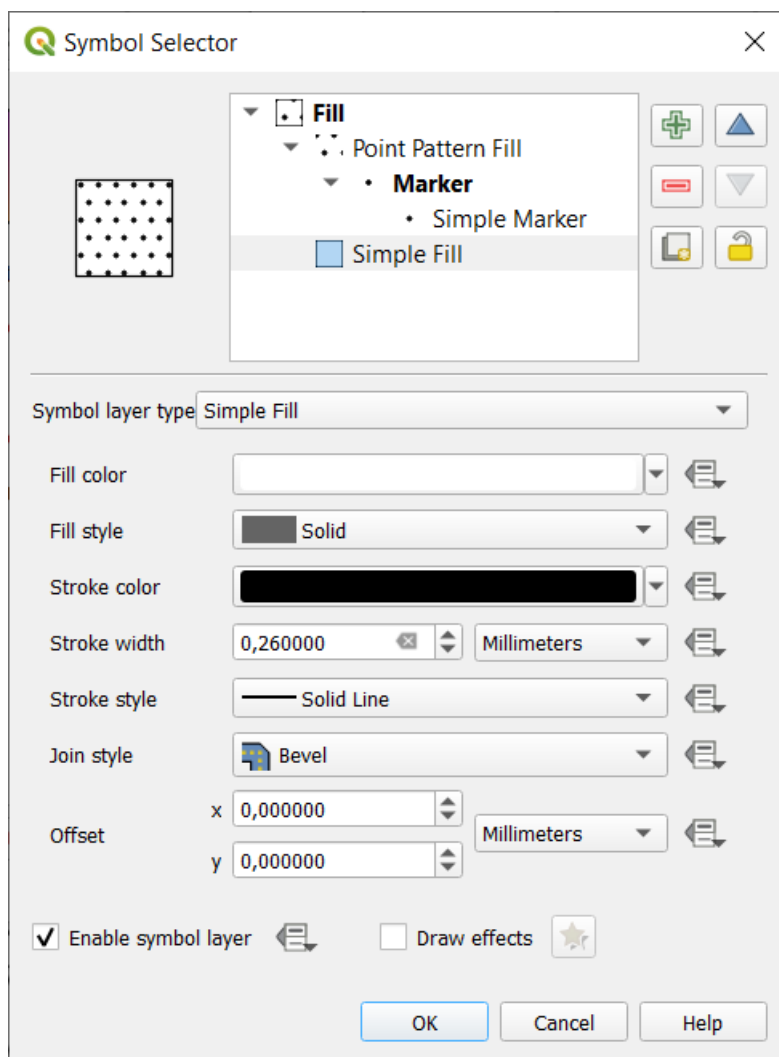


图 12.12: Designing a Simple Fill Symbol

- **Centroid fill**: places a *marker symbol* at the centroid of the visible feature. The position of the marker may not be the real centroid of the feature, because calculation takes into account the polygon(s) clipped to area visible in map canvas for rendering and ignores holes. Use the *geometry generator symbol* if you want the exact centroid.

You can:

- *Force point inside polygon*
- *Draw point on every part of multi-part feature* or place the point only on its biggest part
- display the marker symbol(s) in whole or in part, keeping parts overlapping the current feature geometry (*Clip markers to polygon boundary*) or the geometry part the symbol belongs to (*Clip markers to current part boundary only*)

- **Geometry generator** (see *The Geometry Generator*)
- **Gradient fill**: uses a radial, linear or conical gradient, based on either simple two color gradients or a predefined *gradient color ramp* to fill polygons. The gradient can be rotated and applied on a single feature basis or across

the whole map extent. Also start and end points can be set via coordinates or using the centroid (of feature or map). A data-defined offset can be defined;

- **Line pattern fill:** fills the polygon with a hatching pattern of *line symbol layer*. You can set a rotation, the spacing between lines and an offset from the feature boundary;
- **Point pattern fill:** fills the polygon with a hatching pattern of *marker symbol layer*. You can set the distance and a displacement between rows of markers, and an offset from the feature boundary;
- **Random marker fill:** fills the polygon with a *marker symbol* placed at random locations within the polygon boundary. You can set:
 - the number of marker symbols to render, either as an absolute count or as density-based (the fill density will remain the same on different scale / zoom levels)
 - an optional random number seed, to give consistent placement of markers whenever maps are refreshed (also allows random placement to play nice with QGIS server and tile-based rendering)
 - whether markers rendered near the edges of polygons should be clipped to the polygon boundary or not
- **Raster image fill:** fills the polygon with tiles from a raster image (PNG JPG, BMP ...). The image can be a file on the disk, a remote URL or an embedded file encoded as a string (*more details*). Options include (data defined) opacity, image width, coordinate mode (object or viewport), rotation and offset. The image width can be set using any of the *common units* or as a percentage of the original size.
- **SVG fill:** fills the polygon using *SVG markers*;
- **Shapeburst fill:** buffers a gradient fill, where a gradient is drawn from the boundary of a polygon towards the polygon's centre. Configurable parameters include distance from the boundary to shade, use of color ramps or simple two color gradients, optional blurring of the fill and offsets;
- **Outline: Arrow:** uses a line *arrow symbol* layer to represent the polygon boundary. The settings for the outline arrow are the same as for line symbols.
- **Outline: Hashed line:** uses a *hash line symbol* layer to represent the polygon boundary (the interior rings, the exterior ring or all the rings). The settings for the outline hashed line are the same as for line symbols.
- **Outline: Marker line:** uses a *marker line symbol* layer to represent the polygon boundary (the interior rings, the exterior ring or all the rings). The settings for the outline marker line are same as for line symbols.
- **Outline: simple line:** uses a *simple line symbol* layer to represent the polygon boundary (the interior rings, the exterior ring or all the rings). The settings for the outline simple line are the same as for line symbols. The *Draw line only inside polygon* option displays the polygon borders inside the polygon and can be useful to clearly represent adjacent polygon boundaries.

注解: When geometry type is polygon, you can choose to disable the automatic clipping of lines/polygons to the canvas extent. In some cases this clipping results in unfavourable symbology (e.g. centroid fills where the centroid must always be the actual feature's centroid).

The Geometry Generator

Available with all types of symbols, the *geometry generator* symbol layer allows to use *expression syntax* to generate a geometry on the fly during the rendering process. The resulting geometry does not have to match with the original geometry type and you can add several differently modified symbol layers on top of each other.

Some examples:

```
-- render the centroid of a feature
centroid( $geometry )

-- visually overlap features within a 100 map units distance from a point
-- feature, i.e generate a 100m buffer around the point
```

(下页继续)

```

buffer( $geometry, 100 )

-- Given polygon layer1( id1, layer2_id, ...) and layer2( id2, fieldn...)
-- render layer1 with a line joining centroids of both where layer2_id = id2
make_line( centroid( $geometry ),
           centroid( geometry( get_feature( 'layer2', 'id2', attribute(
               $currentfeature, 'layer2_id' ) ) )
           )

-- Create a nice radial effect of points surrounding the central feature
-- point when used as a MultiPoint geometry generator
collect_geometries(
  array_foreach(
    generate_series( 0, 330, 30 ),
    project( $geometry, .2, radians( @element ) )
  )
)

```

The Vector Field Marker

The vector field marker is used to display vector field data such as earth deformation, tidal flows, and the like. It displays the vectors as lines (preferably arrows) that are scaled and oriented according to selected attributes of data points. It can only be used to render point data; line and polygon layers are not drawn by this symbology.

The vector field is defined by attributes in the data, which can represent the field either by:

- **cartesian** components (x and y components of the field)
- or **polar** coordinates: in this case, attributes define Length and Angle. The angle may be measured either clockwise from north, or Counterclockwise from east, and may be either in degrees or radians.
- or as **height only** data, which displays a vertical arrow scaled using an attribute of the data. This is appropriate for displaying the vertical component of deformation, for example.

The magnitude of field can be scaled up or down to an appropriate size for viewing the field.

12.3 Setting a label

Labels are textual information you can display on vector features or maps. They add details you could not necessarily represent using symbols. Two types of text-related items are available in QGIS:

- *Text Format*: defines the appearance of the text, including *font*, *size*, *colors*, *shadow*, *background*, *buffer*, ...

They can be used to render texts over the map (layout/map title, decorations, scale bar, ...), usually through the *font* widget.

To create a *Text Format* item:

1. Open the  *Style Manager* dialog
2. Activate the *Text format* tab

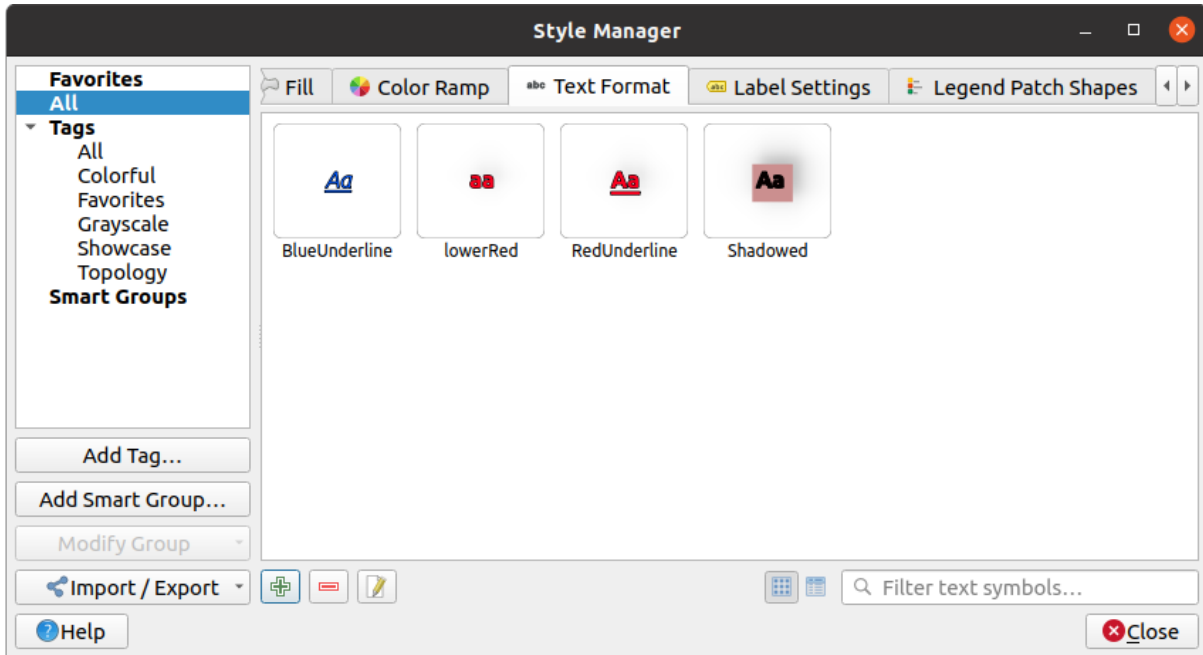



图 12.13: Text formats in Style Manager dialog

3. Press the  Add item button. The *Text Format* dialog opens for *configuration*. As usual, these properties are *data-definable*.
- *Label Settings*: extend the text format settings with properties related to the location or the interaction with other texts or features (*callouts, placement, overlay, scale visibility, mask ...*).

They are used to configure smart labelling for vector layers through the  *Labels* tab of the vector *Layer Properties* dialog or *Layer Styling* panel or using the  *Layer Labeling Options* button of the *Label toolbar*.

To create a *Label Settings* item:

1. Open the  *Style Manager* dialog
2. Activate the *Label Settings* tab

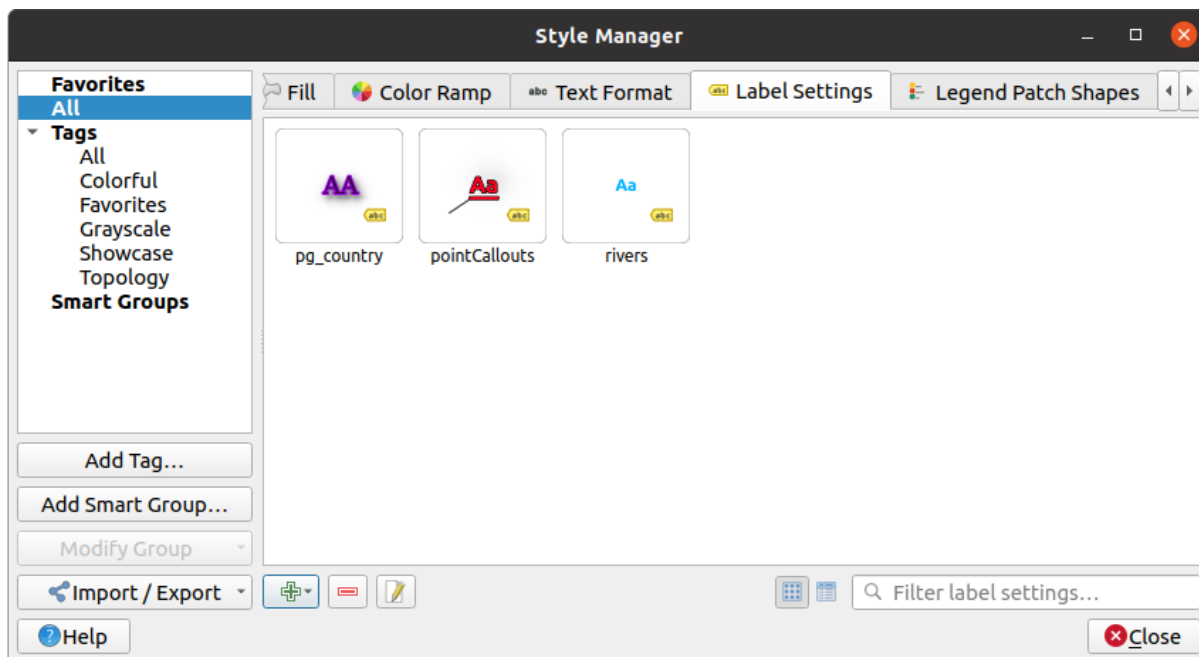



图 12.14: Label Settings in Style Manager dialog

3. Press the  Add item menu and select the entry corresponding to the geometry type of the features you want to label.

The *Label Settings* dialog opens with the following properties. As usual, these properties are *data-definable*.

12.3.1 Formatting the label text

Most of the following properties are common to *Text Format* and *Label Settings* items.

Text tab

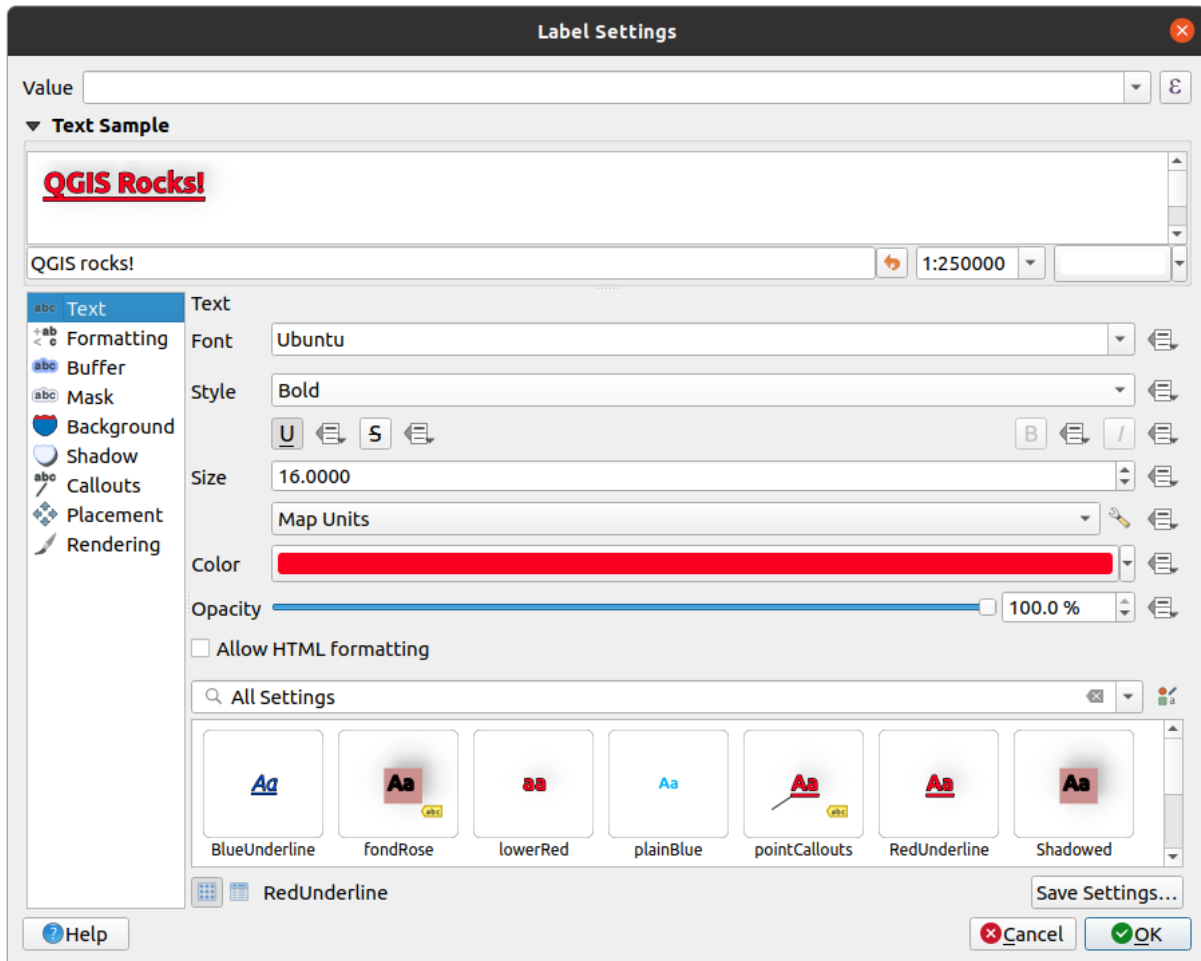


图 12.15: Labels settings - Text tab

In the `abc` Text tab, you can set:

- the *Font*, from the ones available on your machine
- the *Style*: along with the common styles of the font, you can set whether the text should be underlined or striked through
- the *Size* in any *supported unit*
- the *Color*
- and the *Opacity*.

At the bottom of the tab, a widget shows a filterable list of compatible items stored in your *style manager database*. This allows you to easily configure the current text format or label setting based on an existing one, and also save a new item to the style database: Press the *Save format...* or *Save settings...* button and provide a name and tag(s).

注解: When configuring a *Label Settings* item, text format items are also available in this widget. Select one to quickly overwrite the current *textual properties* of the label. Likewise, you can create/overwrite a text format from there.

Formatting tab

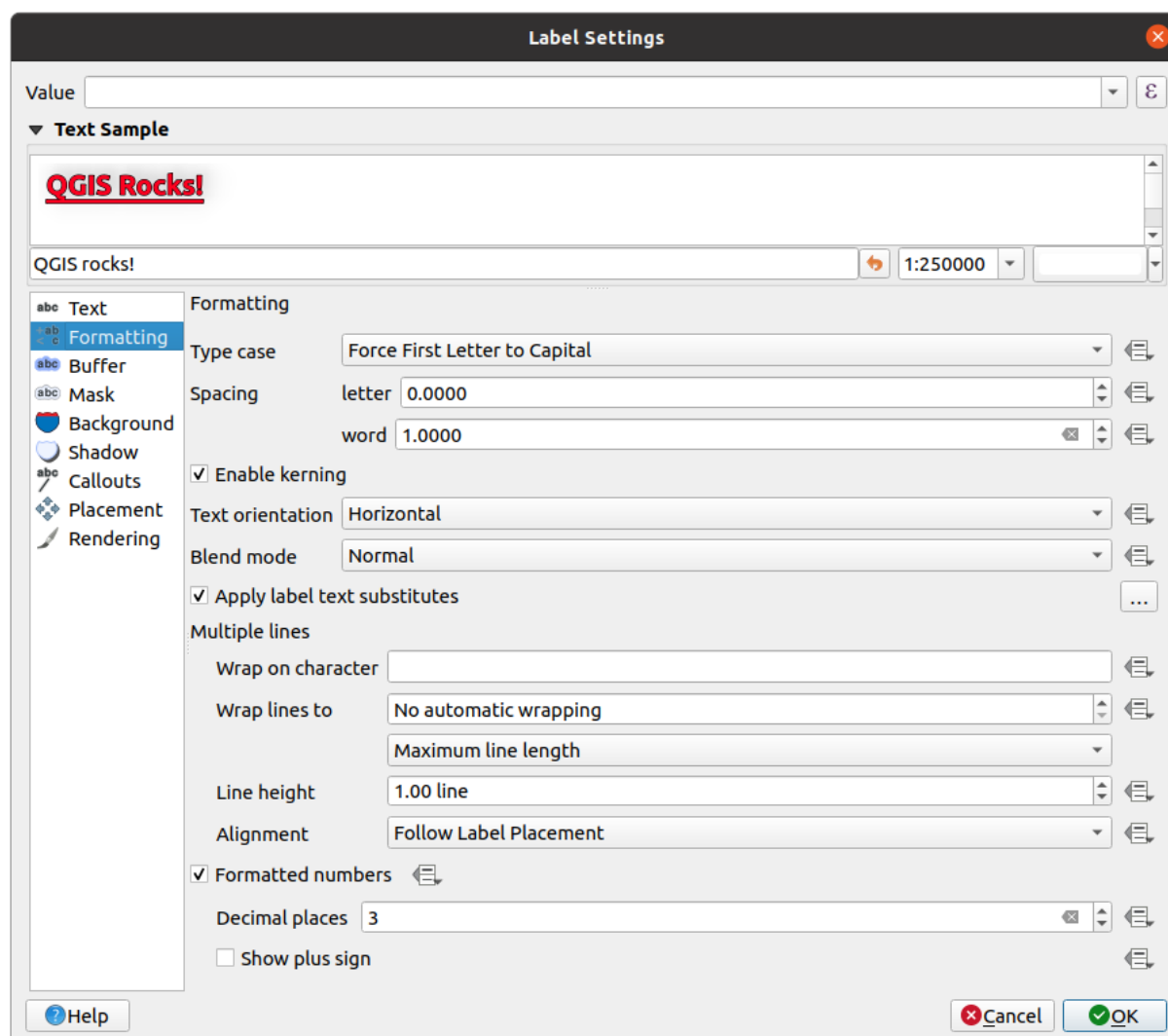


图 12.16: Label settings - Formatting tab

In the **Formatting** tab, you can:

- Use the *Type case* option to change the capitalization style of the text. You have the possibility to render the text as:
 - *No change*
 - *All uppercase*
 - *All lowercase*
 - *Title case*: modifies the first letter of each word into capital, and turns the other letters into lower case if the original text is using a single type case. In case of mixed type cases in the text, the other letters are left untouched.
 - *Force first letter to capital*: modifies the first letter of each word into capital and leaves the other letters in the text untouched.
- Under *Spacing*, change the space between words and between individual letters.
- *Enable kerning* of the text font

- Set the *Text orientation* which can be *Horizontal* or *Vertical*. It can also be *Rotation-based* when setting a label (e.g., to properly label line features in *parallel* placement mode).
- Use the *Blend mode* option to determine how your labels will mix with the map features below them (more details at *Blending Modes*).
- The *Apply label text substitutes* option allows you to specify a list of texts to substitute to texts in feature labels (e.g., abbreviating street types). Replacement texts are used when displaying labels on the map. Users can also export and import lists of substitutes to make reuse and sharing easier.
- Configure *Multiple lines*:
 - Set a character that will force a line break in the text with the *Wrap on character* option
 - Set an ideal line size for auto-wrapping using the *Wrap lines to* option. The size can represent either the *Maximum line length* or the *Minimum line length*.
 - Decide the *Line Height*
 - Format the *Alignment*: typical values available are *Left*, *Right*, *Justify* and *Center*.

When setting point labels properties, the text alignment can also be *Follow label placement*. In that case, the alignment will depend on the final placement of the label relative to the point. E.g., if the label is placed to the left of the point, then the label will be right aligned, while if it is placed to the right, it will be left aligned.

注解: The *Multiple lines* formatting is not yet supported by curve based *label placement*. The options will then be deactivated.

- For line labels you can include *Line direction symbol* to help determine the line directions, with symbols to use to indicate the *Left* or *Right*. They work particularly well when used with the *curved* or *Parallel* placement options from the *Placement* tab. There are options to set the symbols position, and to *Reverse direction*.
- Use the *Formatted numbers* option to format numeric texts. You can set the number of *Decimal places*. By default, 3 decimal places will be used. Use the *Show plus sign* if you want to show the plus sign for positive numbers.

Buffer tab

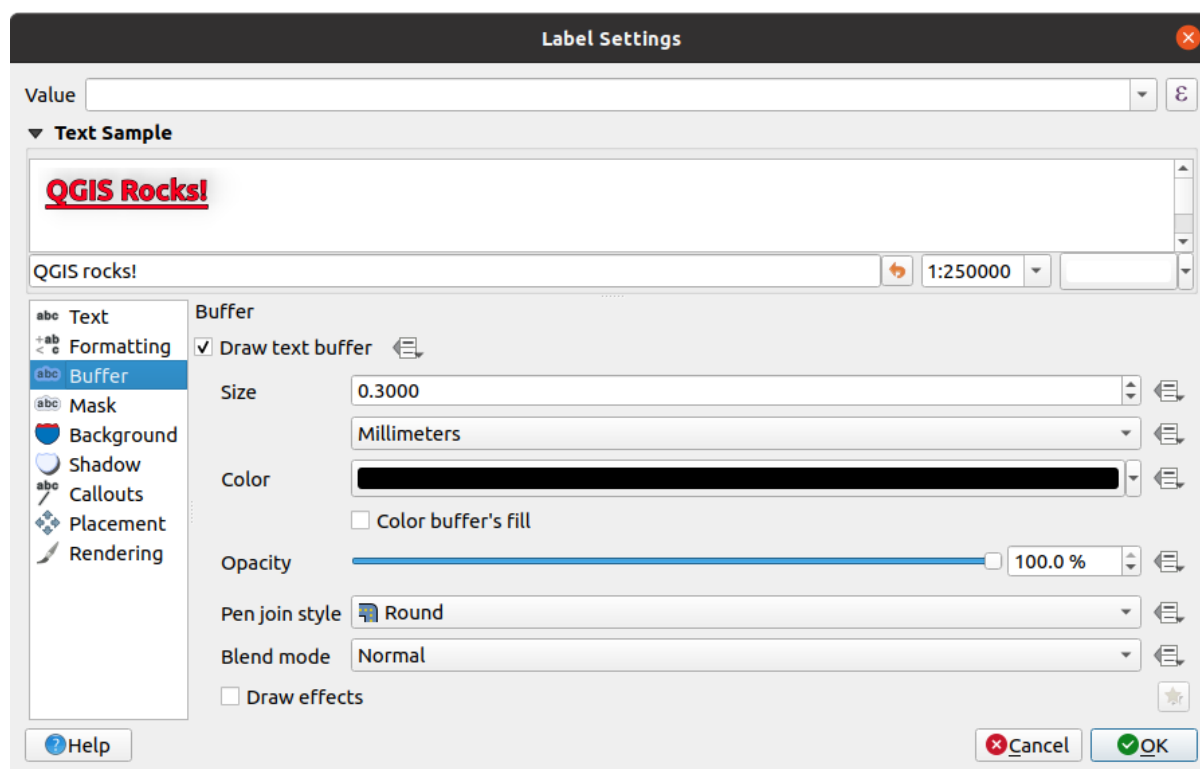




图 12.17: Label settings - Buffer tab

To create a buffer around the label, activate the *Draw text buffer* checkbox in the **abc** *Buffer* tab. Then you can:

- Set the buffer's *Size* in any *supported unit*
- Select the buffer's *Color*
- *Color buffer's fill*: The buffer expands from the label's outline, so, if the option is activated, the label's interior is filled. This may be relevant when using partially transparent labels or with non-normal blending modes, which will allow seeing behind the label's text. Unchecking the option (while using totally transparent labels) will allow you to create outlined text labels.
- Define the buffer's *Opacity*
- Apply a *Pen join style*: it can be *Round*, *Miter* or *Bevel*
- Use the *Blend mode* option to determine how your label's buffer will mix with the map components below them (more details at *Blending Modes*).
- Check *Draw effects* to add advanced  *paint effects* for improving text readability, eg through outer glows and blurs.

Background tab

The  *Background* tab allows you to configure a shape that stays below each label. To add a background, activate the *Draw Background* checkbox and select the *Shape* type. It can be:

- a regular shape such as *Rectangle*, *Square*, *Circle* or *Ellipse*
- an *SVG* symbol from a file, a URL or embedded in the project or style database (*more details*)
- or a *Marker Symbol* you can create or select from the *symbol library*.

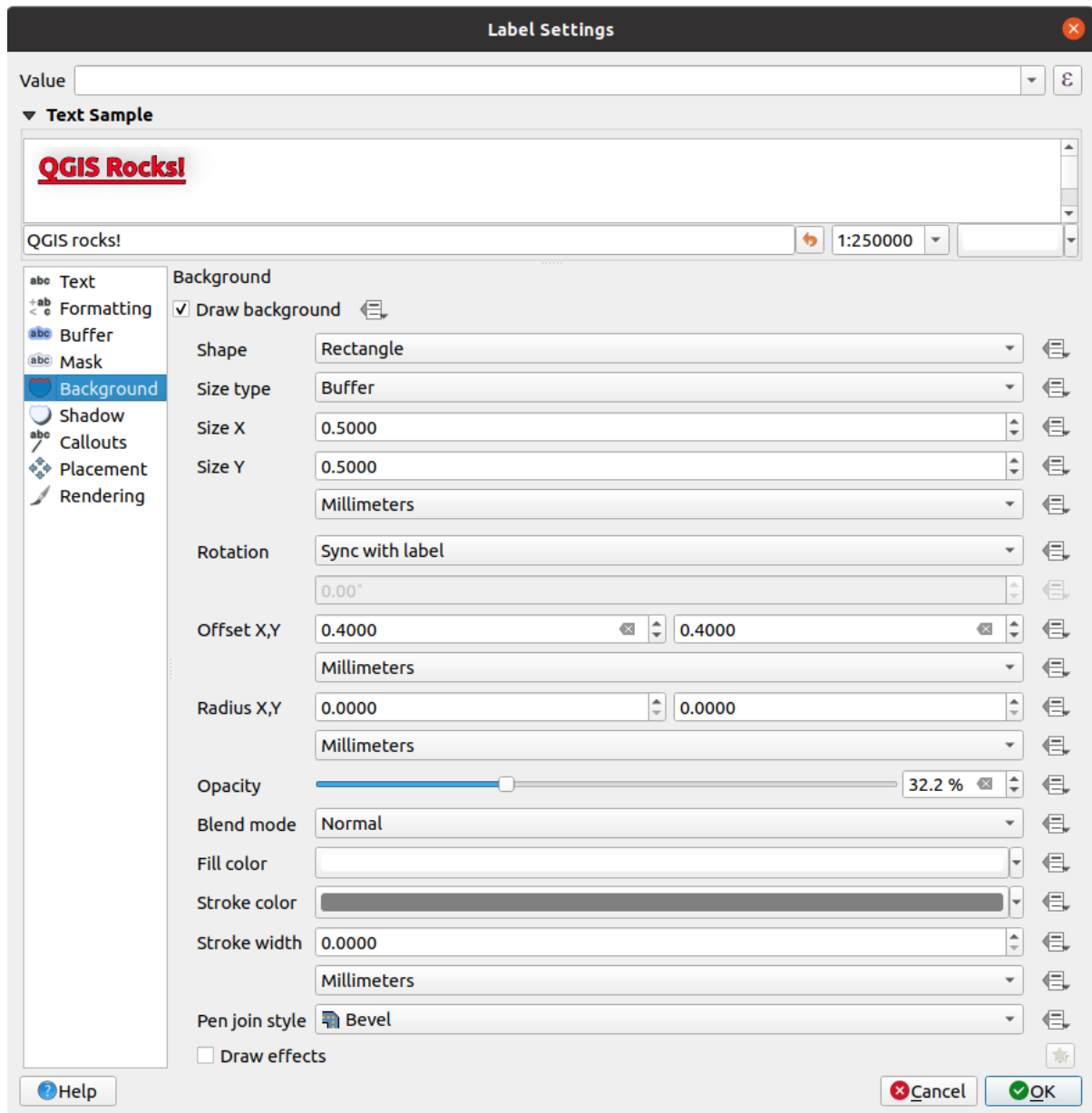



图 12.18: Label settings - Background tab

Depending on the selected shape, you need to configure some of the following properties:

- The *Size type* of the frame, which can be:
 - *Fixed*: using the same size for all the labels, regardless the size of the text
 - or a *Buffer* over the text's bounding box

- The *Size* of the frame in X and Y directions, using any *supported units*
- A *Rotation* of the background, between *Sync with label*, *Offset of label* and *Fixed*. The last two require an angle in degrees.
- An *Offset X,Y* to shift the background item in the X and/or Y directions
- A *Radius X,Y* to round the corners of the background shape (applies to rectangle and square shapes only)
- An *Opacity* of the background
- A *Blend mode* to mix the background with the other items in the rendering (see *Blending Modes*).
- The *Fill color*, *Stroke color* and *Stroke width* for shape types other than the marker symbol. Use the *Load symbol parameters* to revert changes on an SVG symbol to its default settings.
- A *Pen join style*: it can be *Round*, *Miter* or *Bevel* (applies to rectangle and square shapes only)
- *Draw effects* to add advanced  *paint effects* for improving text readability, eg through outer glows and blurs.

Shadow tab

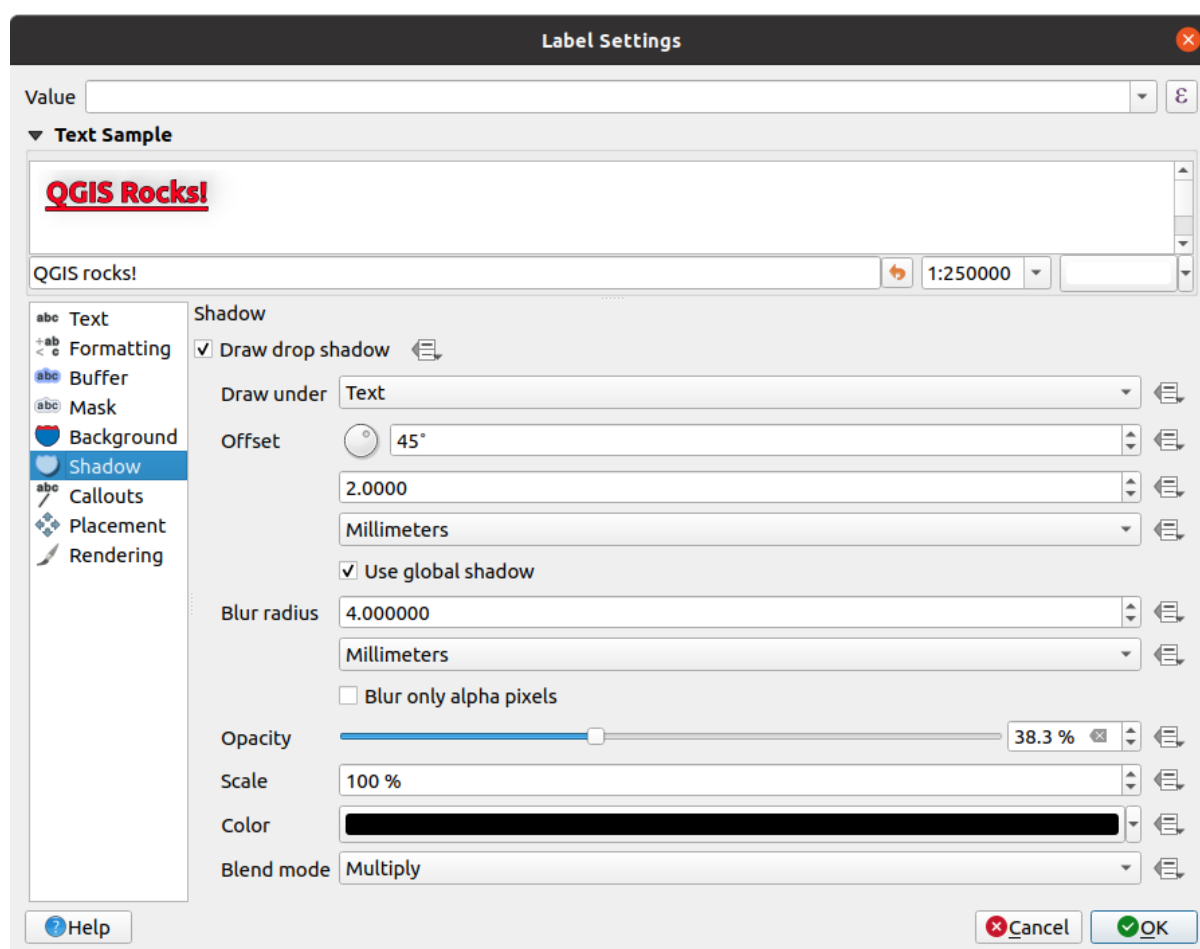




图 12.19: Label settings - Shadow tab

To add a shadow to the text, enable the  *Shadow* tab and activate the  *Draw drop shadow*. Then you can:

- Indicate the item used to generate the shadow with *Draw under*. It can be the *Lowest label component* or a particular component such as the *Text* itself, the *Buffer* or the *Background*.

- Set the shadow's *Offset* from the item being shadowed, ie:
 - The angle: clockwise, it depends on the underlying item orientation
 - The distance of offset from the item being shadowed
 - The units of the offset


If you tick the *Use global shadow* checkbox, then the zero point of the angle is always oriented to the north and doesn't depend on the orientation of the label's item.

- Influence the appearance of the shadow with the *Blur radius*. The higher the number, the softer the shadows, in the units of your choice.
- Define the shadow's *Opacity*
- Rescale the shadow's size using the *Scale* factor
- Choose the shadow's *Color*
- Use the *Blend mode* option to determine how your label's shadow will mix with the map components below them (more details at *Blending Modes*).

12.3.2 Configuring interaction with labels

Other than the text formatting settings exposed above, you can also set how labels interact with each others or with the features.

Mask tab

The  *Mask* tab allows you to define a mask area around the labels. This feature is very useful when you have overlapping symbols and labels with similar colors, and you want to make the labels visible.

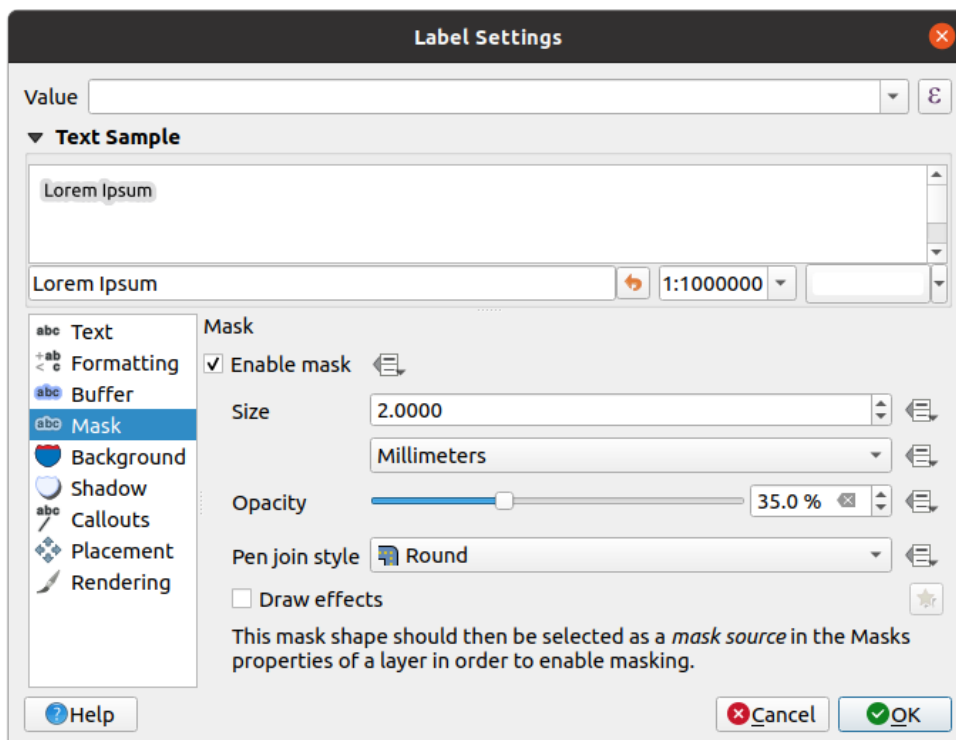




图 12.20: Labels settings - Mask tab

To create masking effects on labels:


1. Activate the *Enable mask* checkbox in the  tab.
2. Then you can set:
 - the mask's *Size* in the *supported units*
 - the *Opacity* of the mask area around the label
 - a *Pen Join Style*
 - *paint effects* through the *Draw effects* checkbox.
3. Select this mask shape as a mask source in the overlapping layer properties  *Mask* tab (see *Masks Properties*).

Callouts tab

A common practice when placing labels on a crowded map is to use **callouts** - labels which are placed outside (or displaced from) their associated feature are identified with a dynamic line connecting the label and the feature. If one of the two endings (either the label or the feature) is moved, the shape of the connector is recomputed.




图 12.21: Labels with various callouts settings

To add a callout to a label, enable the  *Callouts* tab and activate the *Draw callouts*. Then you can:

1. Select the *Style* of connector, one of:
 - *Simple lines*: a straight line, the shortest path
 - *Manhattan style*: a 90° broken line
2. Select the *Line style* with full capabilities of a *line symbol* including layer effects, and data-defined settings
3. Set the *Minimum length* of callout lines
4. Set the *Offset from feature* option: controls the distance from the feature (or its anchor point if a polygon) where callout lines end. Eg, this avoids drawing lines right up against the edges of the features.

5. Set the *Offset from label area* option: controls the distance from the label anchor point (where the callout line ends). This avoids drawing lines right up against the text.
6. *Draw lines to all feature parts* from the feature's label
7. Set the *Anchor point* for the (polygon) feature (the end point of the connector line). Available options:
 - *Pole of inaccessibility*
 - *Point on exterior*
 - *Point on surface*
 - *Centroid*
8. Set the *Label anchor point*: controls where the connector line should join to the label text. Available options:
 - *Closest point*
 - *Centroid*
 - Fixed position at the edge (*Top left, Top center, Top right, Left middle, Right middle, Bottom left, Bottom center* and *Bottom right*).

Placement tab

Choose the  *Placement* tab for configuring label placement and labeling priority. Note that the placement options differ according to the type of vector layer, namely point, line or polygon, and are affected by the global *PAL setting*.

Placement for point layers

Point labels placement modes available are:

- *Cartographic*: point labels are generated with a better visual relationship with the point feature, following ideal cartographic placement rules. Labels can be placed:
 - at a set *Distance* in *supported units*, either from the point feature itself or from the bounds of the symbol used to represent the feature (set in *Distance offset from*). The latter option is especially useful when the symbol size isn't fixed, e.g. if it's set by a data defined size or when using different symbols in a *categorized* renderer.
 - following a *Position priority* that can be customized or set for an individual feature using a data defined list of prioritised positions. This also allows only certain placements to be used, so e.g. for coastal features you can prevent labels being placed over the land.

By default, cartographic mode placements are prioritised in the following order (respecting the *guidelines from Krygier and Wood (2011)* and other cartographic textbooks):

1. top right
 2. top left
 3. bottom right
 4. bottom left
 5. middle right
 6. middle left
 7. top, slightly right
 8. bottom, slightly left.
- *Around Point*: labels are placed in a circle around the feature. equal radius (set in *Distance*) circle around the feature. The placement priority is clockwise from the "top right". The position can be constrained using the data-defined *Quadrant* option.

- *Offset from Point*: labels are placed at an *Offset X, Y* distance from the point feature, in various units, or preferably over the feature. You can use a data-defined *Quadrant* to constrain the placement and can assign a *Rotation* to the label.

Placement for line layers

Label modes for line layers include:

- *Parallel*: draws the label parallel to a generalised line representing the feature, with preference for placement over straighter portions of the line. You can define:
 - *Allowed positions*: *Above line*, *On line*, *Below line* and *Line orientation dependent position* (placing the label at the left or the right of the line). It's possible to select several options at once. In that case, QGIS will look for the optimal label position.
 - *Distance* between the label and the line
- *Curved*: draws the label following the curvature of the line feature. In addition to the parameters available with the *Parallel* mode, you can set the *Maximum angle between curved characters*, either inside or outside.
- *Horizontal*: draws labels horizontally along the length of the line feature.

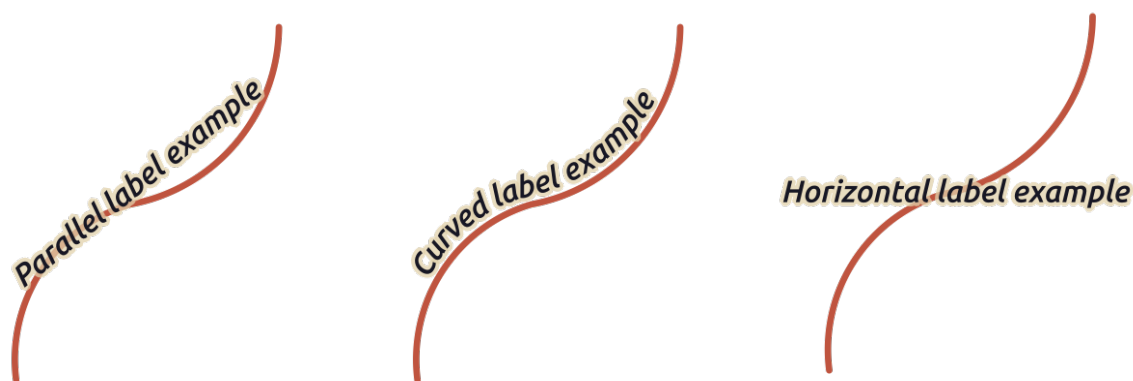






图 12.22: Label placement examples for lines

Next to placement modes, you can set:

- *Repeating Labels Distance* to display multiple times the label over the length of the feature. The distance can be in *Millimeters*, *Points*, *Pixels*, *Meters at scale*, *Map Units* and *Inches*.
- A *Label Overrun Distance* (not available for horizontal mode): specifies the maximal allowable distance a label may run past the end (or start) of line features. Increasing this value can allow for labels to be shown for shorter line features.
- *Label Anchoring*: controls the placement of the labels along the line feature they refer to. Click on *Settings ...* to choose:
 - the position along the line (as a ratio) which labels will be placed close to. It can be data-defined and possible values are:
 - *  *Center of Line*
 - *  *Start of Line*
 - *  *End of Line*
 - * or  *Custom....*

- *Placement Behavior*: use *Preferred Placement Hint* to treat the label anchor only as a hint for the label placement. By choosing *Strict*, labels are placed exactly on the label anchor.

Placement for polygon layers

You can choose one of the following modes for placing labels of polygons:

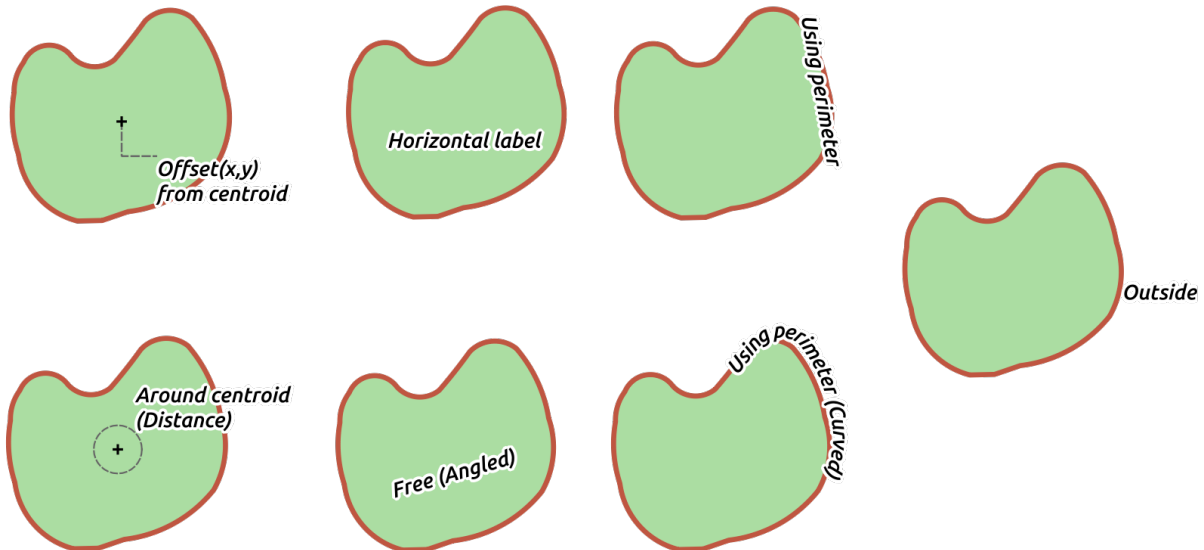


图 12.23: Label placement examples for polygons

- *Offset from Centroid*: labels are placed over the feature centroid or at a fixed *Offset X, Y* distance (in *supported units*) from the centroid. The reference centroid can be determined based on the part of the polygon rendered in the map canvas (*visible polygon*) or the *whole polygon*, no matter if you can see it. You can also:
 - force the centroid point to lay inside their polygon
 - place the label within a specific quadrant
 - assign a rotation
 - *Allow placing labels outside of polygons* when it is not possible to place them inside the polygon. Thanks to data-defined properties, this makes possible to either allow outside labels, prevent outside labels, or force outside labels on a feature-by-feature basis.
- *Around Centroid*: places the label within a preset distance around the centroid, with a preference for the placement directly over the centroid. Again, you can define whether the centroid is the one of the *visible polygon* or the *whole polygon*, and whether to force the centroid point inside the polygon.
- *Horizontal*: places at the best position a horizontal label inside the polygon. The preferred placement is further from the edges of the polygon. It's possible to *Allow placing labels outside of polygons*.
- *Free (Angled)*: places at the best position a rotated label inside the polygon. The rotation respects the polygon's orientation and the preferred placement is further from the edges of the polygon. It's possible to *Allow placing labels outside of polygons*.
- *Using Perimeter*: draws the label parallel to a generalised line representing the polygon boundary, with preference for straighter portions of the perimeter. You can define:
 - *Allowed positions*: *Above line*, *On line*, *Below line* and *Line orientation dependent position* (placing the label at the left or the right of the polygon's boundary). It's possible to select several options at once. In that case, QGIS will look for the optimal label position.

- *Distance* between the label and the polygon's outline
- the *Repeating Labels Distance* to display multiple times the label over the length of the perimeter.
- *Using Perimeter (Curved)*: draws the label following the curvature of the polygon's boundary. In addition to the parameters available with the *Using Perimeter* mode, you can set the *Maximum angle between curved characters polygon*, either inside or outside.
- *Outside Polygons*: always places labels outside the polygons, at a set *Distance*

Common placement settings

Some label placement settings are available for all layer geometry types:

Data Defined

The *Data Defined* group provides direct control on labels placement, on a feature-by-feature basis. It relies on their attributes or an expression to set:

- the *X* and *Y* coordinate
- the text alignment over the custom position set above:
 - *Horizontal*: it can be **Left**, **Center** or **Right**
 - the text *Vertical*: it can be **Bottom**, **Base**, **Half**, **Cap** or **Top**
- the text *Rotation*. Check the *Preserve data rotation values* entry if you want to keep the rotation value in the associated field and apply it to the label, whether the label is pinned or not. If unchecked, unpinning the label rotation is reset and its value cleared from the attribute table.

注解: Data-defined rotation with polygon features is currently supported only with the *Around centroid* placement mode.

注解: Expressions can not be used in combination with the labels map tools (ie the *Rotate label* and *Move label* tools) to *data-define* labels placement. The widget will be reset to the corresponding *auxiliary storage field*.

Priority

In the *Priority* section you can define the placement priority rank of each label, ie if there are different diagrams or labels candidates for the same location, the item with the higher priority will be displayed and the others could be left out.


The priority rank is also used to evaluate whether a label could be omitted due to a greater weighted *obstacle feature*.

Obstacles

In some contexts (eg, high density labels, overlapping features...), the labels placement can result in labels being placed over unrelated features.

An obstacle is a feature over which QGIS avoids placing other features' labels or diagrams. This can be controlled from the *Obstacles* section:


1. Activate the *Features act as obstacles* option to decide that features of the layer should act as obstacles for any label and diagram (including items from other features in the same layer).

Instead of the whole layer, you can select a subset of features to use as obstacles, using the  data-defined override control next to the option.

2. Use the *Settings* button to tweak the obstacle's weighting.
 - For every potential obstacle feature you can assign an *Obstacle weight*: any *label* or *diagram* whose placement priority rank is greater than this value can be placed over. Labels or diagrams with lower rank will be omitted if no other placement is possible.

This weighting can also be data-defined, so that within the same layer, certain features are more likely to be covered than others.
 - For polygon layers, you can choose the kind of obstacle the feature is:
 - **over the feature's interior**: avoids placing labels over the interior of the polygon (prefers placing labels totally outside or just slightly inside the polygon)
 - or **over the feature's boundary**: avoids placing labels over the boundary of the polygon (prefers placing labels outside or completely inside the polygon). This can be useful for layers where the features cover the whole area (administrative units, categorical coverages, ...). In this case, it is impossible to avoid placing labels within these features, and it looks much better when placing them over the boundaries between features is avoided.

Rendering tab

In the  *Rendering* tab, you can tune when the labels can be rendered and their interaction with other labels and features.

Label options


Under *Label options*:

- You find the *scale-based* and the *Pixel size-based* visibility settings.
- The *Label z-index* determines the order in which labels are rendered, as well in relation with other feature labels in the layer (using data-defined override expression), as with labels from other layers. Labels with a higher z-index are rendered on top of labels (from any layer) with lower z-index.

Additionally, the logic has been tweaked so that if two labels have matching z-indexes, then:

- if they are from the same layer, the smaller label will be drawn above the larger label
- if they are from different layers, the labels will be drawn in the same order as their layers themselves (ie respecting the order set in the map legend).

注解: This setting doesn't make labels to be drawn below the features from other layers, it just controls the order in which labels are drawn on top of all the layers' features.

- While rendering labels and in order to display readable labels, QGIS automatically evaluates the position of the labels and can hide some of them in case of collision. You can however choose to  *Show all labels for this layer (including colliding labels)* in order to manually fix their placement (see *The Label Toolbar*).
- With data-defined expressions in *Show label* and *Always Show* you can fine tune which labels should be rendered.
- Allow to *Show upside-down labels*: alternatives are **Never**, **when rotation defined** or **always**.



Feature options

Under *Feature options*:

- You can choose to *Label every part of a multi-part features* and *Limit number of features to be labeled to*.
- Both line and polygon layers offer the option to set a minimum size for the features to be labeled, using *Suppress labeling of features smaller than*.
- For polygon features, you can also filter the labels to show according to whether they completely fit within their feature or not.
- For line features, you can choose to *Merge connected lines to avoid duplicate labels*, rendering a quite airy map in conjunction with the *Distance* or *Repeat* options in the *Placement* tab.

12.4 Creating 3D Symbols

The *Style Manager* helps you create and store 3D symbols for every geometry type to render in the *3D map view*.

As of the other items, enable the  *3D Symbols* tab and expand the  button menu to create:

- *3D point symbols*
- *3D line symbols*
- *3D polygon symbols*

12.4.1 Point Layers

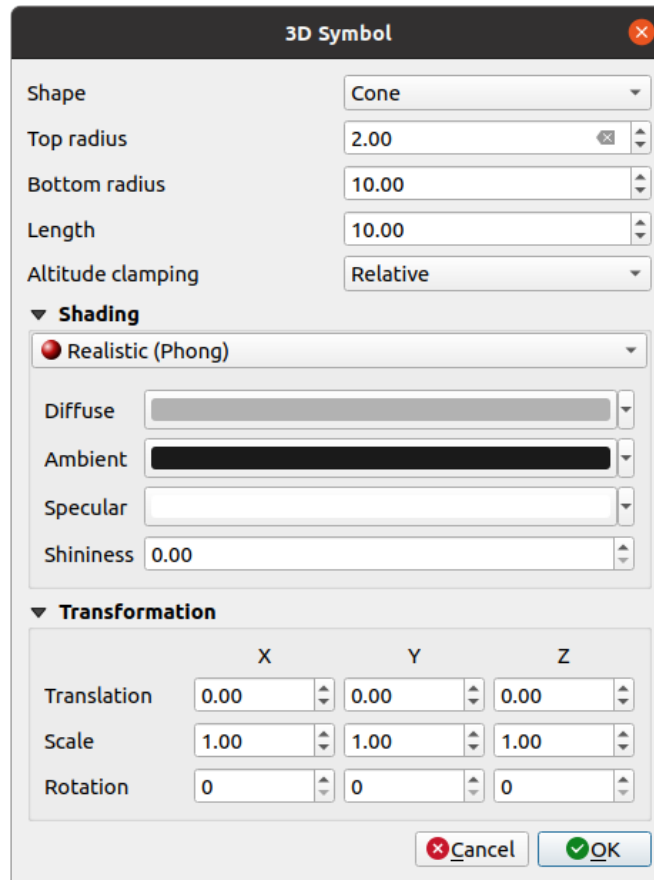


图 12.24: Properties of a 3D point symbol

- You can define different simple 3D shapes like *Sphere*, *Cylinder*, *Cube*, *Cone*, *Plane* and *Torus* defined by their *Radius*, *Size* or *Length*. The unit of size of the 3D shapes refers to the CRS of the project.
- The shading of the 3D shapes can be defined by the menus *Diffuse*, *Ambient*, *Specular* and *Shininess* (see https://en.wikipedia.org/wiki/Phong_reflection_model#Description)
- If you choose *3D Model*, the location will be determined by a simple point coordinate.
- For visualizing 3D point clouds you can use *Billboard* Shapes defined by the *Billboard Height*, *Billboard symbol* and *Altitude clamping*. The symbol will have a stable size.
- *Altitude clamping* can be set to *Absolute*, *Relative* or *Terrain*. The *Absolute* setting can be used when height values of the 3d vectors are provided as absolute measures from 0. *Relative* and *Terrain* add given elevation values to the underlying terrain elevation.
- *Translation* can be used to move objects in x, y and z axis.
- You can define a *Scale factor* for the 3D shape as well as a *Rotation* around the x-, y- and z-axis.

12.4.2 Line layers

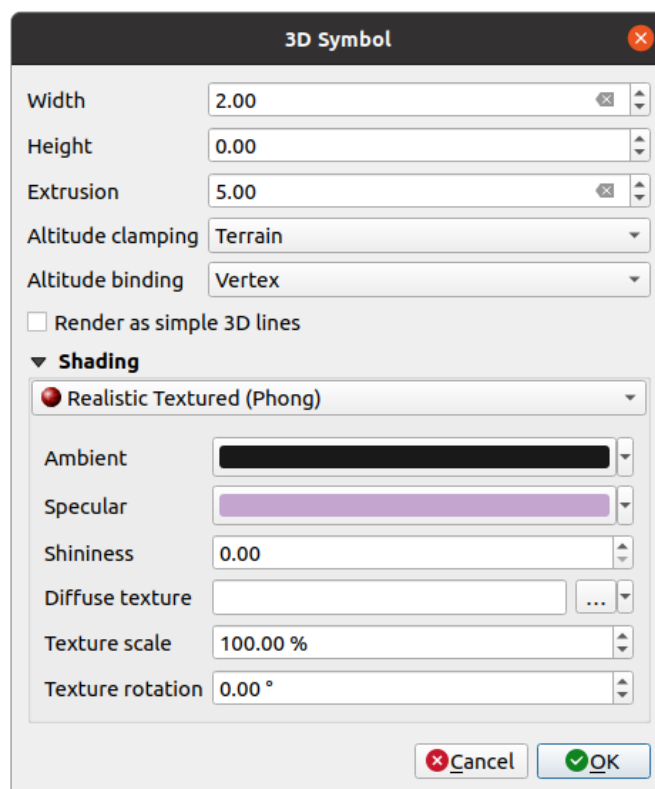


图 12.25: Properties of a 3D line symbol

- Beneath the *Width* and *Height* settings you can define the *Extrusion* of the vector lines. If the lines do not have z-values, you can define the 3d volumes with this setting.
- With the *Altitude clamping* you define the position of the 3D lines relative to the underlying terrain surface, if you have included raster elevation data or other 3D vectors.
- The *Altitude binding* defines how the feature is clamped to the terrain. Either every *Vertex* of the feature will be clamped to the terrain or this will be done by the *Centroid*.
- It is possible to *Render as simple 3D lines*.
- The shading can be defined in the menus *Diffuse*, *Ambient*, *Specular* and *Shininess*.

12.4.3 Polygon Layers

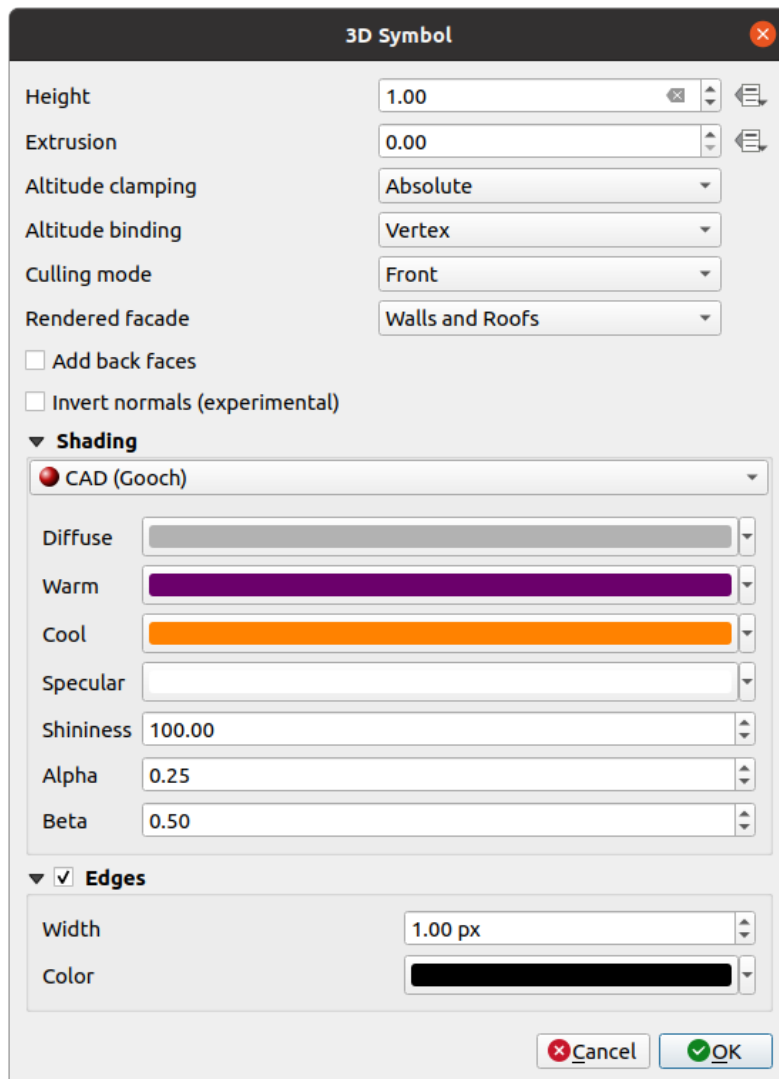




图 12.26: Properties of a 3D polygon symbol

- As for the other ones, *Height* can be defined in CRS units. You can also use the  button to overwrite the value with a custom expression, a variable or an entry of the attribute table
- Again, *Extrusion* is possible for missing z-values. Also for the extrusion you can use the  button in order to use the values of the vector layer and have different results for each polygon:

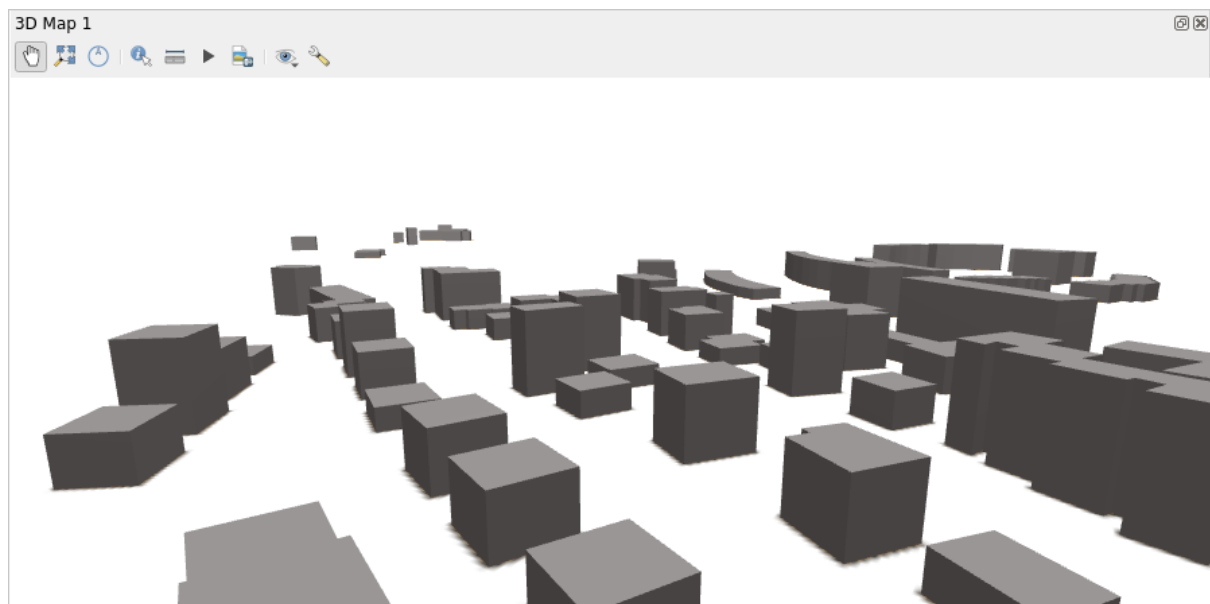


图 12.27: Data Defined Extrusion

- The *Altitude clamping*, *Altitude binding* can be defined as explained above.
- There is an additional option to *Add back faces* and *Invert normals*.
- You can define *Edges* by *Width* and *Color*.

12.4.4 Application example

To go through the settings explained above you can have a look at <https://public.cloudmergin.com/projects/saber/luxembourg/tree>.


13.1 Opening Data

As part of an Open Source Software ecosystem, QGIS is built upon different libraries that, combined with its own providers, offer capabilities to read and often write a lot of formats:

- Vector data formats include GeoPackage, GML, GeoJSON, GPX, KML, Comma Separated Values, ESRI formats (Shapefile, Geodatabase...), MapInfo and MicroStation file formats, AutoCAD DWG/DXF, GRASS and many more... Read the complete list of [supported vector formats](#).
- Raster data formats include GeoTIFF, JPEG, ASCII Gridded XYZ, MBTiles, R or Idrisi rasters, GDAL Virtual, SRTM, Sentinel Data, ERDAS IMAGINE, ArcInfo Binary Grid, ArcInfo ASCII Grid, and many more... Read the complete list of [supported raster formats](#).
- Database formats include PostgreSQL/PostGIS, SQLite/SpatiaLite, Oracle, DB2 or MSSQL Spatial, MySQL...
- Web map and data services (WM(T)S, WFS, WCS, CSW, XYZ tiles, ArcGIS services, ...) are also handled by QGIS providers. See [Working with OGC / ISO protocols](#) for more information about some of these.
- You can read supported files from archived folders and use QGIS native formats such as QML files ([QML - QGIS 样式文件格式](#)) and virtual and memory layers.

More than 80 vector and 140 raster formats are supported by [GDAL](#) and QGIS native providers.

注解: Not all of the listed formats may work in QGIS for various reasons. For example, some require external proprietary libraries, or the GDAL/OGR installation of your OS may not have been built to support the format you want to use. To see the list of available formats, run the command line `ogrinfo --formats` (for vector) and `gdalinfo --formats` (for raster), or check the *Settings* ► *Options* ► *GDAL* menu in QGIS.

In QGIS, depending on the data format, there are different tools to open a dataset, mainly available in the *Layer* ► *Add Layer* ► menu or from the *Manage Layers* toolbar (enabled through *View* ► *Toolbars* menu). However, all these tools point to a unique dialog, the *Data Source Manager* dialog, that you can open with the  Open Data Source Manager button, available on the *Data Source Manager Toolbar*, or by pressing `Ctrl+L`. The *Data Source Manager* dialog ([图 13.1](#)) offers a unified interface to open vector or raster file-based data as well as databases or web services supported by QGIS. It can be set modal or not with the *Modeless data source manager dialog* in the *Settings* ► *Options* ► *General* menu.

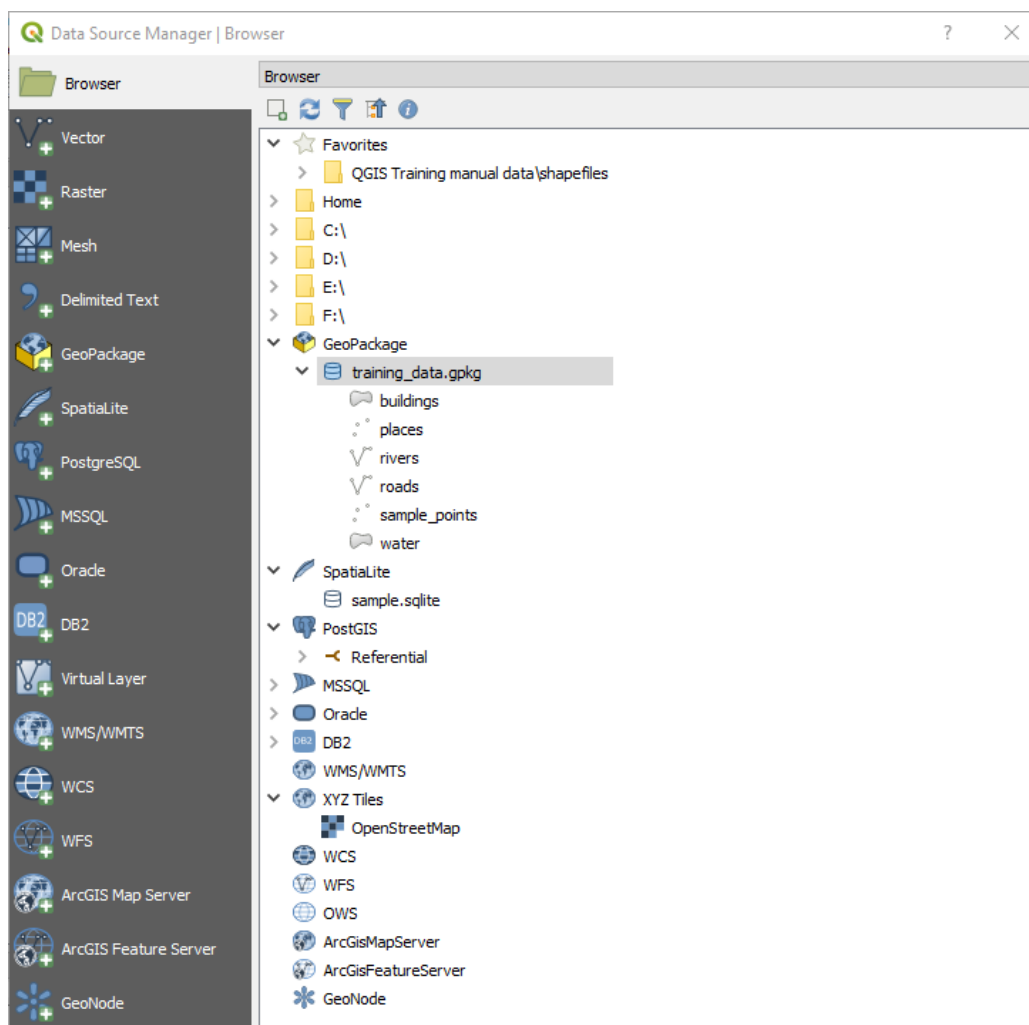



图 13.1: QGIS Data Source Manager dialog

Beside this main entry point, you also have the  *DB Manager* plugin that offers advanced capabilities to analyze and manipulate connected databases. More information on DB Manager capabilities can be found in *DB Manager Plugin*.

There are many other tools, native or third-party plugins, that help you open various data formats.

This chapter will describe only the tools provided by default in QGIS for loading data. It will mainly focus on the *Data Source Manager* dialog but more than describing each tab, it will also explore the tools based on the data provider or format specificities.

13.1.1 The Browser Panel






The *Browser* is one of the main ways to quickly and easily add your data to projects. It's available as:

- a *Data Source Manager* tab, enabled pressing the  Open Data Source Manager button (Ctrl+L);
- as a QGIS panel you can open from the menu *View* ► *Panels* (or  *Settings* ► *Panels*) or by pressing Ctrl+2.

In both cases, the *Browser* helps you navigate in your file system and manage geodata, regardless the type of layer (raster, vector, table), or the datasource format (plain or compressed files, databases, web services).

Exploring the Interface




At the top of the Browser panel, you find some buttons that help you to:

-  **Add Selected Layers**: you can also add data to the map canvas by selecting **Add selected layer(s)** from the layer's context menu;
-  **Refresh** the browser tree;
-  **Filter Browser** to search for specific data. Enter a search word or wildcard and the browser will filter the tree to only show paths to matching DB tables, filenames or folders -- other data or folders won't be displayed. See the Browser Panel(2) example in [Figure 13.2](#). The comparison can be case-sensitive or not. It can also be set to:
 - *Normal*: show items containing the search text
 - *Wildcard(s)*: fine tune the search using the ? and/or * characters to specify the position of the search text
 - *Regular expression*
-  **Collapse All** the whole tree;
-  **Enable/disable properties widget**: when toggled on, a new widget is added at the bottom of the panel showing, if applicable, metadata for the selected item.

The entries in the *Browser* panel are organised hierarchically, and there are several top level entries:

1. *Favorites* where you can place shortcuts to often used locations
2. *Spatial Bookmarks* where you can store often used map extents (see [Spatial Bookmarks](#))
3. *Project Home*: for a quick access to the folder in which (most of) the data related to your project are stored. The default value is the directory where your project file resides.
4. *Home* directory in the file system and the filesystem root directory.
5. Connected local or network drives
6. Then comes a number of container / database types and service protocols, depending on your platform and underlying libraries:

-  *GeoPackage*
-  *SpatiaLite*
-  *PostGIS*
-  *MSSQL*
-  *Oracle*
-  *DB2*
-  *WMS/WMTS*
-  *Vector Tiles*
-  *XYZ Tiles*
-  *WCS*
-  *WFS/OGC API-Features*
-  *OWS*

-  *ArcGIS Map Service*
-  *ArcGIS Feature Service*
-  *GeoNode*

Interacting with the Browser items

The browser supports drag and drop within the browser, from the browser to the canvas and *Layers* panel, and from the *Layers* panel to layer containers (e.g. GeoPackage) in the browser.

Project file items inside the browser can be expanded, showing the full layer tree (including groups) contained within that project. Project items are treated the same way as any other item in the browser, so they can be dragged and dropped within the browser (for example to copy a layer item to a geopackage file) or added to the current project through drag and drop or double click.

The context menu for an element in the *Browser* panel is opened by right-clicking on it.

For file system directory entries, the context menu offers the following:

- *New* ► to create in the selected entry a:
 - *Directory...*
 - *GeoPackage...*
 - *ShapeFile...*
- *Add as a Favorite*: favorite folders can be renamed (*Rename favorite...*) or removed (*Remove favorite*) any time.
- *Hide from Browser*: hidden folders can be toggled to visible from the *Settings* ► *Options* ► *Data Sources* ► *Hidden browser paths* setting
- *Fast Scan this Directory*
- *Open Directory*
- *Open in Terminal*
- *Properties...*
- *Directory Properties...*

For leaf entries that can act as layers in the project, the context menu will have supporting entries. For example, for non-database, non-service-based vector, raster and mesh data sources:

- *Delete File "<name of file>"...*
- *Export Layer --> To File...*
- *Add Layer to Project*
- *Layer Properties*
- *File Properties*

In the *Layer properties* entry, you will find (similar to what you will find in the *vector* and *raster* layer properties once the layers have been added to the project):

- *Metadata* for the layer. Metadata groups: *Information from provider* (if possible, *Path* will be a hyperlink to the source), *Identification*, *Extent*, *Access*, *Fields* (for vector layers), *Bands* (for raster layers), *Contacts*, *Links* (for vector layers), *References* (for raster layers), *History*.
- A *Preview* panel
- The attribute table for vector sources (in the *Attributes* panel).

To add a layer to the project using the *Browser*:

1. Enable the *Browser* as described above. A browser tree with your file system, databases and web services is displayed. You may need to connect databases and web services before they appear (see dedicated sections).
2. Find the layer in the list.
3. Use the context menu, double-click its name, or drag-and-drop it into the *map canvas*. Your layer is now added to the *Layers panel* and can be viewed on the map canvas.

小技巧: Open a QGIS project directly from the browser

You can also open a QGIS project directly from the Browser panel by double-clicking its name or by drag-and-drop into the map canvas.

Once a file is loaded, you can zoom around it using the map navigation tools. To change the style of a layer, open the *Layer Properties* dialog by double-clicking on the layer name or by right-clicking on the name in the legend and choosing *Properties* from the context menu. See section *Symbology Properties* for more information on setting symbology for vector layers.

Right-clicking an item in the browser tree helps you to:

- for a file or a table, display its metadata or open it in your project. Tables can even be renamed, deleted or truncated.
- for a folder, bookmark it into your favourites or hide it from the browser tree. Hidden folders can be managed from the *Settings* ► *Options* ► *Data Sources* tab.
- manage your *spatial bookmarks*: bookmarks can be created, exported and imported as XML files.
- create a connection to a database or a web service.
- refresh, rename or delete a schema.

You can also import files into databases or copy tables from one schema/database to another with a simple drag-and-drop. There is a second browser panel available to avoid long scrolling while dragging. Just select the file and drag-and-drop from one panel to the other.

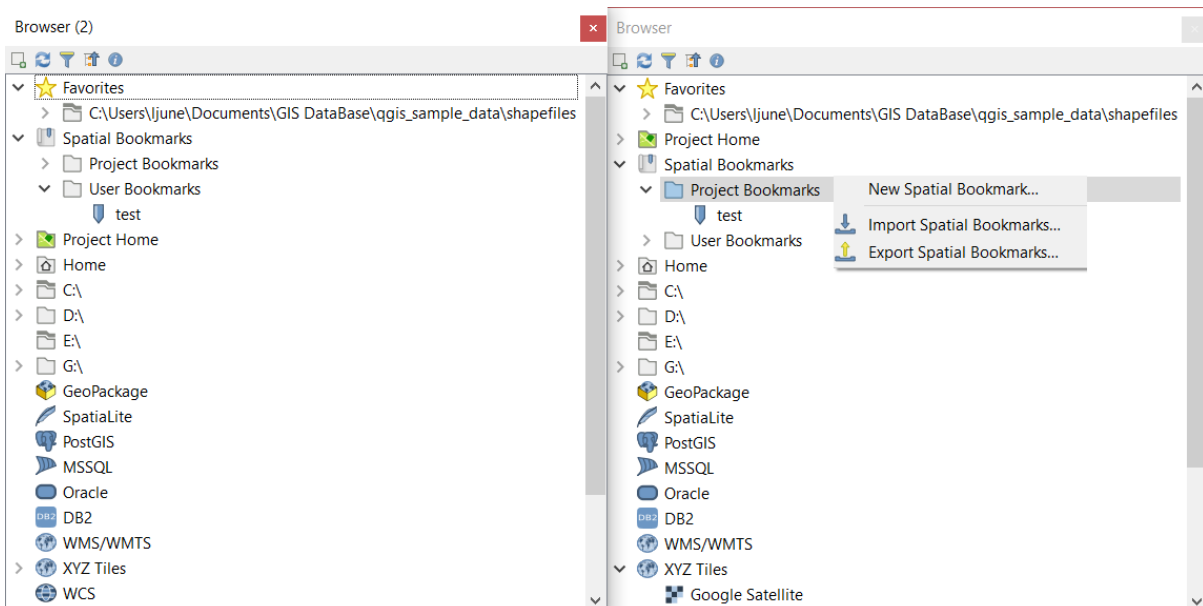



图 13.2: QGIS Browser panels side-by-side

小技巧: Add layers to QGIS by simple drag-and-drop from your OS file browser

You can also add file(s) to the project by drag-and-dropping them from your operating system file browser to the *Layers Panel* or the map canvas.

13.1.2 The DB Manager

The *DB Manager* Plugin is another tool for integrating and managing spatial database formats supported by QGIS (PostGIS, SpatiaLite, GeoPackage, Oracle Spatial, MSSQL, DB2, Virtual layers). It can be activated from the *Plugins* ► *Manage and Install Plugins...* menu.

The  DB Manager Plugin provides several features:

- connect to databases and display their structure and contents
- preview tables of databases
- add layers to the map canvas, either by double-clicking or drag-and-drop.
- add layers to a database from the QGIS Browser or from another database
- create SQL queries and add their output to the map canvas
- create *virtual layers*

More information on DB Manager capabilities is found in *DB Manager Plugin*.

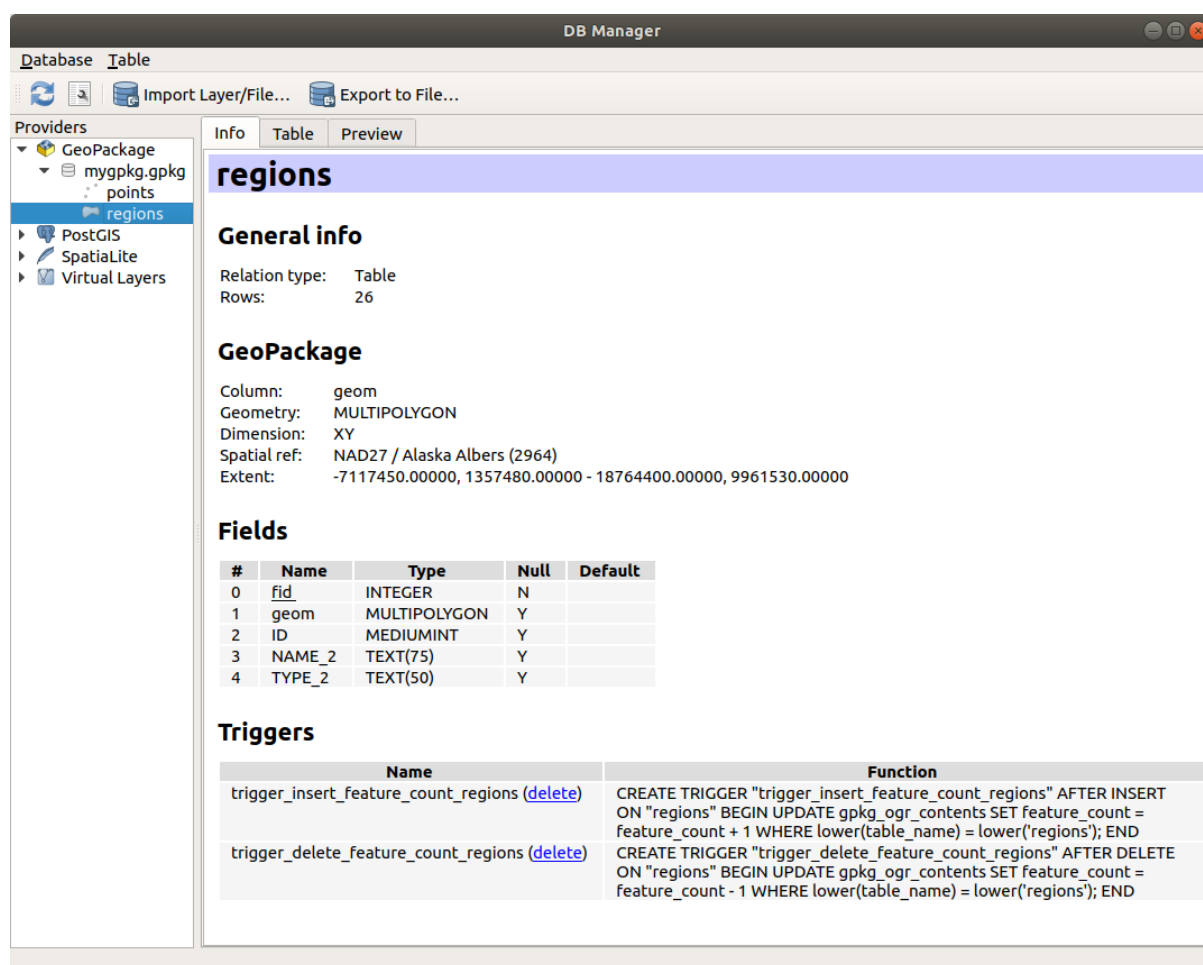


图 13.3: DB Manager dialog




13.1.3 Provider-based loading tools

Beside the Browser Panel and the DB Manager, the main tools provided by QGIS to add layers, you'll also find tools that are specific to data providers.

注解: Some *external plugins* also provide tools to open specific format files in QGIS.

Loading a layer from a file

To load a layer from a file:

1. Open the layer type tab in the *Data Source Manager* dialog, ie click the  Open Data Source Manager button (or press `Ctrl+L`) and enable the target tab or:
 - for vector data (like GML, ESRI Shapefile, Mapinfo and DXF layers): press `Ctrl+Shift+V`, select the *Layer* ► *Add Layer* ►  *Add Vector Layer* menu option or click on the  Add Vector Layer toolbar button.

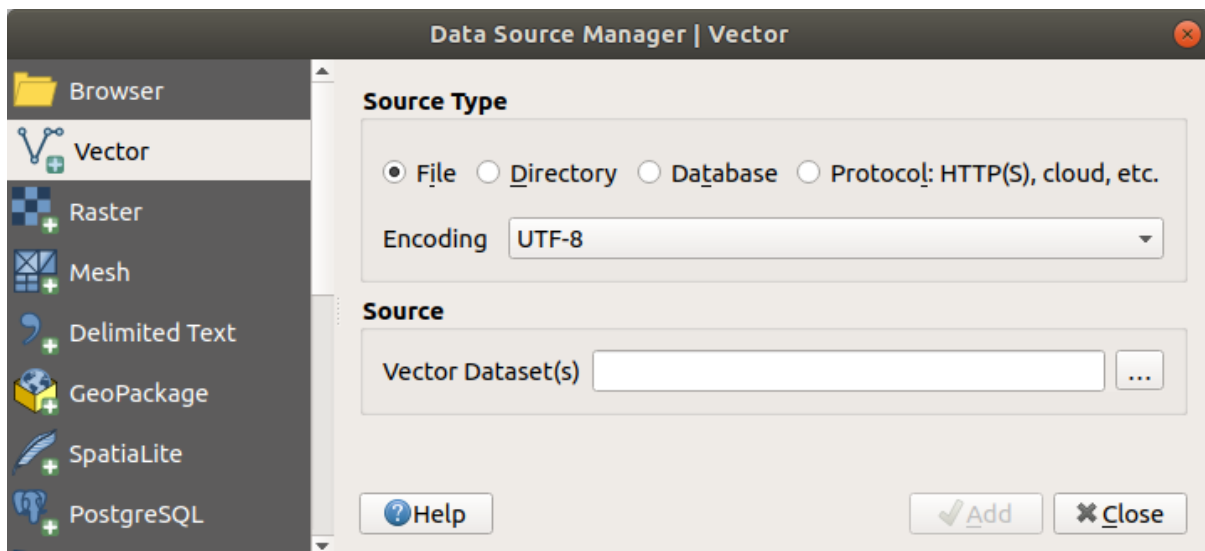




图 13.4: Add Vector Layer Dialog

- for raster data (like GeoTiff, MBTiles, GRIdded Binary and DWG layers): press `Ctrl+Shift+R`, select the *Layer* ► *Add Layer* ►  *Add Raster Layer* menu option or click on the  Add Raster Layer toolbar button.

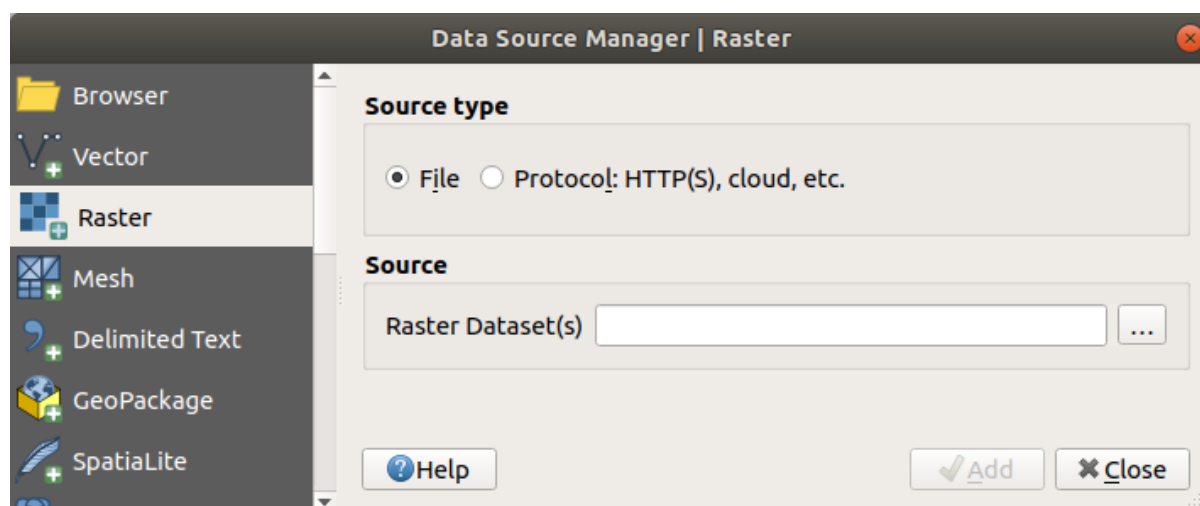


图 13.5: Add Raster Layer Dialog

2. Check *File* source type
3. Click on the ... ^{Browse} button
4. Navigate the file system and load a supported data source. More than one layer can be loaded at the same time by holding down the `Ctrl` key and clicking on multiple items in the dialog or holding down the `Shift` key to select a range of items by clicking on the first and last items in the range. Only formats that have been well tested appear in the formats filter. Other formats can be loaded by selecting `All files` (the top item in the pull-down menu).
5. Press *Open* to load the selected file into *Data Source Manager* dialog

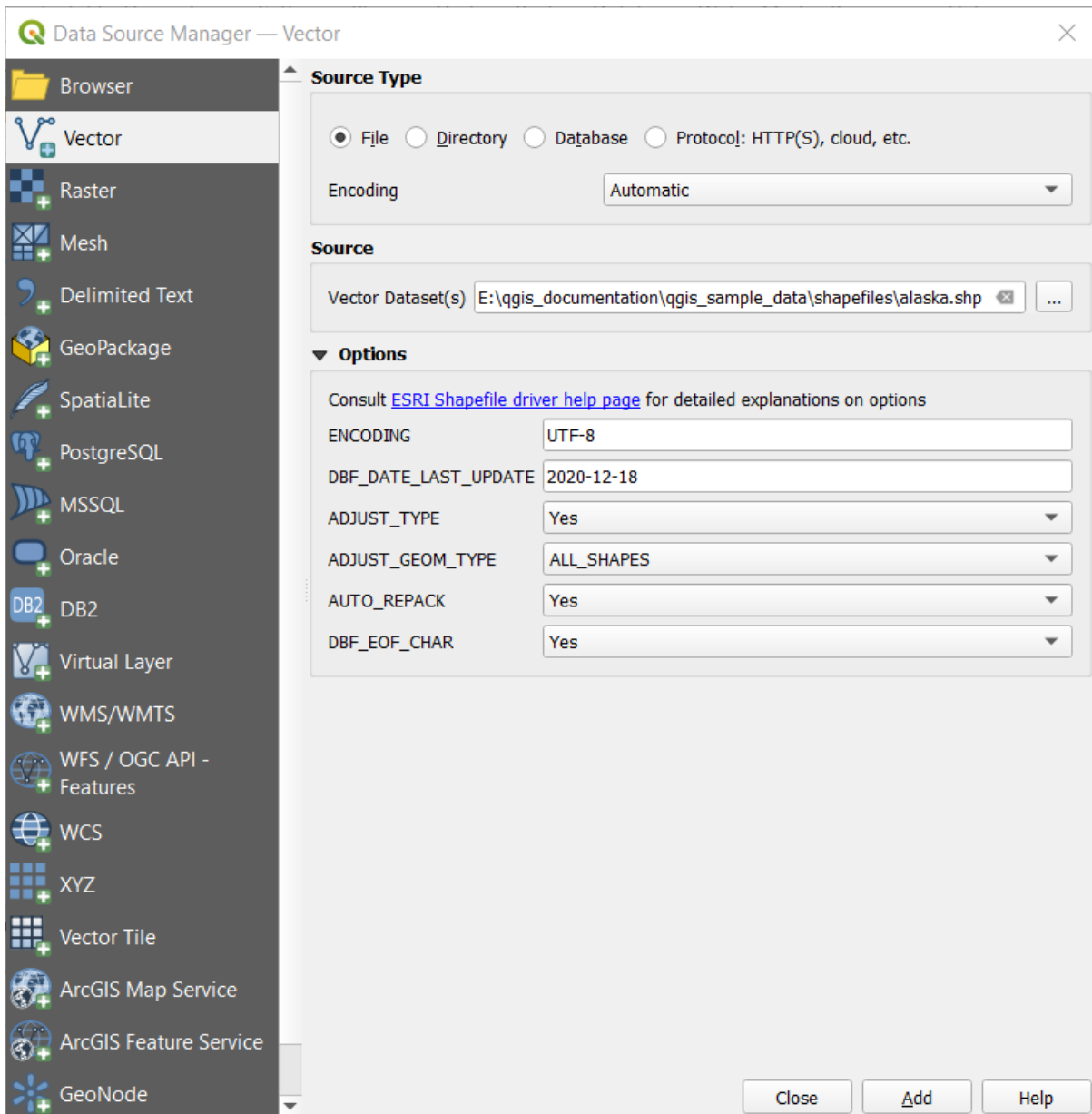


图 13.6: Loading a Shapefile with open options

6. Press *Add* to load the file in QGIS and display them in the map view. 图 13.7 shows QGIS after loading the `alaska.shp` file.

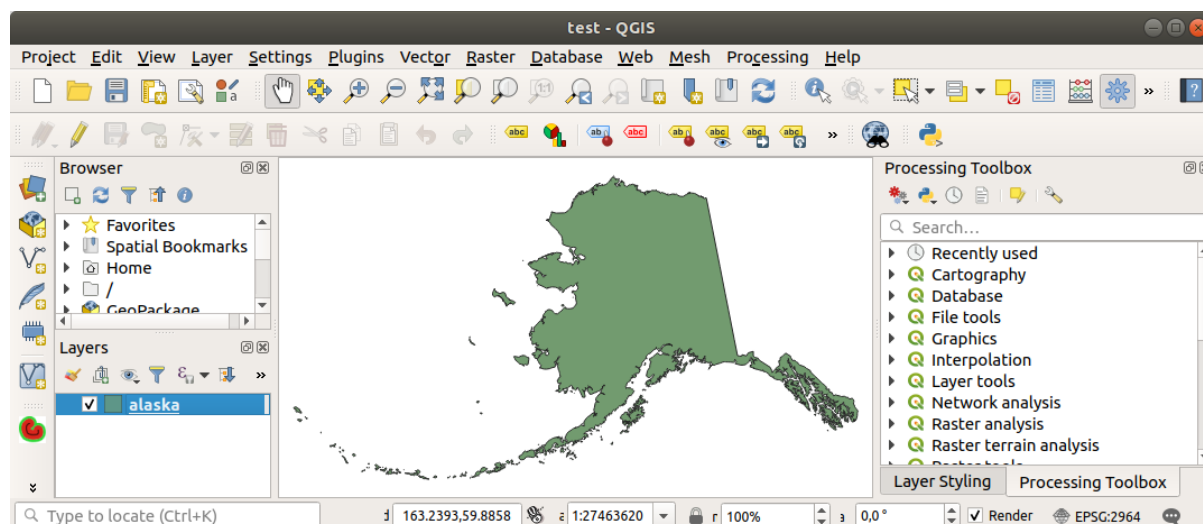


图 13.7: QGIS with Shapefile of Alaska loaded

注解: For loading vector files the GDAL driver offers to define open actions. These will be shown when the vector file is selected. Options are described in detail on <https://gdal.org/drivers/vector/>.

注解: Because some formats like MapInfo (e.g., .tab) or Autocad (.dxf) allow mixing different types of geometry in a single file, loading such datasets opens a dialog to select geometries to use in order to have one geometry per layer.



The  Add Vector Layer and  Add Raster Layer tabs allow loading of layers from source types other than *File*:

- You can load specific vector formats like ArcInfo Binary Coverage, UK. National Transfer Format, as well as the raw TIGER format of the US Census Bureau or OpenfileGDB. To do that, you select *Directory* as *Source type*. In this case, a directory can be selected in the dialog after pressing ... Browse.
- With the *Database* source type you can select an existing database connection or create one to the selected database type. Some possible database types are ODBC, Esri Personal Geodatabase, MSSQL as well as PostgreSQL or MySQL.
Pressing the *New* button opens the *Create a New OGR Database Connection* dialog whose parameters are among the ones you can find in *Creating a stored Connection*. Pressing *Open* lets you select from the available tables, for example of PostGIS enabled databases.
- The *Protocol: HTTP(S), cloud, etc.* source type opens data stored locally or on the network, either publicly accessible, or in private buckets of commercial cloud storage services. Supported protocol types are:
 - HTTP/HTTPS/FTP, with a *URI* and, if required, an *authentication*.
 - Cloud storage such as AWS S3, Google Cloud Storage, Microsoft Azure Blob, Alibaba OSS Cloud, Open Stack Swift Storage. You need to fill in the *Bucket or container* and the *Object key*.
 - service supporting OGC WFS 3 (still experimental), using GeoJSON or GEOJSON – Newline Delimited format or based on CouchDB database. A *URI* is required, with optional *authentication*.
 - For all vector source types it is possible to define the *Encoding* or to use the *Automatic* ► setting.

Loading a mesh layer

A mesh is an unstructured grid usually with temporal and other components. The spatial component contains a collection of vertices, edges and faces in 2D or 3D space. More information on mesh layers at [Working with Mesh Data](#).

To add a mesh layer to QGIS:

1. Open the *Data Source Manager* dialog, either by selecting it from the *Layer* ► menu or clicking the  button.
2. Enable the  *Mesh* tab on the left panel
3. Press the ... ^{Browse} button to select the file. *Various formats* are supported.
4. Select the layer and press *Add*. The layer will be added using the native mesh rendering.

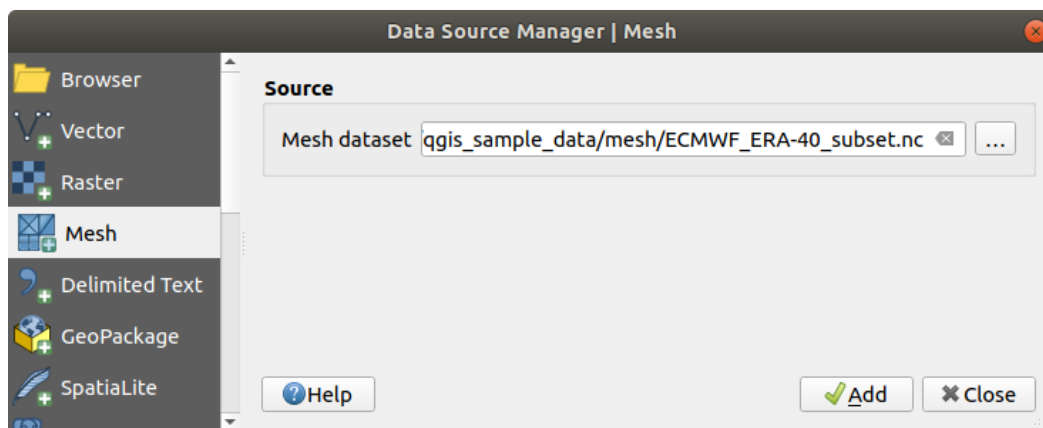





图 13.8: Mesh tab in Data Source Manager

Importing a delimited text file

Delimited text files (e.g. `.txt`, `.csv`, `.dat`, `.wkt`) can be loaded using the tools described above. This way, they will show up as simple tables. Sometimes, delimited text files can contain coordinates / geometries that you could want to visualize. This is what  *Add Delimited Text Layer* is designed for.

1. Click the  *Open Data Source Manager* icon to open the *Data Source Manager* dialog
2. Enable the  *Delimited Text* tab
3. Select the delimited text file to import (e.g., `qgis_sample_data/csv/elevp.csv`) by clicking on the ... ^{Browse} button.
4. In the *Layer name* field, provide the name to use for the layer in the project (e.g. *Elevation*).
5. Configure the settings to meet your dataset and needs, as explained below.

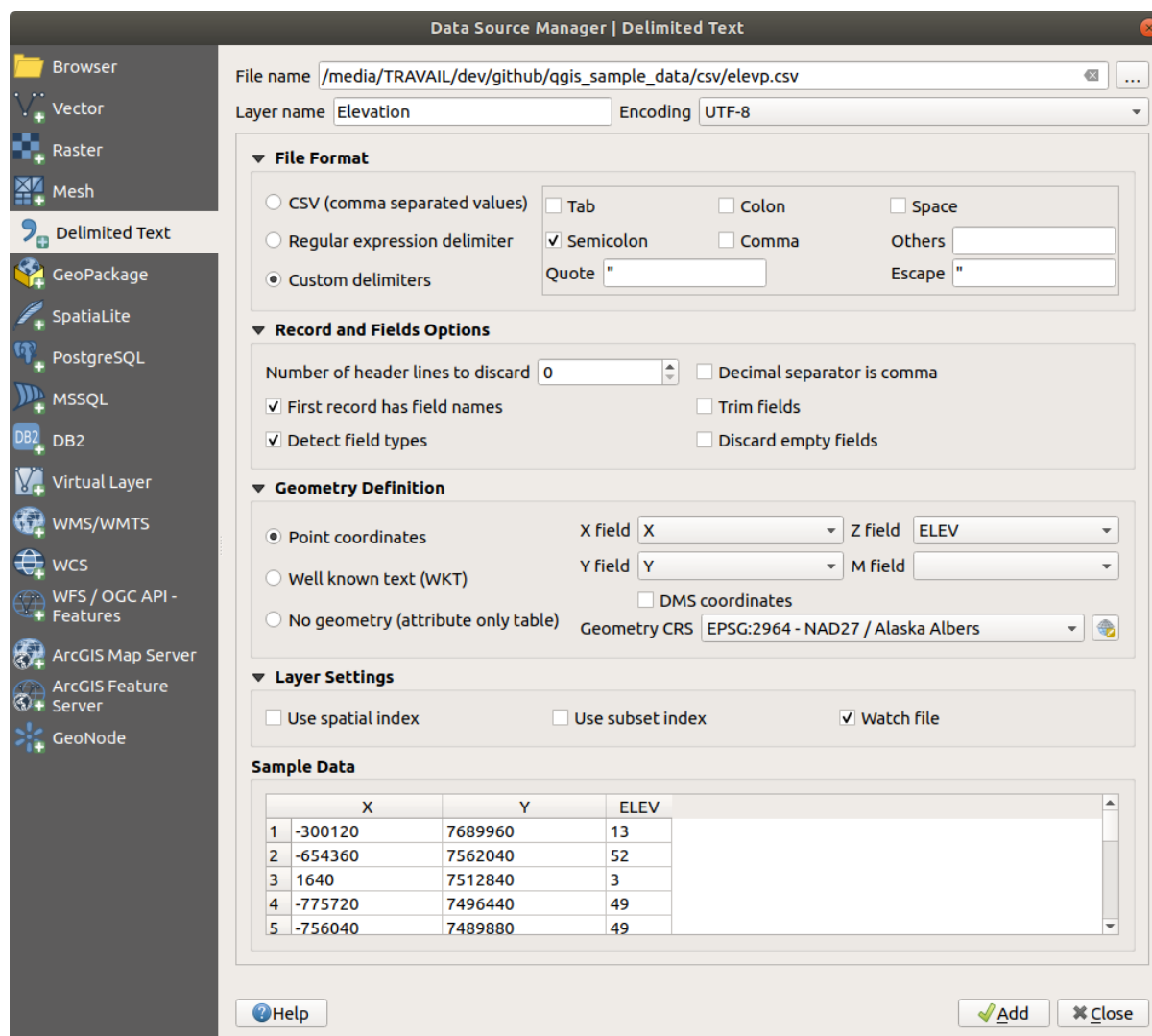


图 13.9: Delimited Text Dialog

File format

Once the file is selected, QGIS attempts to parse the file with the most recently used delimiter, identifying fields and rows. To enable QGIS to correctly parse the file, it is important to select the right delimiter. You can specify a delimiter by choosing between:

- *CSV (comma separated values)* to use the comma character.
- *Regular expression delimiter* and enter text into the *Expression* field. For example, to change the delimiter to tab, use `\t` (this is used in regular expressions for the tab character).
- *Custom delimiters*, choosing among some predefined delimiters like comma, space, tab, semicolon, ...

Records and fields



Some other convenient options can be used for data recognition:

- *Number of header lines to discard*: convenient when you want to avoid the first lines in the file in the import, either because those are blank lines or with another formatting.
- *First record has field names*: values in the first line are used as field names, otherwise QGIS uses the field names `field_1`, `field_2`...
- *Detect field types*: automatically recognizes the field type. If unchecked then all attributes are treated as text fields.
- *Decimal separator is comma*: you can force decimal separator to be a comma.
- *Trim fields*: allows you to trim leading and trailing spaces from fields.
- *Discard empty fields*.

As you set the parser properties, a sample data preview updates at the bottom of the dialog.

Geometry definition

Once the file is parsed, set *Geometry definition* to

- *Point coordinates* and provide the *X field*, *Y field*, *Z field* (for 3-dimensional data) and *M field* (for the measurement dimension) if the layer is of point geometry type and contains such fields. If the coordinates are defined as degrees/minutes/seconds, activate the *DMS coordinates* checkbox. Provide the appropriate *Geometry CRS* using the  *Select CRS* widget.
- *Well known text (WKT)* option if the spatial information is represented as WKT: select the *Geometry field* containing the WKT geometry and choose the appropriate *Geometry field* or let QGIS auto-detect it. Provide the appropriate *Geometry CRS* using the  *Select CRS* widget.
- If the file contains non-spatial data, activate *No geometry (attribute only table)* and it will be loaded as an ordinary table.

Layer settings

Additionally, you can enable:

- *Use spatial index* to improve the performance of displaying and spatially selecting features.
- *Use subset index* to improve performance of *subset filters* (when defined in the layer properties).
- *Watch file* to watch for changes to the file by other applications while QGIS is running.

At the end, click *Add* to add the layer to the map. In our example, a point layer named `Elevation` is added to the project and behaves like any other map layer in QGIS. This layer is the result of a query on the `.csv` source file (hence, linked to it) and would require *to be saved* in order to get a spatial layer on disk.

Importing a DXF or DWG file

DXF and DWG files can be added to QGIS by simple drag-and-drop from the Browser Panel. You will be prompted to select the sublayers you would like to add to the project. Layers are added with random style properties.

注解: For DXF files containing several geometry types (point, line and/or polygon), the name of the layers will be generated as `<filename.dxf> entities <geometry type>`.

To keep the dxf/dwg file structure and its symbology in QGIS, you may want to use the dedicated *Project ► Import/Export ► Import Layers from DWG/DXF...* tool which allows you to:

1. import elements from the drawing file into a GeoPackage database.
2. add imported elements to the project.

In the *DWG/DXF Import* dialog, to import the drawing file contents:

1. Input the location of the *Target package*, i.e. the new GeoPackage file that will store the data. If an existing file is provided, then it will be overwritten.
2. Specify the coordinate reference system of the data in the drawing file.
3. Check *Expand block references* to import the blocks in the drawing file as normal elements.
4. Check *Use curves* to promote the imported layers to a `curved` geometry type.
5. Use the *Import* button to select the DWG/DXF file to use (one per geopackage). The GeoPackage database will be automatically populated with the drawing file content. Depending on the size of the file, this can take some time.

After the `.dwg` or `.dxf` data has been imported into the GeoPackage database, the frame in the lower half of the dialog is populated with the list of layers from the imported file. There you can select which layers to add to the QGIS project:

1. At the top, set a *Group name* to group the drawing files in the project.
2. Check layers to show: Each selected layer is added to an ad hoc group which contains vector layers for the point, line, label and area features of the drawing layer. The style of the layers will resemble the look they originally had in *CAD.
3. Choose if the layer should be visible at opening.
4. Checking the *Merge layers* option places all layers in a single group.
5. Press *OK* to open the layers in QGIS.

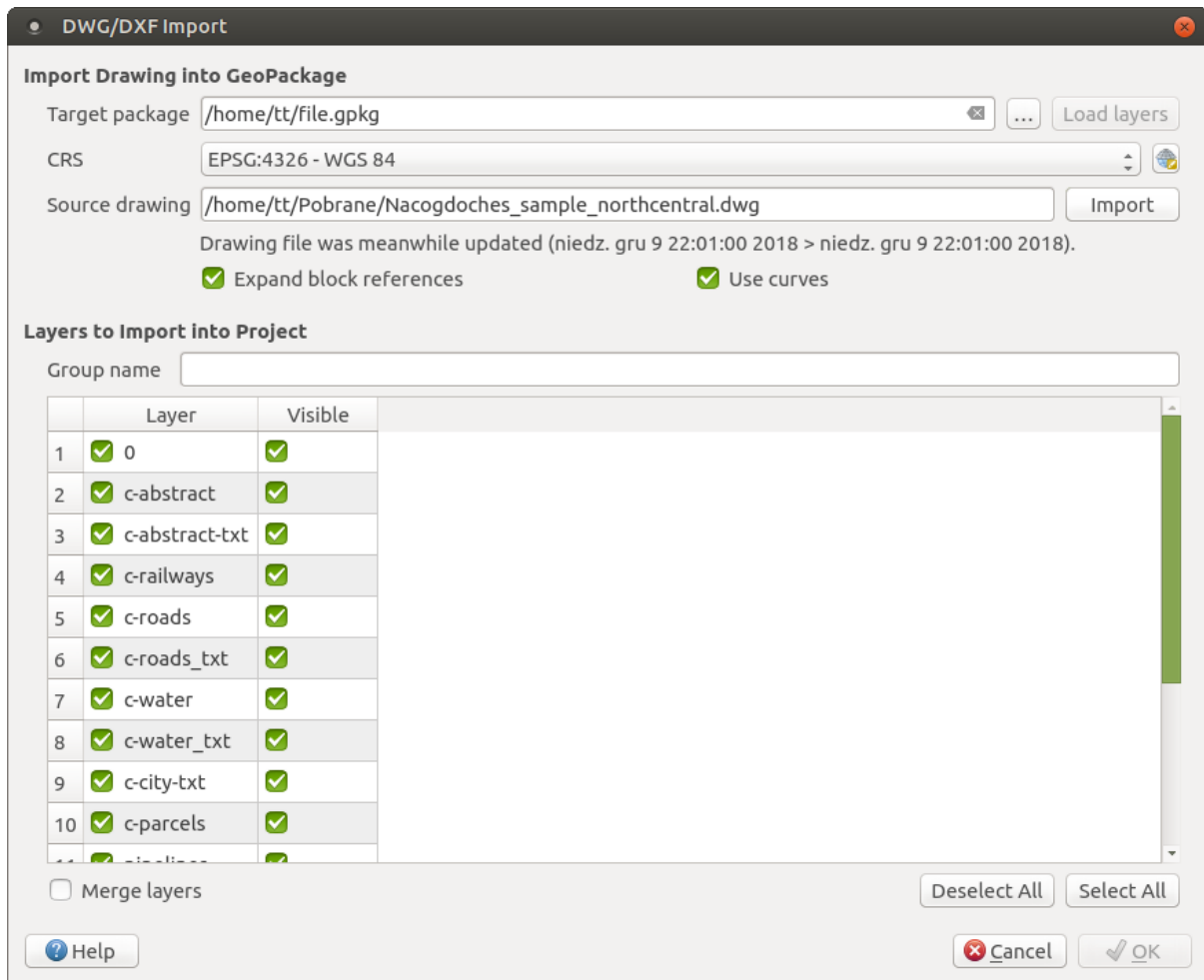



图 13.10: Import dialog for DWG/DXF files



Importing OpenStreetMap Vectors

The OpenStreetMap project is popular because in many countries no free geodata such as digital road maps are available. The objective of the OSM project is to create a free editable map of the world from GPS data, aerial photography and local knowledge. To support this objective, QGIS provides support for OSM data.

Using the *Browser Panel*, you can load an `.osm` file to the map canvas, in which case you'll get a dialog to select sublayers based on the geometry type. The loaded layers will contain all the data of that geometry type in the `.osm` file, and keep the `osm` file data structure.

Spatialite Layers

 The first time you load data from a Spatialite database, begin by:

- clicking on the  Add Spatialite Layer toolbar button
- selecting the  Add Spatialite Layer... option from the *Layer* ► *Add Layer* menu
- or by typing `Ctrl+Shift+L`

This will bring up a window that will allow you either to connect to a Spatialite database already known to QGIS (which you choose from the drop-down menu) or to define a new connection to a new database. To define a new

connection, click on *New* and use the file browser to point to your SpatiaLite database, which is a file with a `.sqlite` extension.

QGIS also supports editable views in SpatiaLite.

GPS

Loading GPS data in QGIS can be done using the core plugin `GPS Tools`. Instructions are found in section *GPS Plugin*.





GRASS

Working with GRASS vector data is described in section *GRASS GIS Integration*.

Database related tools

Creating a stored Connection

In order to read and write tables from a database format QGIS supports you have to create a connection to that database. While *QGIS Browser Panel* is the simplest and recommended way to connect to and use databases, QGIS provides other tools to connect to each of them and load their tables:

-  `Add PostGIS Layer...` or by typing `Ctrl+Shift+D`
-  `Add MSSQL Spatial Layer`
-  `Add Oracle Spatial Layer...` or by typing `Ctrl+Shift+O`
-  `Add DB2 Spatial Layer...` or by typing `Ctrl+Shift+2`

These tools are accessible either from the *Manage Layers Toolbar* and the *Layer ► Add Layer ►* menu. Connecting to SpatiaLite database is described at *SpatiaLite Layers*.

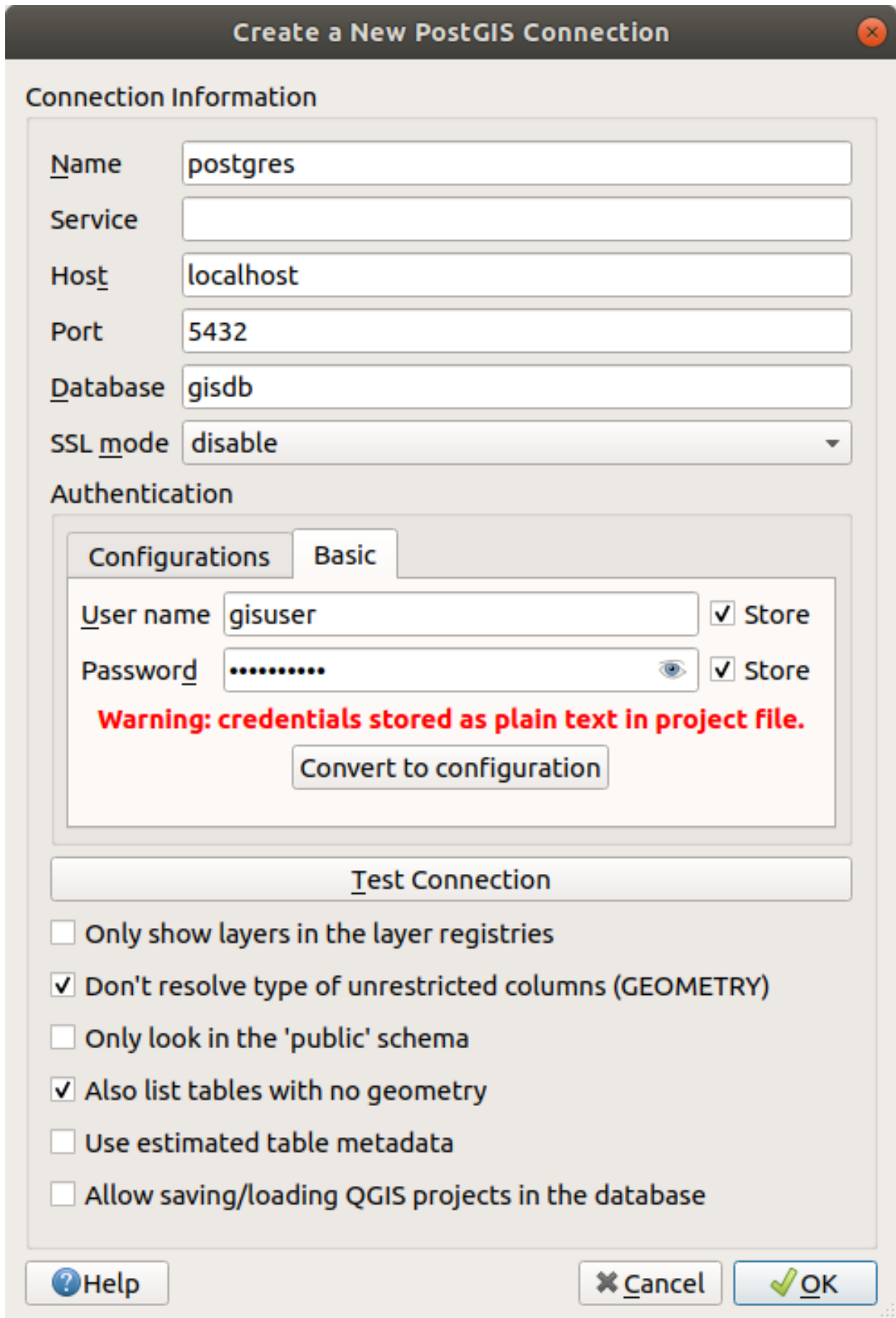
小技巧: Create connection to database from the QGIS Browser Panel

Selecting the corresponding database format in the Browser tree, right-clicking and choosing connect will provide you with the database connection dialog.

Most of the connection dialogs follow a common basis that will be described below using the PostgreSQL database tool as an example. For additional settings specific to other providers, you can find corresponding descriptions at:

- *Connecting to MSSQL Spatial;*
- *Connecting to Oracle Spatial;*
- *Connecting to DB2 Spatial.*

The first time you use a PostGIS data source, you must create a connection to a database that contains the data. Begin by clicking the appropriate button as exposed above, opening an *Add PostGIS Table(s)* dialog (see [图 13.12](#)). To access the connection manager, click on the *New* button to display the *Create a New PostGIS Connection* dialog.



Create a New PostGIS Connection

Connection Information

Name: postgres

Service:

Host: localhost

Port: 5432

Database: gisdb

SSL mode: disable

Authentication

Configurations Basic

User name: gisuser Store

Password: Store

Warning: credentials stored as plain text in project file.

Convert to configuration

Test Connection

Only show layers in the layer registries

Don't resolve type of unrestricted columns (GEOMETRY)

Only look in the 'public' schema

Also list tables with no geometry

Use estimated table metadata

Allow saving/loading QGIS projects in the database

Help Cancel OK

图 13.11: Create a New PostGIS Connection Dialog


The parameters required for a PostGIS connection are explained below. For the other database types, see their differences at *Particular Connection requirements*.

- *Name*: A name for this connection. It can be the same as *Database*.
- *Service*: Service parameter to be used alternatively to hostname/port (and potentially database). This can be defined in `pg_service.conf`. Check the *PostgreSQL Service connection file* section for more details.
- *Host*: Name of the database host. This must be a resolvable host name such as would be used to open a TCP/IP connection or ping the host. If the database is on the same computer as QGIS, simply enter *localhost* here.
- *Port*: Port number the PostgreSQL database server listens on. The default port for PostGIS is 5432.
- *Database*: Name of the database.
- *SSL mode*: SSL encryption setup The following options are available:
 - *Prefer* (the default): I don't care about encryption, but I wish to pay the overhead of encryption if the server supports it.
 - *Require*: I want my data to be encrypted, and I accept the overhead. I trust that the network will make sure I always connect to the server I want.
 - *Verify CA*: I want my data encrypted, and I accept the overhead. I want to be sure that I connect to a server that I trust.
 - *Verify Full*: I want my data encrypted, and I accept the overhead. I want to be sure that I connect to a server I trust, and that it's the one I specify.
 - *Allow*: I don't care about security, but I will pay the overhead of encryption if the server insists on it.
 - *Disable*: I don't care about security, and I don't want to pay the overhead of encryption.
- *Authentication*, basic.
 - *User name*: User name used to log in to the database.
 - *Password*: Password used with *Username* to connect to the database.

You can save any or both of the `User name` and `Password` parameters, in which case they will be used by default each time you need to connect to this database. If not saved, you'll be prompted to supply the credentials to connect to the database in next QGIS sessions. The connection parameters you entered are stored in a temporary internal cache and returned whenever a username/password for the same database is requested, until you end the current QGIS session.

警告: QGIS User Settings and Security

In the *Authentication* tab, saving **username** and **password** will keep unprotected credentials in the connection configuration. Those **credentials will be visible** if, for instance, you share the project file with someone. Therefore, it is advisable to save your credentials in an *Authentication configuration* instead (*Configurations* tab - See *认证系统* for more details) or in a service connection file (see *PostgreSQL Service connection file* for example).

- *Authentication*, configurations. Choose an authentication configuration. You can add configurations using the  button. Choices are:
 - Basic authentication
 - PKI PKCS#12 authentication
 - PKI paths authentication
 - PKI stored identity certificate

Optionally, depending on the type of database, you can activate the following checkboxes:

- *Only show layers in the layer registries*

- Don't resolve type of unrestricted columns (*GEOMETRY*)
- Only look in the 'public' schema
- Also list tables with no geometry
- Use estimated table metadata
- Allow saving/loading QGIS projects in the database - more details [here](#)

小技巧: Use estimated table metadata to speed up operations

When initializing layers, various queries may be needed to establish the characteristics of the geometries stored in the database table. When the *Use estimated table metadata* option is checked, these queries examine only a sample of the rows and use the table statistics, rather than the entire table. This can drastically speed up operations on large datasets, but may result in incorrect characterization of layers (e.g. the feature count of filtered layers will not be accurately determined) and may even cause strange behaviour if columns that are supposed to be unique actually are not.

Once all parameters and options are set, you can test the connection by clicking the *Test Connection* button or apply it by clicking the *OK* button. From *Add PostGIS Table(s)*, click now on *Connect*, and the dialog is filled with tables from the selected database (as shown in [图 13.12](#)).

Particular Connection requirements

Because of database type particularities, provided options are not the same. Database specific options are described below.

PostgreSQL Service connection file

The service connection file allows PostgreSQL connection parameters to be associated with a single service name. That service name can then be specified by a client and the associated settings will be used.

It's called `.pg_service.conf` under *nix systems (GNU/Linux, macOS etc.) and `pg_service.conf` on Windows.

The service file can look like this:

```
[water_service]
host=192.168.0.45
port=5433
dbname=gisdb
user=paul
password=paulspass

[wastewater_service]
host=dbserver.com
dbname=water
user=waterpass
```

注解: There are two services in the above example: `water_service` and `wastewater_service`. You can use these to connect from QGIS, pgAdmin, etc. by specifying only the name of the service you want to connect to (without the enclosing brackets). If you want to use the service with `psql` you need to do something like `export PGSERVICE=water_service` before doing your `psql` commands.

You can find all the PostgreSQL parameters [here](#)

注解: If you don't want to save the passwords in the service file you can use the `.pg_pass` option.

On *nix operating systems (GNU/Linux, macOS etc.) you can save the `.pg_service.conf` file in the user's home directory and PostgreSQL clients will automatically be aware of it. For example, if the logged user is `web`, `.pg_service.conf` should be saved in the `/home/web/` directory in order to directly work (without specifying any other environment variables).

You can specify the location of the service file by creating a `PGSERVICEFILE` environment variable (e.g. run the `export PGSERVICEFILE=/home/web/.pg_service.conf` command under your *nix OS to temporarily set the `PGSERVICEFILE` variable)

You can also make the service file available system-wide (all users) either by placing the `.pg_service.conf` file in `pg_config --sysconfdir` or by adding the `PGSYSCONFDIR` environment variable to specify the directory containing the service file. If service definitions with the same name exist in the user and the system file, the user file takes precedence.

警告: There are some caveats under Windows:

- The service file should be saved as `pg_service.conf` and not as `.pg_service.conf`.
- The service file should be saved in Unix format in order to work. One way to do it is to open it with `Notepad++` and `Edit ▶ EOL Conversion ▶ UNIX Format ▶ File save`.
- You can add environmental variables in various ways; a tested one, known to work reliably, is `Control Panel ▶ System and Security ▶ System ▶ Advanced system settings ▶ Environment Variables` adding `PGSERVICEFILE` with the path - e.g. `C:\Users\John\pg_service.conf`
- After adding an environment variable you may also need to restart the computer.

Connecting to Oracle Spatial

The spatial features in Oracle Spatial aid users in managing geographic and location data in a native type within an Oracle database. In addition to some of the options in *Creating a stored Connection*, the connection dialog proposes:

- **Database:** SID or `SERVICE_NAME` of the Oracle instance;
- **Port:** Port number the Oracle database server listens on. The default port is 1521;
- **Options:** Oracle connection specific options (e.g. `OCI_ATTR_PREFETCH_ROWS`, `OCI_ATTR_PREFETCH_MEMORY`). The format of the options string is a semicolon separated list of option names or `option=value` pairs;
- **Workspace:** Workspace to switch to;
- **Schema:** Schema in which the data are stored

Optionally, you can activate the following checkboxes:

- *Only look in metadata table:* restricts the displayed tables to those that are in the `all_sdo_geom_metadata` view. This can speed up the initial display of spatial tables.
- *Only look for user's tables:* when searching for spatial tables, restricts the search to tables that are owned by the user.
- *Also list tables with no geometry:* indicates that tables without geometry should also be listed by default.
- *Use estimated table statistics for the layer metadata:* when the layer is set up, various metadata are required for the Oracle table. This includes information such as the table row count, geometry type and spatial extents of the data in the geometry column. If the table contains a large number of rows, determining this metadata can be time-consuming. By activating this option, the following fast table metadata operations are done: Row count is determined from `all_tables.num_rows`. Table extents are always determined with the

SDO_TUNE.EXTENTS_OF function, even if a layer filter is applied. Table geometry is determined from the first 100 non-null geometry rows in the table.

- *Only existing geometry types*: only lists the existing geometry types and don't offer to add others.
- *Include additional geometry attributes*.

小技巧: Oracle Spatial Layers

Normally, an Oracle Spatial layer is defined by an entry in the **USER_SDO_METADATA** table.

To ensure that selection tools work correctly, it is recommended that your tables have a **primary key**.

Connecting to DB2 Spatial

In addition to some of the options described in *Creating a stored Connection*, the connection to a DB2 database (see *DB2 Spatial Layers* for more information) can be specified using either a *Service/DSN* name defined to ODBC or *Driver, Host* and *Port*.

An ODBC **Service/DSN** connection requires the service name defined to ODBC.

A driver/host/port connection requires:

- **Driver**: Name of the DB2 driver. Typically this would be IBM DB2 ODBC DRIVER.
- **DB2 Host**: Name of the database host. This must be a resolvable host name such as would be used to open a TCP/IP connection or ping the host. If the database is on the same computer as QGIS, simply enter *localhost* here.
- **DB2 Port**: Port number the DB2 database server listens on. The default DB2 LUW port is 50000. The default DB2 z/OS port is 446.

小技巧: DB2 Spatial Layers

A DB2 Spatial layer is defined by a row in the **DB2GSE.ST_GEOMETRY_COLUMNS** view.

注解: In order to work effectively with DB2 spatial tables in QGIS, it is important that tables have an INTEGER or BIGINT column defined as PRIMARY KEY and if new features are going to be added, this column should also have the GENERATED characteristic.

It is also helpful for the spatial column to be registered with a specific spatial reference identifier (most often 4326 for WGS84 coordinates). A spatial column can be registered by calling the `ST_Register_Spatial_Column` stored procedure.

Connecting to MSSQL Spatial

In addition to some of the options in *Creating a stored Connection*, creating a new MSSQL connection dialog proposes you to fill a **Provider/DSN** name. You can also display available databases.

Loading a Database Layer

Once you have one or more connections defined to a database (see section *Creating a stored Connection*), you can load layers from it. Of course, this requires that data are available. See section *Importing Data into PostgreSQL* for a discussion on importing data into a PostGIS database.

To load a layer from a database, you can perform the following steps:

1. Open the "Add <database> table(s)" dialog (see *Creating a stored Connection*).
2. Choose the connection from the drop-down list and click *Connect*.
3. Select or unselect *Also list tables with no geometry*.
4. Optionally, use some *Search Options* to reduce the list of tables to those matching your search. You can also set this option before you hit the *Connect* button, speeding up the database fetching.
5. Find the layer(s) you wish to add in the list of available layers.
6. Select it by clicking on it. You can select multiple layers by holding down the *Shift* or *Ctrl* key while clicking.
7. If applicable, use the *Set Filter* button (or double-click the layer) to start the *Query Builder* dialog (see section *Query Builder*) and define which features to load from the selected layer. The filter expression appears in the *sql* column. This restriction can be removed or edited in the *Layer Properties* ► *General* ► *Provider Feature Filter* frame.
8. The checkbox in the *Select at id* column that is activated by default gets the feature ids without the attributes and generally speeds up the data loading.
9. Click on the *Add* button to add the layer to the map.

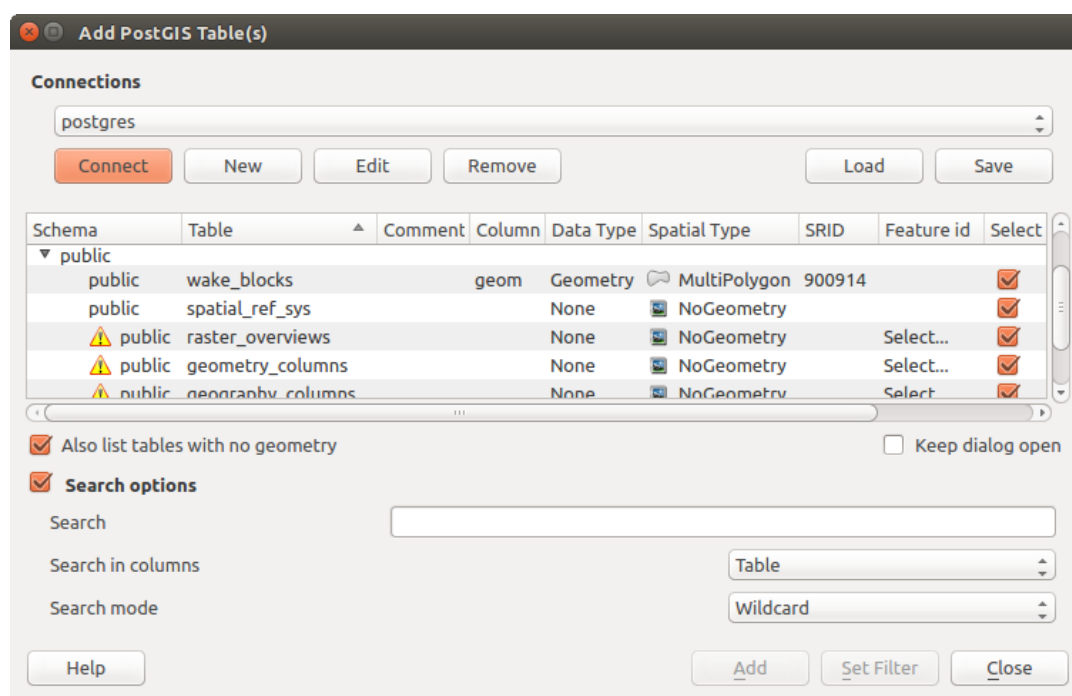


图 13.12: Add PostGIS Table(s) Dialog

小技巧: Use the Browser Panel to speed up loading of database table(s)

Adding DB tables from the *Data Source Manager* may sometimes be time consuming as QGIS fetches statistics and properties (e.g. geometry type and field, CRS, number of features) for each table beforehand. To avoid this, once *the*

connection is set, it is better to use the *Browser Panel* or the *DB Manager* to drag and drop the database tables into the map canvas.

13.1.4 QGIS Custom formats

QGIS proposes two custom formats:

- Temporary Scratch Layer: a memory layer that is bound to the project (see *Creating a new Temporary Scratch Layer* for more information)
- Virtual Layers: a layer resulting from a query on other layer(s) (see *Creating virtual layers* for more information)

13.1.5 QLR - QGIS Layer Definition File

Layer definitions can be saved as a *Layer Definition File* (QLR - .qlr) using *Export ► Save As Layer Definition File...* in the layer context menu.


The QLR format makes it possible to share "complete" QGIS layers with other QGIS users. QLR files contain links to the data sources and all the QGIS style information necessary to style the layer.

QLR files are shown in the Browser Panel and can be used to add layers (with their saved styles) to the Layers Panel. You can also drag and drop QLR files from the system file manager into the map canvas.

13.1.6 Connecting to web services

With QGIS you can get access to different types of OGC web services (WM(T)S, WFS(-T), WCS, CSW, ...). Thanks to QGIS Server, you can also publish such services. QGIS-Server-manual contains descriptions of these capabilities.

Using Vector Tiles services

Vector Tiles services can be found in the *Vector Tiles* top level entry in the *Browser*. You can add a service by opening the context menu with a right-click and choosing *New Generic Connection ...*. You set up a service by adding a *Name* and a *URL*. The Vector Tiles Service must provide tiles in *.pbf* format. The dialog provides two menus to define the *Min. Zoom Level* and the *Max. Zoom Level*. Vector Tiles have a pyramid structure. By using these options you have the opportunity to individually generate layers from the tile pyramid. These layers will then be used to render the Vector Tile in QGIS. For Mercator projection (used by OpenStreetMap Vector Tiles) Zoom Level 0 represents the whole world at a scale of 1:500.000.000. Zoom Level 14 represents the scale 1:35.000.  13.13 shows the dialog with the MapTiler planet Vector Tiles service configuration.

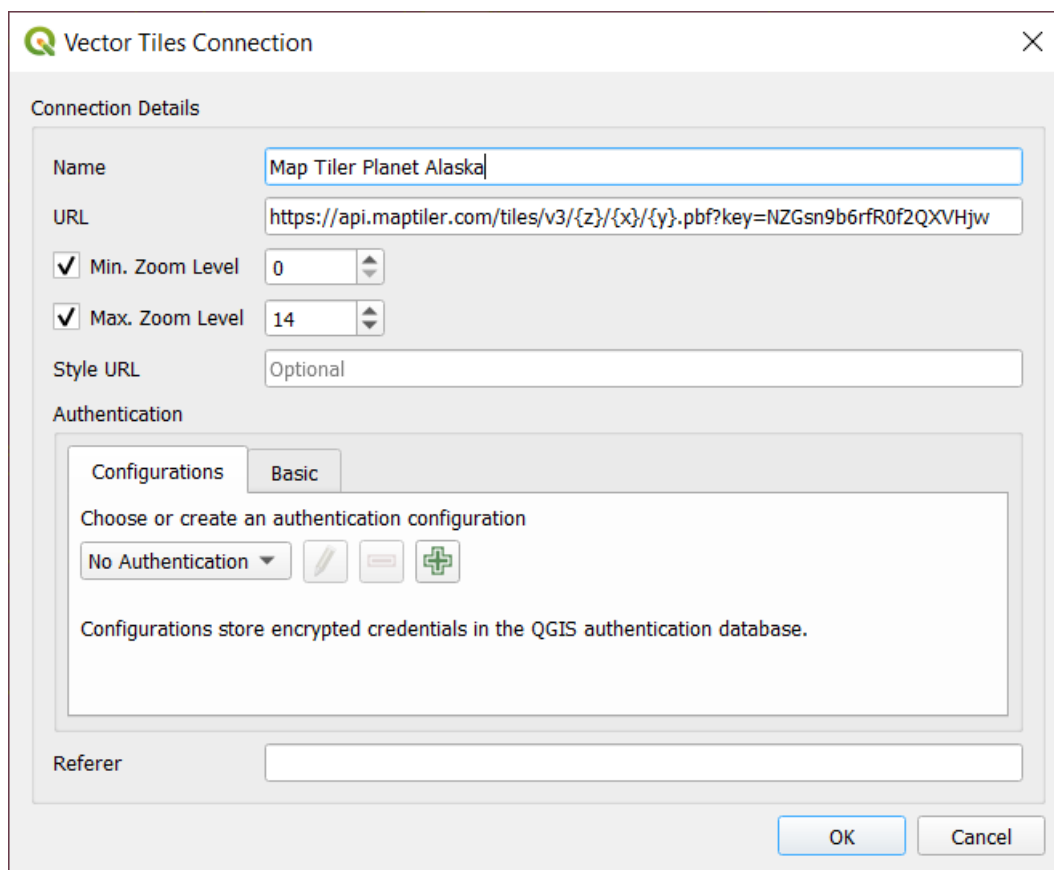


图 13.13: Vector Tiles - Maptiler Planet configuration

By using *New ArcGIS Vector Tile Service Connection ...* you can connect to ArcGIS Vector Tile Services.

Using XYZ Tile services

XYZ Tile services can be found in the *XYZ Tiles* top level entry in the *Browser*. By default, the OpenStreetMap XYZ Tile service is configured. You can add other services that use the XYZ Tile protocol by choosing *New Connection* in the XYZ Tiles context menu (right-click to open). 图 13.14 shows the dialog with the OpenStreetMap XYZ Tile service configuration.

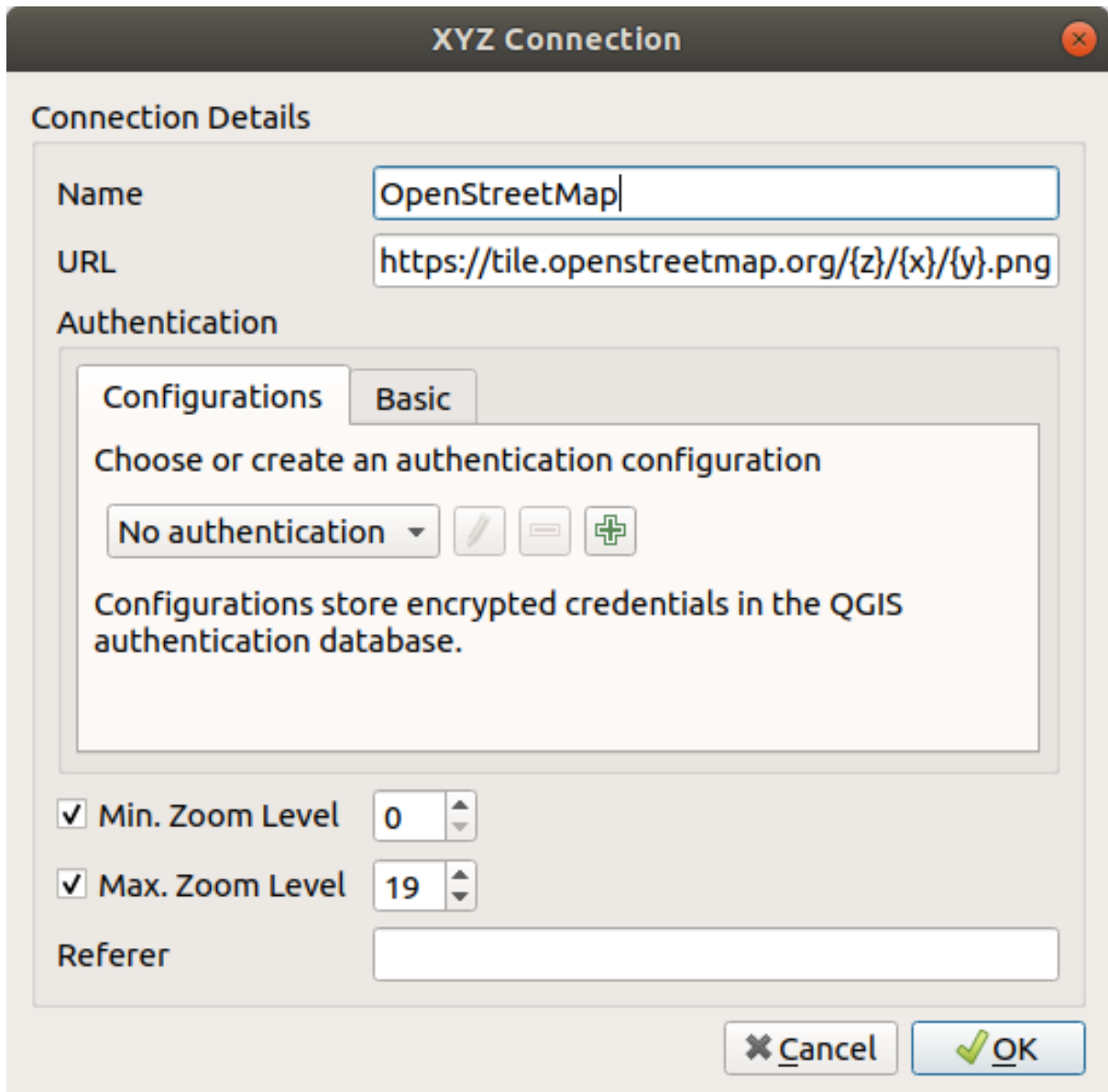


图 13.14: XYZ Tiles - OpenStreetMap 配置

Configurations can be saved (*Save Connections*) to XML and loaded (*Load Connections*) through the context menu. Authentication configuration is supported. The XML file for OpenStreetMap looks like this:

```
<!DOCTYPE connections>
<qgsXYZTilesConnections version="1.0">
  <xyztiles url="https://tile.openstreetmap.org/{z}/{x}/{y}.png"
    zmin="0" zmax="19" tilePixelRatio="0" password="" name="OpenStreetMap"
    username="" authcfg="" referer="" />
</qgsXYZTilesConnections>
```

Once a connection to a XYZ tile service is set, right-click over the entry to:

- *Edit...* the XYZ connection settings
- *Delete* the connection
- *Export layer...* ► *To File, saving it as a raster*
- *Add layer to project*: a double-click also adds the layer

- View the *Layer Properties...* and get access to metadata and a preview of the data provided by the service. More settings are available when the layer has been loaded into the project.

Examples of XYZ Tile services:

- OpenStreetMap Monochrome: *URL:* `http://tiles.wmflabs.org/bw-mapnik/{z}/{x}/{y}.png`, *Min. Zoom Level:* 0, *Max. Zoom Level:* 19.
- Google Maps: *URL:* `https://mt1.google.com/vt/lyrs=m&x={x}&y={y}&z={z}`, *Min. Zoom Level:* 0, *Max. Zoom Level:* 19.
- Open Weather Map Temperature: *URL:* `http://tile.openweathermap.org/map/temp_new/{z}/{x}/{y}.png?appid={api_key}` *Min. Zoom Level:* 0, *Max. Zoom Level:* 19.

13.2 Creating Layers

Layers can be created in many ways, including:

- empty layers from scratch
- layers from existing layers
- layers from the clipboard
- layers as a result of an SQL-like query based on one or many layers (*virtual layers*)

QGIS also provides tools to import/export from/to different formats.

13.2.1 Creating new vector layers

QGIS allows you to create new layers in different formats. It provides tools for creating GeoPackage, Shapefile, SpatiaLite, GPX format and Temporary Scratch layers (aka memory layers). Creation of a *new GRASS layer* is supported within the GRASS plugin.

Creating a new GeoPackage layer

To create a new GeoPackage layer, press the  *New GeoPackage Layer...* button in the *Layer ► Create Layer ►* menu or from the *Data Source Manager* toolbar. The *New GeoPackage Layer* dialog will be displayed as shown in  13.15.

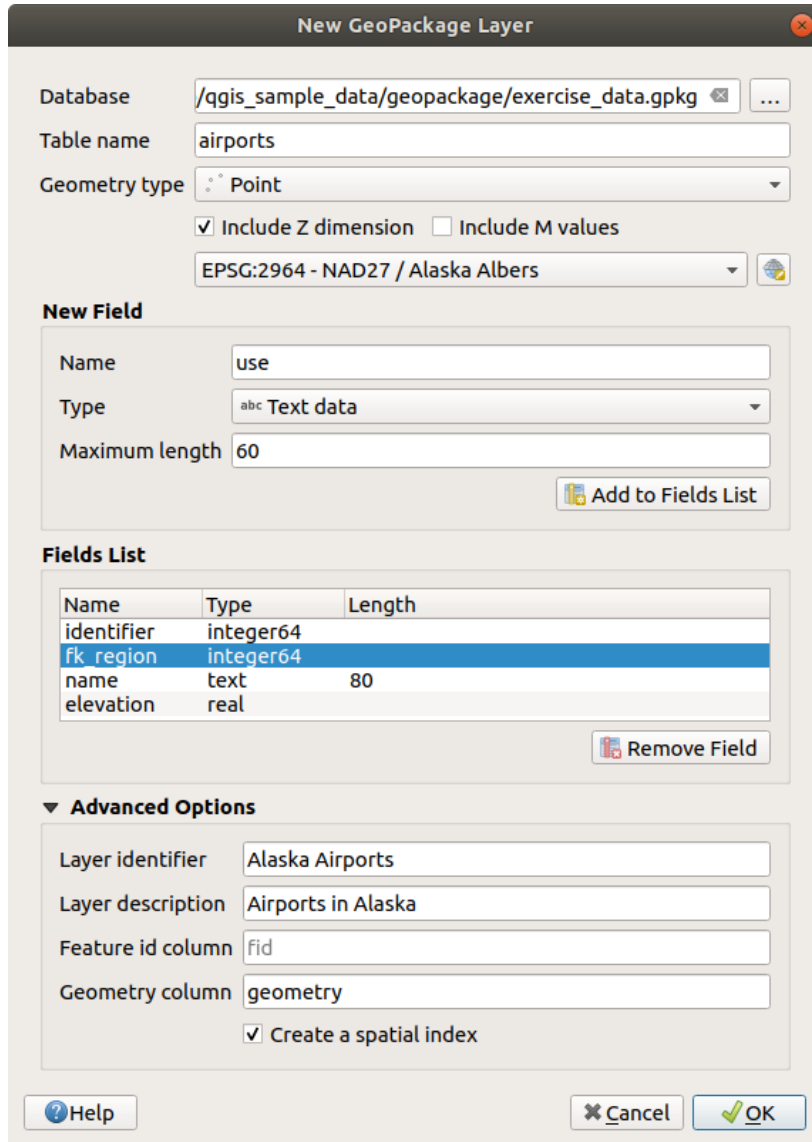




图 13.15: Creating a New GeoPackage layer dialog

1. The first step is to indicate the database file location. This can be done by pressing the ... button to the right of the *Database* field and select an existing GeoPackage file or create a new one. QGIS will automatically add the right extension to the name you provide.
2. Give the new layer / table a name (*Table name*)
3. Define the *Geometry type*. If not a geometryless layer, you can specify whether it should *Include Z dimension* and/or *Include M values*.
4. Specify the coordinate reference system using the  button

To add fields to the layer you are creating:

1. Enter the *Name* of the field


2. Select the data *Type*. Supported types are *Text data*, *Whole number* (both integer and integer64), *Decimal number*, *Date* and *Date and time*, *Binary (BLOB)* and *Boolean*.
3. Depending on the selected data format, enter the *Maximum length* of values.
4. Click on the  *Add to Fields List* button
5. Reproduce the steps above for each field you need to add
6. Once you are happy with the attributes, click *OK*. QGIS will add the new layer to the legend, and you can edit it as described in section *Digitizing an existing layer*.

By default, when creating a GeoPackage layer, QGIS generates a *Feature id column* called `fid` which acts as the primary key of the layer. The name can be changed. The geometry field, if available, is named `geometry`, and you can choose to *Create a spatial index* on it. These options can be found under the *Advanced Options* together with the *Layer identifier* (short human readable name of the layer) and the *Layer description*.

Further management of GeoPackage layers can be done with the *DB Manager*.

Creating a new Shapefile layer

To create a new ESRI Shapefile format layer, press the  *New Shapefile Layer...* button in the *Layer ► Create Layer ►* menu or from the *Data Source Manager* toolbar. The *New Shapefile Layer* dialog will be displayed as shown in [图 13.16](#).

1. Provide a path and file name using the ... button next to *File name*. QGIS will automatically add the right extension to the name you provide.
2. Next, indicate the *File encoding* of the data
3. Choose the *Geometry type* of the layer: *No Geometry* (resulting in a `.DBF` format file), *point*, *multipoint*, *line* or *polygon*
4. Specify whether the geometry should have additional dimensions: *None*, *Z (+ M values)* or *M values*
5. Specify the coordinate reference system using the  button

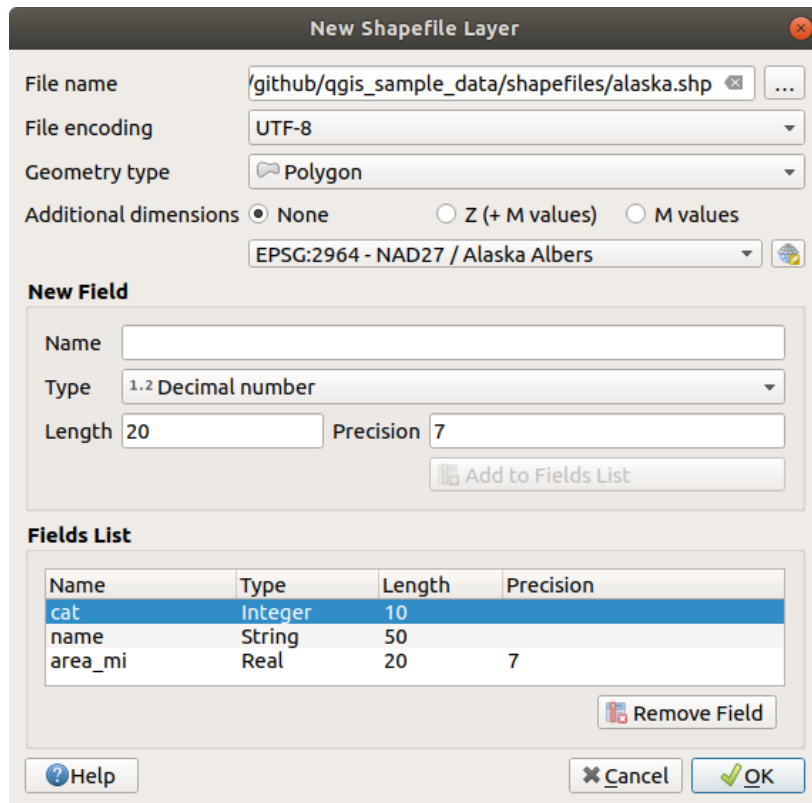



图 13.16: Creating a new Shapefile layer dialog

To add fields to the layer you are creating:

1. Enter the *Name* of the field
2. Select the data *Type*. Only *Decimal number*, *Whole number*, *Text data* and *Date* attributes are supported.
3. Depending on the selected data format, enter the *Length* and *Precision*.
4. Click on the  *Add to Fields List* button
5. Reproduce the steps above for each field you need to add
6. Once you are happy with the attributes, click *OK*. QGIS will add the new layer to the legend, and you can edit it as described in section *Digitizing an existing layer*.

By default, a first integer *id* column is added but can be removed.

Creating a new SpatiaLite layer

To create a new SpatiaLite layer, press the  *New SpatiaLite Layer...* button in the *Layer ► Create Layer ►* menu or from the *Data Source Manager* toolbar. The *New SpatiaLite Layer* dialog will be displayed as shown in .

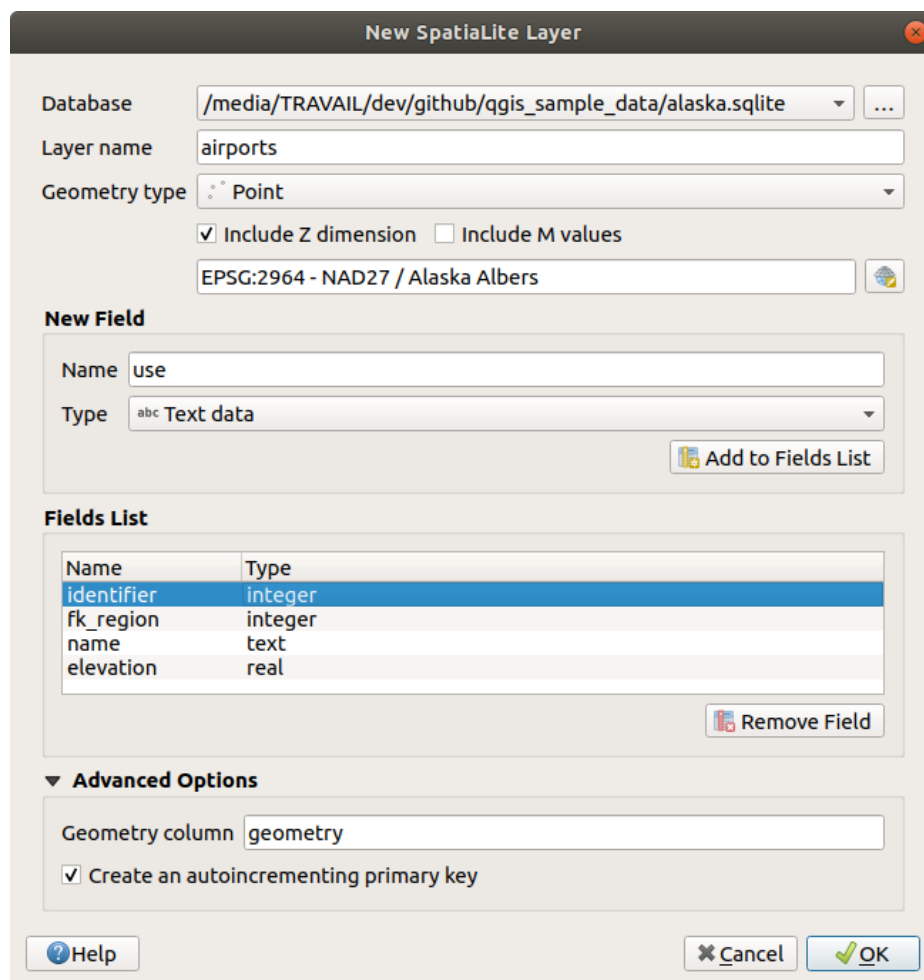




图 13.17: Creating a New SpatiaLite layer dialog

1. The first step is to indicate the database file location. This can be done by pressing the ... button to the right of the *Database* field and select an existing SpatiaLite file or create a new one. QGIS will automatically add the right extension to the name you provide.
2. Provide a name (*Layer name*) for the new layer
3. Define the *Geometry type*. If not a geometryless layer, you can specify whether it should *Include Z dimension* and/or *Include M values*.
4. Specify the coordinate reference system using the  button.

To add fields to the layer you are creating:


1. Enter the *Name* of the field
2. Select the data *Type*. Supported types are *Text data*, *Whole number* and *Decimal number*.
3. Click on the  *Add to Fields List* button
4. Reproduce the steps above for each field you need to add

- Once you are happy with the attributes, click *OK*. QGIS will add the new layer to the legend, and you can edit it as described in section *Digitizing an existing layer*.

If desired, you can select *Create an autoincrementing primary key* under the *Advanced Options* section. You can also rename the *Geometry column* (*geometry* by default).

Further management of SpatiaLite layers can be done with *DB Manager*.


Creating a new GPX layer



To create a new GPX file, you first need to load the GPS plugin. *Plugins* ►  *Plugin Manager...* opens the Plugin Manager Dialog. Activate the *GPS Tools* checkbox.

When this plugin is loaded, choose *Create Layer* ►  *Create new GPX Layer...* from the *Layer* menu. In the dialog, choose where to save the new file and press *Save*. Three new layers are added to the *Layers Panel*: `waypoints`, `routes` and `tracks`.

Creating a new Temporary Scratch Layer

Temporary Scratch Layers are in-memory layers, meaning that they are not saved on disk and will be discarded when QGIS is closed. They can be handy for storing features you temporarily need or as intermediate layers during geoprocessing operations.

To create a new Temporary Scratch layer, choose the  *New Temporary Scratch Layer...* entry in the *Layer* ► *Create Layer* ► menu or in the *Data Source Manager* toolbar. The *New Temporary Scratch Layer* dialog will be displayed as shown in [图 13.18](#). Then:

- Provide the *Layer name*
- Select the *Geometry type*. Here you can create a:
 - No `geometry` type layer, served as simple table,
 - `Point` or `MultiPoint` layer,
 - `LineString/CompoundCurve` or `MultiLineString/MultiCurve` layer,
 - `Polygon/CurvePolygon` or `MultiPolygon/MultiSurface` layer.
- For geometric types, specify the dimensions of the dataset: check whether it should *Include Z dimension* and/or *Include M values*
- Specify the coordinate reference system using the  button.
- Add fields to the layer. Note that unlike many formats, temporary layers can be created without any fields. This step is thus optional.
 - Enter the *Name* of the field
 - Select the data *Type*: *Text*, *Whole number*, *Decimal number*, *Boolean*, *Date*, *Time*, *Date & Time* and *Binary (BLOB)* are supported.
 - Depending on the selected data format, enter the *Length* and *Precision*
 - Click on the  *Add to Fields List* button
 - Repeat the steps above for each field you need to add
- Once you are happy with the settings, click *OK*. QGIS will add the new layer to the *Layers panel*, and you can edit it as described in section *Digitizing an existing layer*.

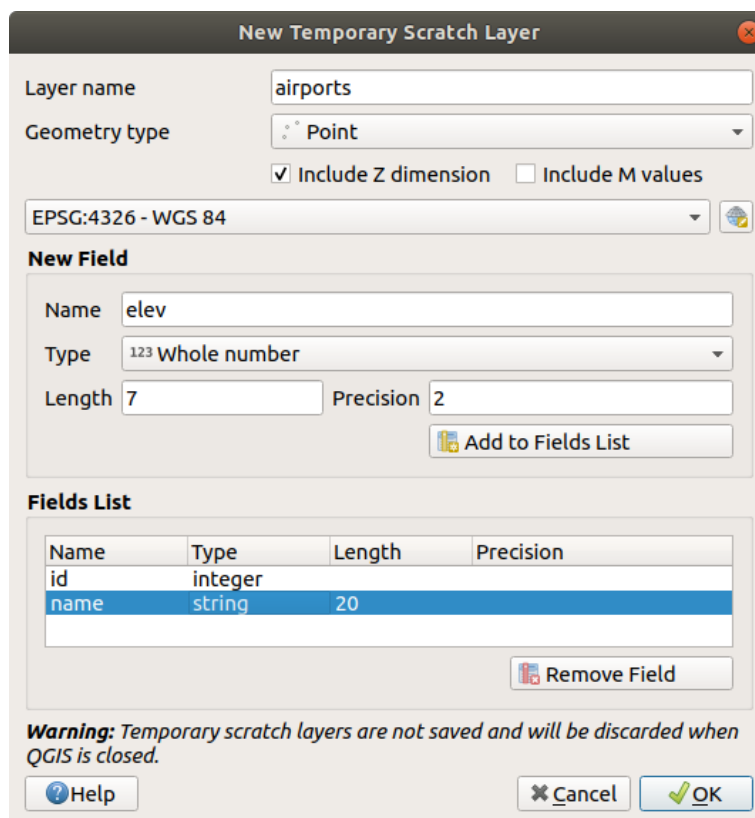



图 13.18: Creating a new Temporary Scratch layer dialog

You can also create prepopulated temporary scratch layers using e.g. the clipboard (see *Creating new layers from the clipboard*) or as a result of a *Processing algorithm*.

小技巧: Permanently store a memory layer on disk

To avoid data loss when closing a project with temporary scratch layers, you can save these layers to any vector format supported by QGIS:

- clicking the  indicator icon next to the layer;
- selecting the *Make permanent* entry in the layer contextual menu;
- using the *Export* ► entry from the contextual menu or the *Layer* ► *Save As...* menu.

Each of these commands opens the *Save Vector Layer as* dialog described in the *Creating new layers from an existing layer* section and the saved file replaces the temporary one in the *Layers* panel.

13.2.2 Creating new layers from an existing layer

Both raster and vector layers can be saved in a different format and/or reprojected to a different coordinate reference system (CRS) using the *Layer* ► *Save As...* menu or right-clicking on the layer in the *Layers panel* and selecting:

- *Export* ► *Save As...* for raster layers
- *Export* ► *Save Features As...* or *Export* ► *Save Selected Features As...* for vector layers.
- Drag and drop the layer from the layer tree to the PostGIS entry in the *Browser Panel*. Note that you must have a PostGIS connection in the *Browser Panel*.

Common parameters

The *Save Layer as...* dialog shows several parameters to change the behavior when saving the layer. Among the common parameters for raster and vector are:

- *File name*: the location of the file on the disk. It can refer to the output layer or to a container that stores the layer (for example database-like formats such as GeoPackage, SpatiaLite or Open Document Spreadsheets).
- *CRS*: can be changed to reproject the data
- *Extent* (possible values are **layer**, **Map view** or **user-defined** extent)
- *Add saved file to map*: to add the new layer to the canvas

However, some parameters are specific to raster and vector formats:

Raster specific parameters

Depending on the format of export, some of these options may not be available:

- *Output mode* (it can be **raw data** or **rendered image**)
- *Format*: exports to any raster format GDAL can write to, such as GeoTiff, GeoPackage, MBTiles, Geospatial PDF, SAGA GIS Binary Grid, Intergraph Raster, ESRI .hdr Labelled...
- *Resolution*
- *Create Options*: use advanced options (file compression, block sizes, colorimetry...) when generating files, either from the *predefined create profiles* related to the output format or by setting each parameter.
- *Pyramids* creation
- *VRT Tiles* in case you opted to *Create VRT*
- *No data values*

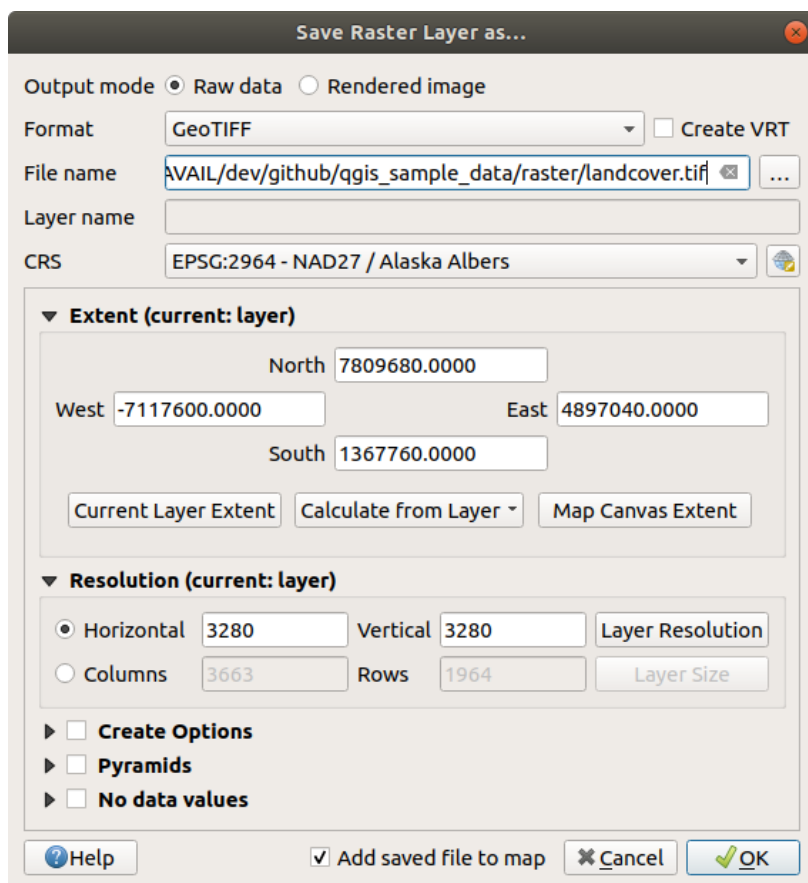


图 13.19: Saving as a new raster layer


Vector specific parameters

Depending on the format of export, some of these options may be available:

- *Format*: exports to any vector format GDAL can write to, such as GeoPackage, GML, ESRI Shapefile, AutoCAD DXF, ESRI FileGDB, Mapinfo TAB or MIF, Spatialite, CSV, KML, ODS, ...
- *Layer name*: available when the *File name* refers to a container-like format, this entry represents the output layer.
- *Encoding*
- *Save only selected features*
- *Select fields to export and their export options*. In case you set your fields behavior with some *Edit widgets*, e.g. value map, you can keep the displayed values in the layer by checking *Replace all selected raw fields values by displayed values*.
- *Symbology export*: can be used mainly for DXF export and for all file formats who manage OGR feature styles (see note below) as DXF, KML, tab file formats:
 - **No symbology**: default style of the application that reads the data
 - **Feature symbology**: save style with OGR Feature Styles (see note below)
 - **Symbol Layer symbology**: save with OGR Feature Styles (see note below) but export the same geometry multiple times if there are multiple symbology symbol layers used
 - A **Scale** value can be applied to the latest options

注解: *OGR Feature Styles* are a way to store style directly in the data as a hidden attribute. Only some formats can handle this kind of information. KML, DXF and TAB file formats are such formats. For advanced details, you can read the [OGR Feature Styles specification](#) document.

- *Geometry*: you can configure the geometry capabilities of the output layer
 - *geometry type*: keeps the original geometry of the features when set to **Automatic**, otherwise removes or overrides it with any type. You can add an empty geometry column to an attribute table and remove the geometry column of a spatial layer.
 - *Force multi-type*: forces creation of multi-geometry features in the layer.
 - *Include z-dimension* to geometries.
-

小技巧: Overriding layer geometry type makes it possible to do things like save a geometryless table (e.g. `.csv` file) into a shapefile **WITH** any type of geometry (point, line, polygon), so that geometries can then be manually added to rows with the  Add Part tool.

- *Datasource Options*, *Layer Options* or *Custom Options* which allow you to configure advanced parameters depending on the output format. Some are described in [Exploring Data Formats and Fields](#) but for full details, see the [GDAL driver documentation](#). Each file format has its own custom parameters, e.g. for the `GeoJSON` format have a look at the [GDAL GeoJSON documentation](#).

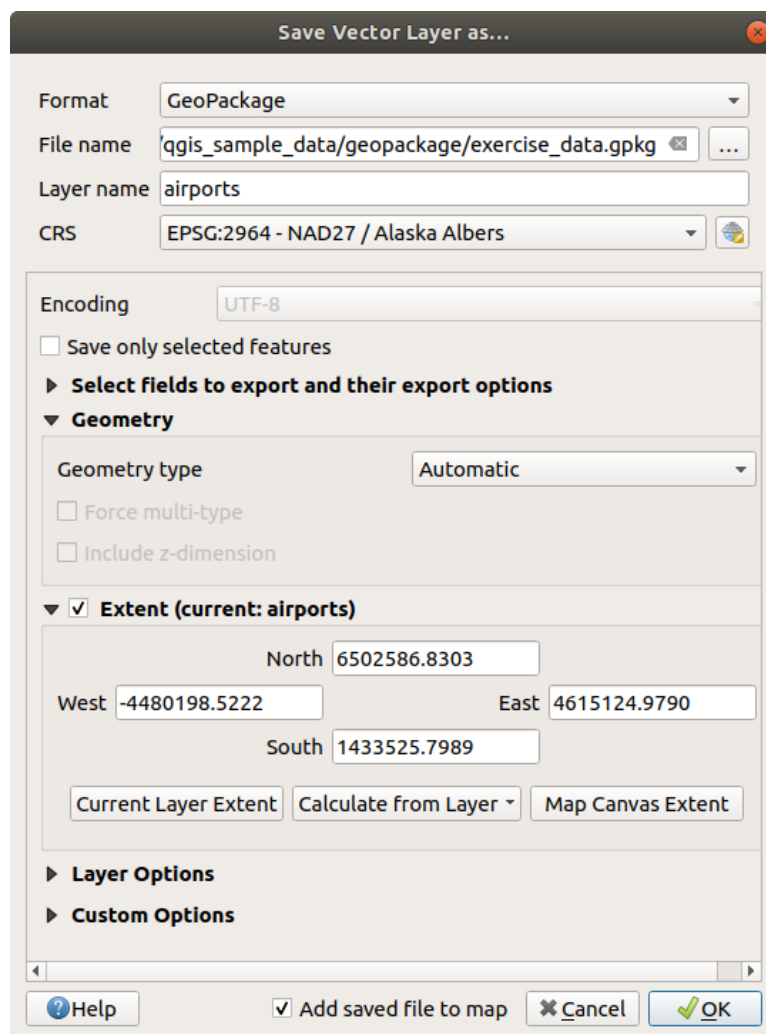


图 13.20: Saving as a new vector layer

When saving a vector layer into an existing file, depending on the capabilities of the output format (Geopackage, SpatiaLite, FileGDB...), the user can decide whether to:

- overwrite the whole file
- overwrite only the target layer (the layer name is configurable)
- append features to the existing target layer
- append features, add new fields if there are any.

For formats like ESRI Shapefile, MapInfo .tab, feature append is also available.

13.2.3 Creating new DXF files

Besides the *Save As...* dialog which provides options to export a single layer to another format, including *.DXF, QGIS provides another tool to export multiple layers as a single DXF layer. It's accessible in the *Project ► Import/Export ► Export Project to DXF...* menu.

In the *DXF Export* dialog:

1. Provide the destination file.
2. Choose the symbology mode and scale (see the *OGR Feature Styles* note), if applicable.
3. Select the data *Encoding*.

4. Select the *CRS* to apply: the selected layers will be reprojected to the given *CRS*.
5. Select the layers to include in the DXF files either by checking them in the table widget or automatically picking them from an existing *map theme*. The *Select All* and *Deselect All* buttons can help to quickly set the data to export.

For each layer, you can choose whether to export all the features in a single DXF layer or rely on a field whose values are used to split the features into layers in the DXF output.

Optionally, you can also choose to:

- Use the layer title as name if set instead of the layer name itself;
- Export features intersecting the current map extent;
- Force 2d output (eg. to support polyline width);
- Export label as *MTEXT* elements or *TEXT* elements.

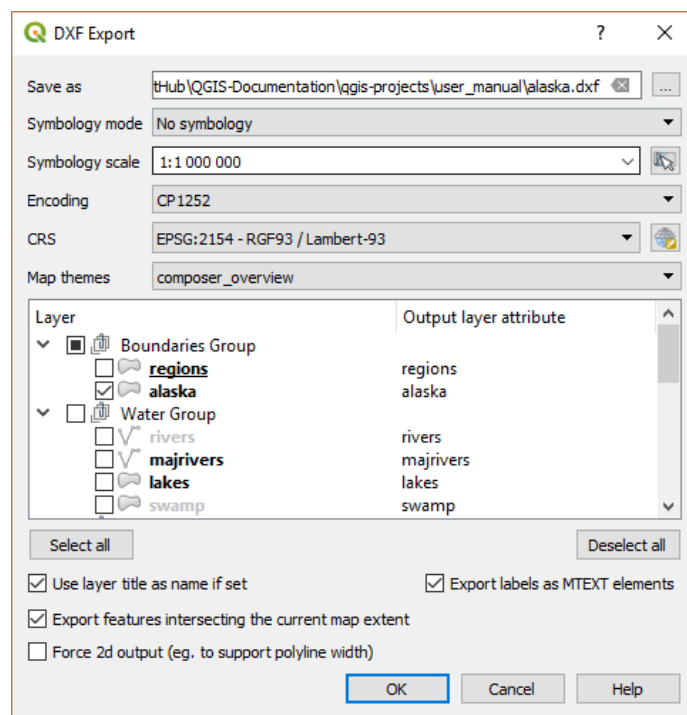


图 13.21: Exporting a project to DXF dialog

13.2.4 Creating new layers from the clipboard

Features that are on the clipboard can be pasted into a new layer. To do this, Select some features, copy them to the clipboard, and then paste them into a new layer using *Edit* ► *Paste Features as* ► and choosing:

- *New Vector Layer...*: the *Save vector layer as...* dialog appears (see *Creating new layers from an existing layer* for parameters)
- or *Temporary Scratch Layer...*: you need to provide a name for the layer



A new layer, filled with selected features and their attributes is created (and added to map canvas).

注解: Creating layers from the clipboard is possible with features selected and copied within QGIS as well as features from another application, as long as their geometries are defined using well-known text (WKT).

13.2.5 Creating virtual layers

A virtual layer is a special kind of vector layer. It allows you to define a layer as the result of an SQL query involving any number of other vector layers that QGIS is able to open. Virtual layers do not carry data by themselves and can be seen as views.

To create a virtual layer, open the virtual layer creation dialog by:

- choosing the  *Add/Edit Virtual Layer* entry in the *Layer ► Add Layer ►* menu;
- enabling the  *Add Virtual Layer* tab in the *Data Source Manager* dialog;
- or using the *DB Manager* dialog tree.

The dialog allows you to specify a *Layer name* and an SQL *Query*. The query can use the name (or id) of loaded vector layers as tables, as well as their field names as columns.

For example, if you have a layer called `airports`, you can create a new virtual layer called `public_airports` with an SQL query like:

```
SELECT *
FROM airports
WHERE USE = "Civilian/Public"
```

The SQL query will be executed, regardless of the underlying provider of the `airports` layer, even if this provider does not directly support SQL queries.

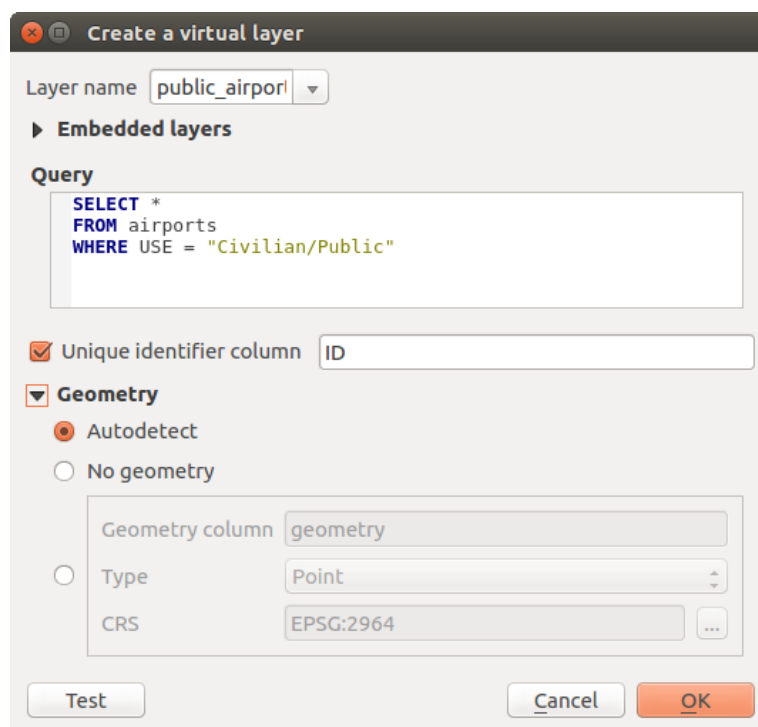


图 13.22: Create virtual layers dialog

Joins and complex queries can also be created, for example, to join `airports` and country information:

```
SELECT airports.*, country.population
FROM airports
JOIN country
ON airports.country = country.name
```

注解: It's also possible to create virtual layers using the SQL window of *DB Manager Plugin*.

Embedding layers for use in queries

Besides the vector layers available in the map canvas, the user can add layers to the *Embedded layers* list, which can be used in queries without the need to have them showing in the map canvas or Layers panel.

To embed a layer, click *Add* and provide the *Local name*, *Provider*, *Encoding* and the path to the *Source*.

The *Import* button allows adding layers in the map canvas into the Embedded layers list. Those layers can then be removed from the Layers panel without breaking existent queries.

Supported query language

The underlying engine uses SQLite and SpatiaLite to operate.

It means you can use all of the SQL your local installation of SQLite understands.

Functions from SQLite and spatial functions from SpatiaLite can also be used in a virtual layer query. For instance, creating a point layer out of an attribute-only layer can be done with a query similar to:

```
SELECT id, MakePoint(x, y, 4326) as geometry
FROM coordinates
```

Functions of QGIS expressions can also be used in a virtual layer query.

To refer the geometry column of a layer, use the name *geometry*.

Contrary to a pure SQL query, all the fields of a virtual layer query must be named. Don't forget to use the *as* keyword to name your columns if they are the result of a computation or a function call.

Performance issues

With default parameters, the virtual layer engine will try its best to detect the type of the different columns of the query, including the type of the geometry column if one is present.

This is done by introspecting the query when possible or by fetching the first row of the query (*LIMIT 1*) as a last resort. Fetching the first row of the result just to create the layer may be undesirable for performance reasons.

The creation dialog parameters:

- *Unique identifier column*: specifies a field of the query that represents unique integer values that QGIS can use as row identifiers. By default, an autoincrementing integer value is used. Defining a unique identifier column speeds up the selection of rows by id.
- *No geometry*: forces the virtual layer to ignore any geometry field. The resulting layer is an attribute-only layer.
- *Geometry Column*: specifies the name of the geometry column.
- *Geometry Type*: specifies the type of the geometry.
- *Geometry CRS*: specifies the coordinate reference system of the virtual layer.

Special comments

The virtual layer engine tries to determine the type of each column of the query. If it fails, the first row of the query is fetched to determine column types.

The type of a particular column can be specified directly in the query by using some special comments.

The syntax is the following: `/*:type*/`. It has to be placed just after the name of a column. `type` can be either `int` for integers, `real` for floating point numbers or `text`.

For instance:

```
SELECT id+1 as nid /*:int*/
FROM table
```

The type and coordinate reference system of the geometry column can also be set thanks to special comments with the following syntax `/*:gtype:srid*/` where `gtype` is the geometry type (`point`, `linestring`, `polygon`, `multipoint`, `multilinestring` or `multipolygon`) and `srid` an integer representing the EPSG code of a coordinate reference system.

Use of indexes

When requesting a layer through a virtual layer, the source layer indices will be used in the following ways:

- if an `=` predicate is used on the primary key column of the layer, the underlying data provider will be asked for a particular id (FilterFid)
- for any other predicates (`>`, `<=`, `!=`, etc.) or on a column without a primary key, a request built from an expression will be used to request the underlying vector data provider. It means indexes may be used on database providers if they exist.

A specific syntax exists to handle spatial predicates in requests and triggers the use of a spatial index: a hidden column named `_search_frame_` exists for each virtual layer. This column can be compared for equality to a bounding box. Example:

```
SELECT *
FROM vtab
WHERE _search_frame_=BuildMbr(-2.10,49.38,-1.3,49.99,4326)
```

Spatial binary predicates like `ST_Intersects` are sped up significantly when used in conjunction with this spatial index syntax.

13.3 Exploring Data Formats and Fields

13.3.1 Raster data

GIS raster data are matrices of discrete cells that represent features / phenomena on, above or below the earth's surface. Each cell in the raster grid has the same size, and cells are usually rectangular (in QGIS they will always be rectangular). Typical raster datasets include remote sensing data, such as aerial photography, or satellite imagery and modelled data, such as elevation or temperature.

Unlike vector data, raster data typically do not have an associated database record for each cell. They are geocoded by pixel resolution and the X/Y coordinate of a corner pixel of the raster layer. This allows QGIS to position the data correctly on the map canvas.

The GeoPackage format is convenient for storing raster data when working with QGIS. The popular and powerful GeoTiff format is a good alternative.

QGIS makes use of georeference information inside the raster layer (e.g., GeoTiff) or an associated *world file* to properly display the data.

13.3.2 Vector Data

Many of the features and tools available in QGIS work the same, regardless the vector data source. However, because of the differences in format specifications (GeoPackage, ESRI Shapefile, MapInfo and MicroStation file formats, AutoCAD DXF, PostGIS, SpatiaLite, DB2, Oracle Spatial, MSSQL Spatial databases, and many more), QGIS may handle some of their properties differently. Support is provided by the [OGR Simple Feature Library](#). This section describes how to work with these specificities.

注解: QGIS supports (multi)point, (multi)line, (multi)polygon, CircularString, CompoundCurve, CurvePolygon, MultiCurve, MultiSurface feature types, all optionally with Z and/or M values.

You should also note that some drivers don't support some of these feature types, like CircularString, CompoundCurve, CurvePolygon, MultiCurve, MultiSurface feature type. QGIS will convert them.

GeoPackage

The [GeoPackage](#) (GPKG) format is platform-independent, and is implemented as a SQLite database container, and can be used to store both vector and raster data. The format was defined by the Open Geospatial Consortium (OGC), and was published in 2014.

GeoPackage can be used to store the following in a SQLite database:

- **vector** features
- **tile matrix sets of imagery** and **raster** maps
- attributes (non-spatial data)
- extensions

Since QGIS version 3.8, GeoPackage can also store QGIS projects. GeoPackage layers can have JSON fields.

GeoPackage is the default format for vector data in QGIS.

ESRI Shapefile format

The ESRI Shapefile format is still one of the most used vector file formats, even if it has some limitations compared to for instance GeoPackage and SpatiaLite.

An ESRI Shapefile format dataset consists of several files. The following three are required:

1. `.shp` file containing the feature geometries
2. `.dbf` file containing the attributes in dBase format
3. `.shx` index file

An ESRI Shapefile format dataset can also include a file with a `.prj` suffix, which contains projection information. While it is very useful to have a projection file, it is not mandatory. A Shapefile format dataset can contain additional files. For further details, see the the ESRI technical specification at <https://www.esri.com/library/whitepapers/pdfs/shapefile.pdf>.

GDAL 3.1 has read-write support for compressed ESRI Shapefile format (`shz` and `shp.zip`).

Improving Performance for ESRI Shapefile format datasets


To improve the drawing performance for an ESRI Shapefile format dataset, you can create a spatial index. A spatial index will improve the speed of both zooming and panning. Spatial indexes used by QGIS have a `.qix` extension.

Use these steps to create the index:

1. Load an ESRI Shapefile format dataset (see *The Browser Panel*)

2. Open the *Layer Properties* dialog by double-clicking on the layer name in the legend or by right-clicking and choosing *Properties...* from the context menu
3. In the *Source* tab, click the *Create Spatial Index* button

Problem loading a .prj file

If you load an ESRI Shapefile format dataset with a `.prj` file and QGIS is not able to read the coordinate reference system from that file, you will need to define the proper projection manually in the *Layer Properties* ► *Source* tab of the layer by clicking the  `Select CRS` button. This is due to the fact that `.prj` files often do not provide the complete projection parameters as used in QGIS and listed in the *CRS* dialog.

For the same reason, if you create a new ESRI Shapefile format dataset with QGIS, two different projection files are created: a `.prj` file with limited projection parameters, compatible with ESRI software, and a `.qpj` file, providing all the parameters of the CRS. Whenever QGIS finds a `.qpj` file, it will be used instead of the `.prj`.

Delimited Text Files

Delimited text files are very common and widely used because of their simplicity and readability -- data can be viewed and edited in a plain text editor. A delimited text file is tabular data with columns separated by a defined character and rows separated by line breaks. The first row usually contains the column names. A common type of delimited text file is a CSV (Comma Separated Values), with columns separated by commas. Delimited text files can also contain positional information (see *Storing geometry information in delimited text files*).

QGIS allows you to load a delimited text file as a layer or an ordinary table (see *The Browser Panel* or *Importing a delimited text file*). First check that the file meets the following requirements:

1. The file must have a delimited header row of field names. This must be the first line of the data (ideally the first row in the text file).
2. If geometry should be enabled, the file must contain field(s) that define the geometry. These field(s) can have any name.
3. The X and Y coordinates fields (if geometry is defined by coordinates) must be specified as numbers. The coordinate system is not important.
4. If you have a CSV file with non-string columns, you must have an accompanying CSVT file (see section *Using CSVT file to control field formatting*).

The elevation point data file `elevp.csv` in the QGIS sample dataset (see section [下载示例数据](#)) is an example of a valid text file:

```
X;Y;ELEV
-300120;7689960;13
-654360;7562040;52
1640;7512840;3
[...]
```

Some things to note about the text file:

1. The example text file uses `;` (semicolon) as delimiter (any character can be used to delimit the fields).
2. The first row is the header row. It contains the fields X, Y and ELEV.
3. No quotes (") are used to delimit text fields
4. The X coordinates are contained in the X field
5. The Y coordinates are contained in the Y field

Storing geometry information in delimited text files

Delimited text files can contain geometry information in two main forms:

- As coordinates in separate columns (eg. Xcol, Ycol...), for point geometry data;
- As well-known text (WKT) representation of geometry in a single column, for any geometry type.

Features with curved geometries (CircularString, CurvePolygon and CompoundCurve) are supported. Here are some examples of geometry types in a delimited text file with geometries coded as WKT:

```
Label;WKT_geom
LineString;LINESTRING(10.0 20.0, 11.0 21.0, 13.0 25.5)
CircularString;CIRCULARSTRING(268 415,227 505,227 406)
CurvePolygon;CURVEPOLYGON(CIRCULARSTRING(1 3, 3 5, 4 7, 7 3, 1 3))
CompoundCurve;COMPOUNDCURVE((5 3, 5 13), CIRCULARSTRING(5 13, 7 15,
9 13), (9 13, 9 3), CIRCULARSTRING(9 3, 7 1, 5 3))
```

Delimited text files also support Z and M coordinates in geometries:

```
LINESTRINGZ(10.0 20.0 30.0, 11.0 21.0 31.0, 11.0 22.0 30.0)
```

Using CSV file to control field formatting

When loading CSV files, the OGR driver assumes all fields are strings (i.e. text) unless it is told otherwise. You can create a CSV file to tell OGR (and QGIS) the data type of the different columns:

Type	名称	Example
Whole number	Integer	4
Decimal number	Real	3.456
Date	Date (YYYY-MM-DD)	2016-07-28
Time	Time (HH:MM:SS+nn)	18:33:12+00
Date & Time	DateTime (YYYY-MM-DD HH:MM:SS+nn)	2016-07-28 18:33:12+00

The CSV file is a **ONE line** plain text file with the data types in quotes and separated by commas, e.g.:

```
"Integer", "Real", "String"
```

You can even specify width and precision of each column, e.g.:

```
"Integer(6)", "Real(5.5)", "String(22)"
```

This file is saved in the same folder as the .csv file, with the same name, but .csvt as the extension.

You can find more information at [GDAL CSV Driver](#).

PostGIS Layers

PostGIS layers are stored in a PostgreSQL database. The advantages of PostGIS are spatial indexing, filtering and querying capabilities. Using PostGIS, vector functions such as select and identify work more accurately than they do with OGR layers in QGIS.

小技巧: PostGIS Layers

Normally, a PostGIS layer is identified by an entry in the geometry_columns table. QGIS can load layers that do not have an entry in the geometry_columns table. This includes both tables and views. Refer to your PostgreSQL manual for information on creating views.

This section contains some details on how QGIS accesses PostgreSQL layers. Most of the time, QGIS should simply provide you with a list of database tables that can be loaded, and it will load them on request. However, if you have trouble loading a PostgreSQL table into QGIS, the information below may help you understand QGIS messages and give you directions for modifying the PostgreSQL table or view definition to allow QGIS to load it.

Primary key

QGIS requires that PostgreSQL layers contain a column that can be used as a unique key for the layer. For tables, this usually means that the table needs a primary key, or a column with a unique constraint on it. In QGIS, this column needs to be of type int4 (an integer of size 4 bytes). Alternatively, the ctid column can be used as primary key. If a table lacks these items, the oid column will be used instead. Performance will be improved if the column is indexed (note that primary keys are automatically indexed in PostgreSQL).

QGIS offers a checkbox **Select at id** that is activated by default. This option gets the ids without the attributes, which is faster in most cases.

View

If the PostgreSQL layer is a view, the same requirement exists, but views do not always have primary keys or columns with unique constraints on them. You have to define a primary key field (has to be integer) in the QGIS dialog before you can load the view. If a suitable column does not exist in the view, QGIS will not load the layer. If this occurs, the solution is to alter the view so that it does include a suitable column (a type of integer and either a primary key or with a unique constraint, preferably indexed).


As for table, a checkbox **Select at id** is activated by default (see above for the meaning of the checkbox). It can make sense to disable this option when you use expensive views.

QGIS layer_style table and database backup

If you want to make a backup of your PostGIS database using the `pg_dump` and `pg_restore` commands, and the default layer styles as saved by QGIS fail to restore afterwards, you need to set the XML option to DOCUMENT before the restore command:

```
SET XML OPTION DOCUMENT;
```

Filter database side

QGIS allows to filter features already on server side. Check *Settings* ► *Options* ► *Data Sources* ►  *Execute expressions on server-side if possible* to do so. Only supported expressions will be sent to the database. Expressions using unsupported operators or functions will gracefully fallback to local evaluation.


Support of PostgreSQL data types

Data types supported by the PostgreSQL provider include: integer, float, boolean, binary object, varchar, geometry, timestamp, array, hstore and json.

Importing Data into PostgreSQL

Data can be imported into PostgreSQL/PostGIS using several tools, including the DB Manager plugin and the command line tools `shp2pgsql` and `ogr2ogr`.

数据库管理器

QGIS comes with a core plugin named  `DB Manager`. It can be used to load data, and it includes support for schemas. See section *DB Manager Plugin* for more information.

shp2pgsql

PostGIS includes a utility called `shp2pgsql`, that can be used to import Shapefile format datasets into a PostGIS-enabled database. For example, to import a Shapefile format dataset named `lakes.shp` into a PostgreSQL database named `gis_data`, use the following command:

```
shp2pgsql -s 2964 lakes.shp lakes_new | psql gis_data
```

This creates a new layer named `lakes_new` in the `gis_data` database. The new layer will have a spatial reference identifier (SRID) of 2964. See section *Working with Projections* for more information about spatial reference systems and projections.

小技巧: Exporting datasets from PostGIS

There is also a tool for exporting PostGIS datasets to Shapefile format: `pgsql2shp`. It is shipped within your PostGIS distribution.


ogr2ogr

In addition to `shp2pgsql` and `DB Manager`, there is another tool for feeding geographical data in PostGIS: `ogr2ogr`. It is part of your GDAL installation.



To import a Shapefile format dataset into PostGIS, do the following:

```
ogr2ogr -f "PostgreSQL" PG:"dbname=postgis host=myhost.de user=postgres
password=topsecret" alaska.shp
```

This will import the Shapefile format dataset `alaska.shp` into the PostGIS database `postgis` using the user `postgres` with the password `topsecret` on the host server `myhost.de`.

Note that OGR must be built with PostgreSQL to support PostGIS. You can verify this by typing (in ):

```
ogrinfo --formats | grep -i post
```

If you prefer to use the PostgreSQL's `COPY` command instead of the default `INSERT INTO` method, you can export the following environment variable (at least available on  and ):

```
export PG_USE_COPY=YES
```

`ogr2ogr` does not create spatial indexes like `shp2pgsql` does. You need to create them manually, using the normal SQL command `CREATE INDEX` afterwards, as an extra step (as described in the next section *Improving Performance*).

Improving Performance

Retrieving features from a PostgreSQL database can be time-consuming, especially over a network. You can improve the drawing performance of PostgreSQL layers by ensuring that a PostgreSQL spatial index exists on each layer in the database. PostGIS supports creation of a GiST (Generalized Search Tree) index to speed up spatial searching (GiST index information is taken from the PostGIS documentation available at <https://postgis.net>).

小技巧: You can use the DBManager to create an index for your layer. You should first select the layer and click on *Table* ► *Edit table*, go to *Indexes* tab and click on *Add Spatial Index*.

The syntax for creating a GiST index is:

```
CREATE INDEX [indexname] ON [tablename]
  USING GIST ( [geometryfield] GIST_GEOMETRY_OPS );
```

Note that for large tables, creating the index can take a long time. Once the index is created, you should perform a `VACUUM ANALYZE`. See the PostGIS documentation (POSTGIS-PROJECT in *Literature and Web References*) for more information.

The following example creates a GiST index:

```
gsherman@madison:~/current$ psql gis_data
Welcome to psql 8.3.0, the PostgreSQL interactive terminal.

Type: \copyright for distribution terms
      \h for help with SQL commands
      \? for help with psql commands
      \g or terminate with semicolon to execute query
      \q to quit

gis_data=# CREATE INDEX sidx_alaska_lakes ON alaska_lakes
gis_data=# USING GIST (the_geom GIST_GEOMETRY_OPS);
CREATE INDEX
gis_data=# VACUUM ANALYZE alaska_lakes;
VACUUM
gis_data=# \q
gsherman@madison:~/current$
```

Vector layers crossing 180° longitude

Many GIS packages don't wrap vector maps with a geographic reference system (lat/lon) crossing the 180 degrees longitude line (http://postgis.refractor.net/documentation/manual-2.0/ST_Shift_Longitude.html). As result, if we open such a map in QGIS, we could see two widely separated locations, that should appear near each other. In [图 13.23](#), the tiny point on the far left of the map canvas (Chatham Islands) should be within the grid, to the right of the New Zealand main islands.



图 13.23: Map in lat/lon crossing the 180° longitude line

A work-around is to transform the longitude values using PostGIS and the `ST_Shift_Longitude` function. This function reads every point/vertex in every component of every feature in a geometry, and if the longitude coordinate

is $< 0^\circ$, it adds 360° to it. The result is a $0^\circ - 360^\circ$ version of the data to be plotted in a 180° -centric map.

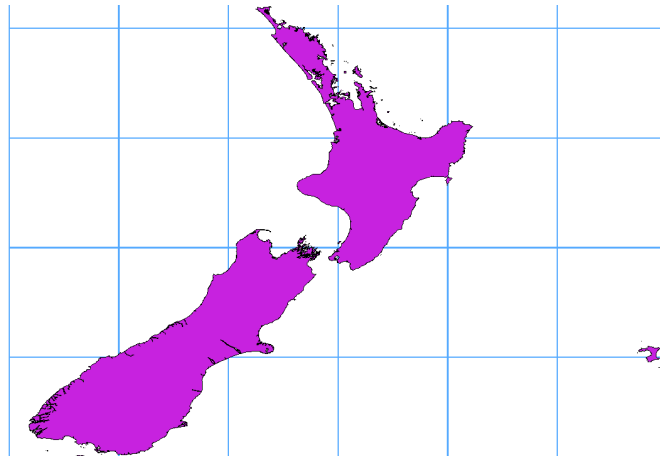


图 13.24: Crossing 180° longitude applying the `ST_Shift_Longitude` function

Usage

- Import data into PostGIS (*Importing Data into PostgreSQL*) using, for example, the DB Manager plugin.
- Use the PostGIS command line interface to issue the following command (in this example, "TABLE" is the actual name of your PostGIS table): `gis_data=# update TABLE set the_geom=ST_Shift_Longitude(the_geom);`
- If everything went well, you should receive a confirmation about the number of features that were updated. Then you'll be able to load the map and see the difference (*Figure_vector_crossing_map*).

Spatialite Layers

If you want to save a vector layer using the Spatialite format, you can do this by following instructions at *Creating new layers from an existing layer*. You select Spatialite as *Format* and enter both *File name* and *Layer name*.

Also, you can select SQLite as format and then add `SPATIALITE=YES` in the *Custom Options* ► *Data source* field. This tells GDAL to create a Spatialite database. See also <https://gdal.org/drivers/vector/sqlite.html>.

QGIS also supports editable views in Spatialite. For Spatialite data management, you can also use the core plugin *DB Manager*.

If you want to create a new Spatialite layer, please refer to section *Creating a new Spatialite layer*.

GeoJSON specific parameters

When *exporting layers* to GeoJSON, there are some specific *Layer Options* available. These options come from GDAL which is responsible for the writing of the file:

- `COORDINATE_PRECISION` the maximum number of digits after the decimal separator to write in coordinates. Defaults to 15 (note: for Lat Lon coordinates 6 is considered enough). Truncation will occur to remove trailing zeros.
- `RFC7946` by default GeoJSON 2008 will be used. If set to YES, the updated RFC 7946 standard will be used. Default is NO (thus GeoJSON 2008). See <https://gdal.org/drivers/vector/geojson.html#rfc-7946-write-support> for the main differences, in short: only EPSG:4326 is allowed, other crs's will be transformed, polygons will be written such as to follow the right-hand rule for orientation, values of a "bbox" array are [west, south, east, north], not [minx, miny, maxx, maxy]. Some extension member names are forbidden in FeatureCollection, Feature and Geometry objects, the default coordinate precision is 7 decimal digits

- `WRITE_BBOX` set to YES to include the bounding box of the geometries at the feature and feature collection level

Besides GeoJSON there is also an option to export to "GeoJSON - Newline Delimited" (see <https://gdal.org/drivers/vector/geojsonseq.html>). Instead of a FeatureCollection with Features, you can stream one type (probably only Features) sequentially separated with newlines.

GeoJSON - Newline Delimited has some specific Layer options available too:

- `COORDINATE_PRECISION` see above (same as for GeoJSON)
- `RS` whether to start records with the `RS=0x1E` character. The difference is how the features are separated: only by a newline (LF) character (Newline Delimited JSON, `geojsonl`) or by also prepending a record-separator (RS) character (giving GeoJSON Text Sequences, `geojsons`). Default to NO. Files are given the `.json` extension if extension is not provided.

DB2 Spatial Layers

IBM DB2 for Linux, Unix and Windows (DB2 LUW), IBM DB2 for z/OS (mainframe) and IBM DashDB products allow users to store and analyse spatial data in relational table columns. The DB2 provider for QGIS supports the full range of visualization, analysis and manipulation of spatial data in these databases.

User documentation on these capabilities can be found at the [DB2 z/OS KnowledgeCenter](#), [DB2 LUW KnowledgeCenter](#) and [DB2 DashDB KnowledgeCenter](#).

For more information about working with the DB2 spatial capabilities, check out the [DB2 Spatial Tutorial](#) on IBM DeveloperWorks.

The DB2 provider currently only supports the Windows environment through the Windows ODBC driver.

The client running QGIS needs to have one of the following installed:

- DB2 LUW
- IBM Data Server Driver Package
- IBM Data Server Client

To open a DB2 data in QGIS, see the [The Browser Panel](#) or [Loading a Database Layer](#) section.

If you are accessing a DB2 LUW database on the same machine or using DB2 LUW as a client, the DB2 executables and supporting files need to be included in the Windows path. This can be done by creating a batch file like the following with the name `db2.bat` and including it in the directory `%OSGEO4W_ROOT%/etc/ini`:

```
@echo off
REM Point the following to where DB2 is installed
SET db2path=C:\Program Files (x86)\sqllib
REM This should usually be ok - modify if necessary
SET gskpath=C:\Program Files (x86)\ibm\gsk8
SET Path=%db2path%\BIN;%db2path%\FUNCTION;%gskpath%\lib64;%gskpath%\lib;%path%
```






















14.1 The Vector Properties Dialog

The *Layer Properties* dialog for a vector layer provides general settings to manage appearance of layer features in the map (symbolology, labeling, diagrams), interaction with the mouse (actions, map tips, form design). It also provides information about the layer.

To access the *Layer Properties* dialog:

- In the *Layers* panel, double-click the layer or right-click and select *Properties...* from the pop-up menu;
- Go to *Layer ► Layer Properties...* menu when the layer is selected.

The vector *Layer Properties* dialog provides the following sections:

 <i>Information</i>	 <i>Source</i>	 <i>Symbolology</i> ^[1]
 <i>Labels</i> ^[1]	 <i>Mask</i> ^[1]	 <i>3D View</i> ^[1]
 <i>Diagrams</i>	 <i>Fields</i>	 <i>Attributes Form</i>
 <i>Joins</i>	 <i>Auxiliary Storage</i>	 <i>Actions</i>
 <i>Display</i>	 <i>Rendering</i>	 <i>Temporal</i>
 <i>Variables</i>	 <i>Metadata</i>	 <i>Dependencies</i>
 <i>Legend</i>	 <i>QGIS Server</i>	 <i>Digitizing</i>
<i>External plugins</i> ^[2] tabs		

^[1] Also available in the *Layer styling panel*


^[2] *External plugins* you install can optionally add tabs to this dialog. Those are not presented in this document. Refer to their documentation.

小技巧: Share full or partial properties of the layer styles

The *Style* menu at the bottom of the dialog allows you to import or export these or part of these properties from/to several destination (file, clipboard, database). See [Managing Custom Styles](#).


注解: Because properties (symbolology, label, actions, default values, forms...) of embedded layers (see [Nesting Projects](#)) are pulled from the original project file and to avoid changes that may break this behavior, the layer properties dialog is made unavailable for these layers.

14.1.1 Information Properties

 The *Information* tab is read-only and represents an interesting place to quickly grab summarized information and metadata on the current layer. Provided information are:

- based on the provider of the layer (format of storage, path, geometry type, data source encoding, extent...);
- picked from the *filled metadata* (access, links, contacts, history...);
- or related to its geometry (spatial extent, CRS...) or its attributes (number of fields, characteristics of each...).

14.1.2 Source Properties

 Use this tab to define general settings for the vector layer.

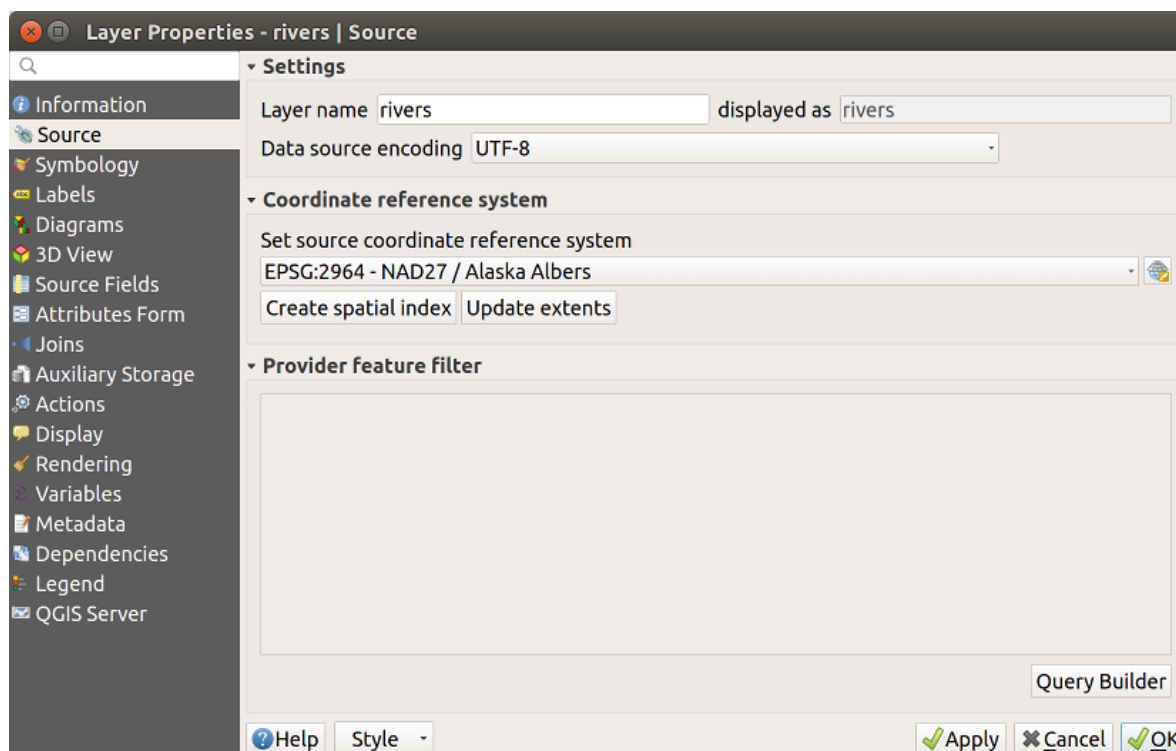



图 14.1: Source tab in vector Layer Properties dialog

Other than setting the *Layer name* to display in the *Layers Panel*, available options include:

坐标参照系

- Displays the layer's *Coordinate Reference System (CRS)*. You can change the layer's CRS, selecting a recently used one in the drop-down list or clicking on  Select CRS button (see *Coordinate Reference System Selector*). Use this process only if the CRS applied to the layer is a wrong one or if none was applied. If you wish to reproject your data into another CRS, rather use layer reprojection algorithms from Processing or *Save it into another layer*.
- *Create spatial index* (only for OGR-supported formats).
- *Update extents* information for a layer.

Query Builder

The *Query Builder* dialog is accessible through the eponym button at the bottom of the *Source* tab in the Layer Properties dialog, under the *Provider feature filter* group.

The Query Builder provides an interface that allows you to define a subset of the features in the layer using a SQL-like WHERE clause and to display the result in the main window. As long as the query is active, only the features corresponding to its result are available in the project.

You can use one or more layer attributes to define the filter in the Query Builder. The use of more than one attribute is shown in [图 14.2](#). In the example, the filter combines the attributes

- toa (DateTime field: `cast("toa" as character) > '2017-05-17' and cast("toa" as character) < '2019-12-24T18:00:00'`),
- name (String field: `"name" > 'S'`) and
- FID (Integer field: `FID > 10`)

using the AND, OR and NOT operators and parenthesis. This syntax (including the DateTime format for the toa field) works for GeoPackage datasets.

The filter is made at the data provider (OGR, PostgreSQL, MSSQL...) level. So the syntax depends on the data provider (DateTime is for instance not supported for the ESRI Shapefile format). The complete expression:

```
cast("toa" as character) > '2017-05-17' AND
cast("toa" as character) < '2019-12-24T18:00:00' AND
NOT ("name" > 'S' OR FID > 10)
```

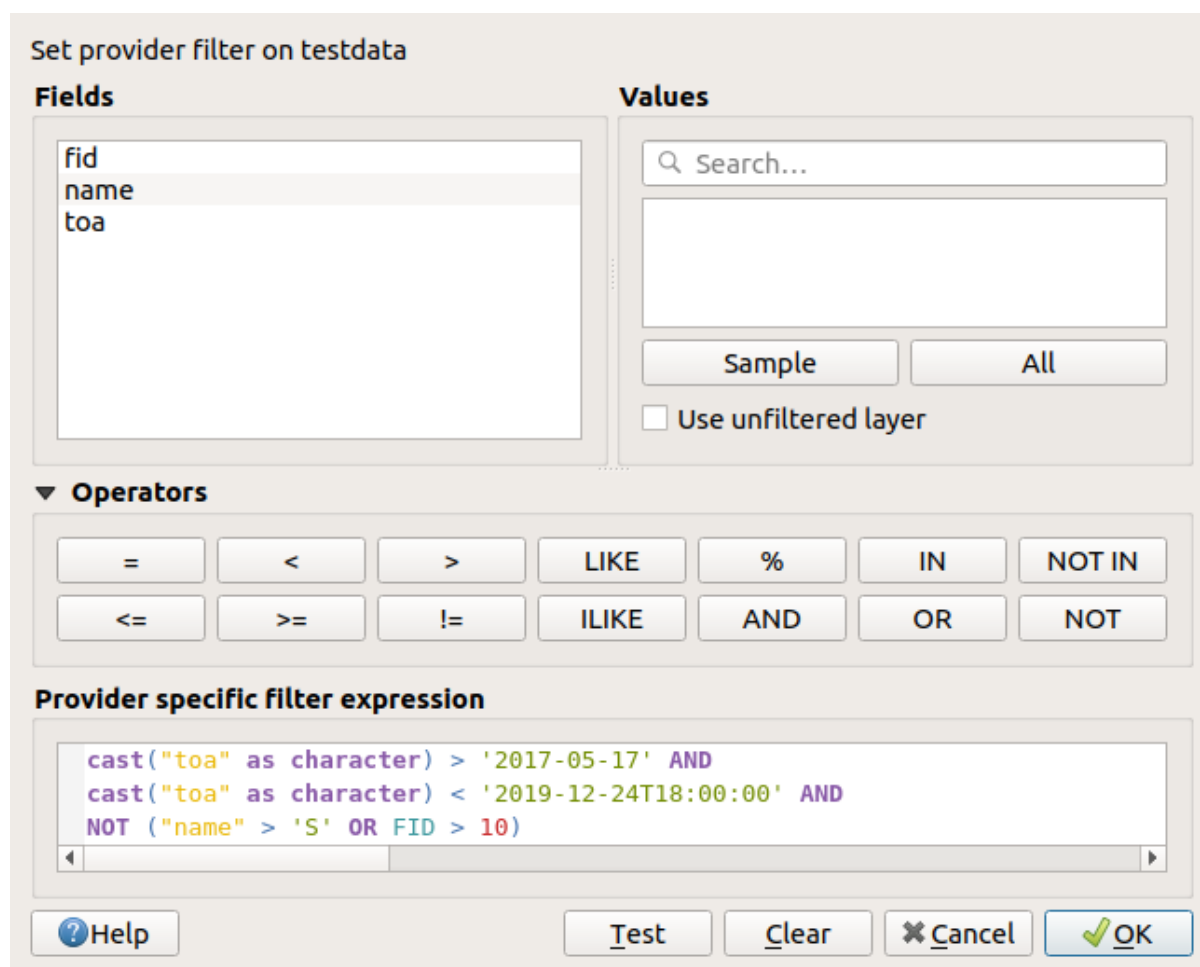


图 14.2: Query Builder

You can also open the *Query Builder* dialog using the *Filter...* option from the *Layer* menu or the layer contextual menu. The *Fields*, *Values* and *Operators* sections in the dialog help you to construct the SQL-like query exposed in the *Provider specific filter expression* box.

The **Fields** list contains all the fields of the layer. To add an attribute column to the SQL WHERE clause field, double-click its name or just type it into the SQL box.


The **Values** frame lists the values of the currently selected field. To list all unique values of a field, click the *All* button. To instead list the first 25 unique values of the column, click the *Sample* button. To add a value to the SQL WHERE clause field, double click its name in the Values list. You can use the search box at the top of the Values frame to easily browse and find attribute values in the list.

The **Operators** section contains all usable operators. To add an operator to the SQL WHERE clause field, click the appropriate button. Relational operators ($=$, $>$, $<$, ...), string comparison operator (*LIKE*), and logical operators (*AND*, *OR*, ...) are available.

The *Test* button helps you check your query and displays a message box with the number of features satisfying the current query. Use the *Clear* button to wipe the SQL query and revert the layer to its original state (ie, fully load all the features).

When a filter is applied, QGIS treats the resulting subset acts as if it were the entire layer. For example if you applied the filter above for 'Borough' (`"TYPE_2" = 'Borough'`), you can not display, query, save or edit *Anchorage*, because that is a 'Municipality' and therefore not part of the subset.

小技巧: Filtered layers are indicated in the Layers Panel

In the *Layers* panel, filtered layer is listed with a  Filter icon next to it indicating the query used when the mouse hovers over the button. Double-click the icon opens the *Query Builder* dialog for edit.

14.1.3 Symbology Properties



The Symbology tab provides you with a comprehensive tool for rendering and symbolizing your vector data. You can use tools that are common to all vector data, as well as special symbolizing tools that were designed for the different kinds of vector data. However all types share the following dialog structure: in the upper part, you have a widget that helps you prepare the classification and the symbol to use for features and at the bottom the *Layer rendering* widget.

小技巧: Switch quickly between different layer representations

Using the *Styles ► Add* menu at the bottom of the *Layer Properties* dialog, you can save as many styles as needed. A style is the combination of all properties of a layer (such as symbology, labeling, diagram, fields form, actions...) as you want. Then, simply switch between styles from the context menu of the layer in *Layers Panel* to automatically get different representations of your data.

小技巧: Export vector symbology

You have the option to export vector symbology from QGIS into Google *.kml, *.dxf and MapInfo *.tab files. Just open the right mouse menu of the layer and click on *Save As...* to specify the name of the output file and its format. In the dialog, use the *Symbology export* menu to save the symbology either as *Feature symbology ►* or as *Symbol layer symbology ►*. If you have used symbol layers, it is recommended to use the second setting.


Features rendering

The renderer is responsible for drawing a feature together with the correct symbol. Regardless layer geometry type, there are four common types of renderers: single symbol, categorized, graduated and rule-based. For point layers, there are a point displacement and a heatmap renderers available while polygon layers can also be rendered with the inverted polygons and 2.5 D renderers.

There is no continuous color renderer, because it is in fact only a special case of the graduated renderer. The categorized and graduated renderers can be created by specifying a symbol and a color ramp - they will set the colors for symbols appropriately. For each data type (points, lines and polygons), vector symbol layer types are available. Depending on the chosen renderer, the dialog provides different additional sections.

注解: If you change the renderer type when setting the style of a vector layer the settings you made for the symbol will be maintained. Be aware that this procedure only works for one change. If you repeat changing the renderer type the settings for the symbol will get lost.

Single Symbol Renderer

The  *Single Symbol* renderer is used to render all features of the layer using a single user-defined symbol. See *The Symbol Selector* for further information about symbol representation.

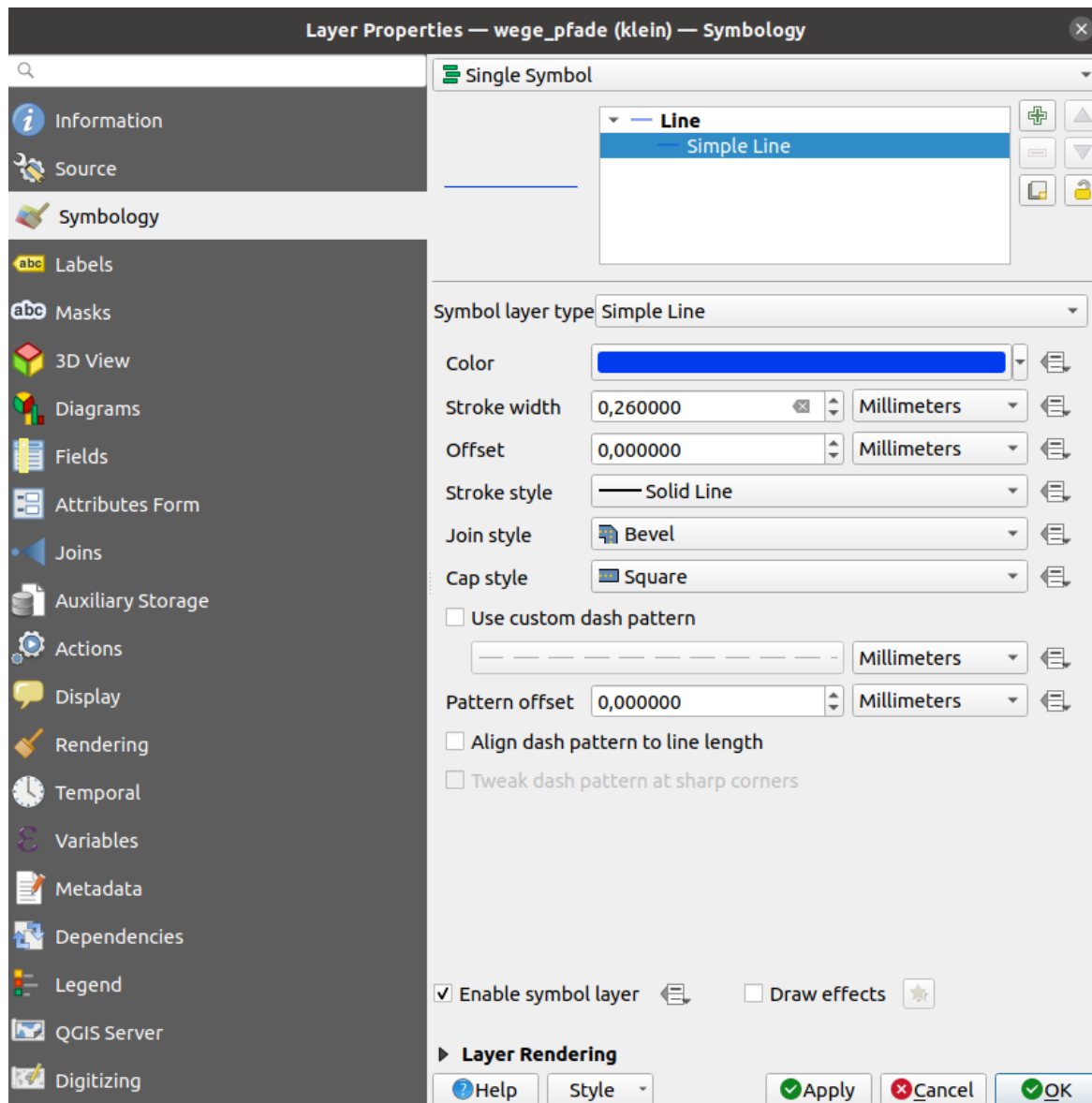



图 14.3: Single symbol line properties


No Symbols Renderer

The  *No Symbols* renderer is a special use case of the Single Symbol renderer as it applies the same rendering to all features. Using this renderer, no symbol will be drawn for features, but labeling, diagrams and other non-symbol parts will still be shown.

Selections can still be made on the layer in the canvas and selected features will be rendered with a default symbol. Features being edited will also be shown.

This is intended as a handy shortcut for layers which you only want to show labels or diagrams for, and avoids the need to render symbols with totally transparent fill/border to achieve this.

Categorized Renderer

The  *Categorized* renderer is used to render the features of a layer, using a user-defined symbol whose aspect reflects the discrete values of a field or an expression.

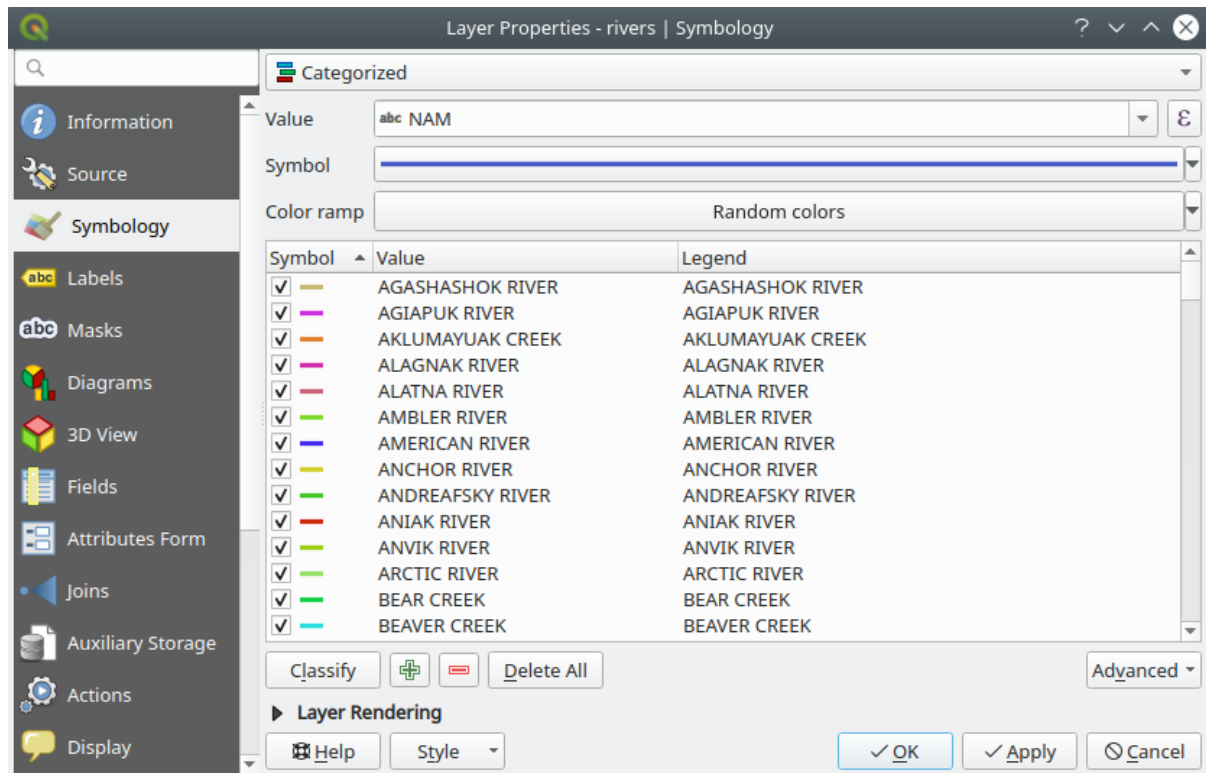



图 14.4: Categorized Symbolizing options

To use categorized symbology for a layer:

1. Select the *Value* of classification: it can be an existing field or an *expression* you can type in the box or build using the associated  button. Using expressions for categorizing avoids the need to create an ad hoc field for symbology purposes (eg, if your classification criteria are derived from one or more attributes).

The expression used to classify features can be of any type; eg, it can:

- be a comparison. In this case, QGIS returns values 1 (**True**) and 0 (**False**). Some examples:

```
myfield >= 100
$id = @atlas_featureid
```

(下页继续)

(续上页)

```
myfield % 2 = 0
within( $geometry, @atlas_geometry )
```

- combine different fields:

```
concat( field_1, ' ', field_2 )
```

- be a calculation on fields:

```
myfield % 2
year( myfield )
field_1 + field_2
substr( field_1, -3 )
```

- be used to transform linear values to discrete classes, e.g.:


```
CASE WHEN x > 1000 THEN 'Big' ELSE 'Small' END
```

- combine several discrete values into a single category, e.g.:

```
CASE
WHEN building IN ('residence', 'mobile home') THEN 'residential'
WHEN building IN ('commercial', 'industrial') THEN 'Commercial and
↪Industrial'
END
```

小技巧: While you can use any kind of expression to categorize features, for some complex expressions it might be simpler to use *rule-based rendering*.



2. Configure the *Symbol*, which will be used as base symbol for all the classes;
3. Indicate the *Color ramp*, ie the range of colors from which the color applied to each symbol is selected.

Besides the common options of the *color ramp widget*, you can apply a  *Random Color Ramp* to the categories. You can click the *Shuffle Random Colors* entry to regenerate a new set of random colors if you are not satisfied.

4. Then click on the *Classify* button to create classes from the distinct values of the provided field or expression.
5. *Apply* the changes if the *live update* is not in use and each feature on the map canvas will be rendered with the symbol of its class.

By default, QGIS appends an *all other values* class to the list. While empty at the beginning, this class is used as a default class for any feature not falling into the other classes (eg, when you create features with new values for the classification field / expression).

Further tweaks can be done to the default classification:

- You can  Add new categories,  Remove selected categories or *Delete All* of them.
- A class can be disabled by unchecking the checkbox to the left of the class name; the corresponding features are hidden on the map.
- Drag-and-drop the rows to reorder the classes
- To change the symbol, the value or the legend of a class, double click the item.

Right-clicking over selected item(s) shows a contextual menu to:

- *Copy Symbol* and *Paste Symbol*, a convenient way to apply the item's representation to others
- *Change Color...* of the selected symbol(s)
- *Change Opacity...* of the selected symbol(s)

- *Change Output Unit...* of the selected symbol(s)
- *Change Width...* of the selected line symbol(s)
- *Change Size...* of the selected point symbol(s)
- *Change Angle...* of the selected point symbol(s)
- *Merge Categories*: Groups multiple selected categories into a single one. This allows simpler styling of layers with a large number of categories, where it may be possible to group numerous distinct categories into a smaller and more manageable set of categories which apply to multiple values.

小技巧: Since the symbol kept for the merged categories is the one of the topmost selected category in the list, you may want to move the category whose symbol you wish to reuse to the top before merging.




- *Unmerge Categories* that were previously merged

The *Advanced* menu gives access to options to speed classification or fine-tune the symbols rendering:


- *Match to saved symbols*: Using the *symbols library*, assigns to each category a symbol whose name represents the classification value of the category
- *Match to symbols from file...*: Provided a file with symbols, assigns to each category a symbol whose name represents the classification value of the category
- *Symbol levels...* to define the order of symbols rendering.

小技巧: Edit categories directly from the *Layers* panel

When a layer symbology is based on a *categorized*, *graduated* or *rule-based* symbology mode, you can edit each of the categories from the *Layers* Panel. Right-click on a sub-item of the layer and you will:


-  *Toggle items visibility*
 -  *Show all items*
 -  *Hide all items*
 - Modify the symbol color thanks to the *color selector* wheel
 - *Edit symbol...* from the *symbol selector* dialog
 - *Copy symbol*
 - *Paste symbol*
-

Graduated Renderer

The  *Graduated* renderer is used to render all the features from a layer, using an user-defined symbol whose color or size reflects the assignment of a selected feature's attribute to a class.

Like the Categorized Renderer, the Graduated Renderer allows you to define rotation and size scale from specified columns.

Also, analogous to the Categorized Renderer, it allows you to select:

- The value (using the fields listbox or the  *Set value expression* function)
- The symbol (using the Symbol selector dialog)
- The legend format and the precision
- The method to use to change the symbol: color or size

- The colors (using the color Ramp list) if the color method is selected
- The size (using the size domain and its unit)

Then you can use the Histogram tab which shows an interactive histogram of the values from the assigned field or expression. Class breaks can be moved or added using the histogram widget.

注解: You can use Statistical Summary panel to get more information on your vector layer. See *Statistical Summary Panel*.

Back to the Classes tab, you can specify the number of classes and also the mode for classifying features within the classes (using the Mode list). The available modes are:

- Equal Count (Quantile): each class will have the same number of elements (the idea of a boxplot).
- Equal Interval: each class will have the same size (e.g. with the values from 1 to 16 and four classes, each class will have a size of four).
- Logarithmic scale: suitable for data with a wide range of values. Narrow classes for low values and wide classes for large values (e.g. for decimal numbers with range [0..100] and two classes, the first class will be from 0 to 10 and the second class from 10 to 100).
- Natural Breaks (Jenks): the variance within each class is minimized while the variance between classes is maximized.
- Pretty Breaks: computes a sequence of about $n+1$ equally spaced nice values which cover the range of the values in x . The values are chosen so that they are 1, 2 or 5 times a power of 10. (based on pretty from the R statistical environment <https://www.rdocumentation.org/packages/base/topics/pretty>).
- Standard Deviation: classes are built depending on the standard deviation of the values.

The listbox in the center part of the *Symbolology* tab lists the classes together with their ranges, labels and symbols that will be rendered.

Click on **Classify** button to create classes using the chosen mode. Each classes can be disabled unchecking the checkbox at the left of the class name.

To change symbol, value and/or label of the class, just double click on the item you want to change.

Right-clicking over selected item(s) shows a contextual menu to:

- *Copy Symbol* and *Paste Symbol*, a convenient way to apply the item's representation to others
- *Change Color...* of the selected symbol(s)
- *Change Opacity...* of the selected symbol(s)
- *Change Output Unit...* of the selected symbol(s)
- *Change Width...* of the selected line symbol(s)
- *Change Size...* of the selected point symbol(s)
- *Change Angle...* of the selected point symbol(s)

The example in [图 14.5](#) shows the graduated rendering dialog for the `major_rivers` layer of the QGIS sample dataset.

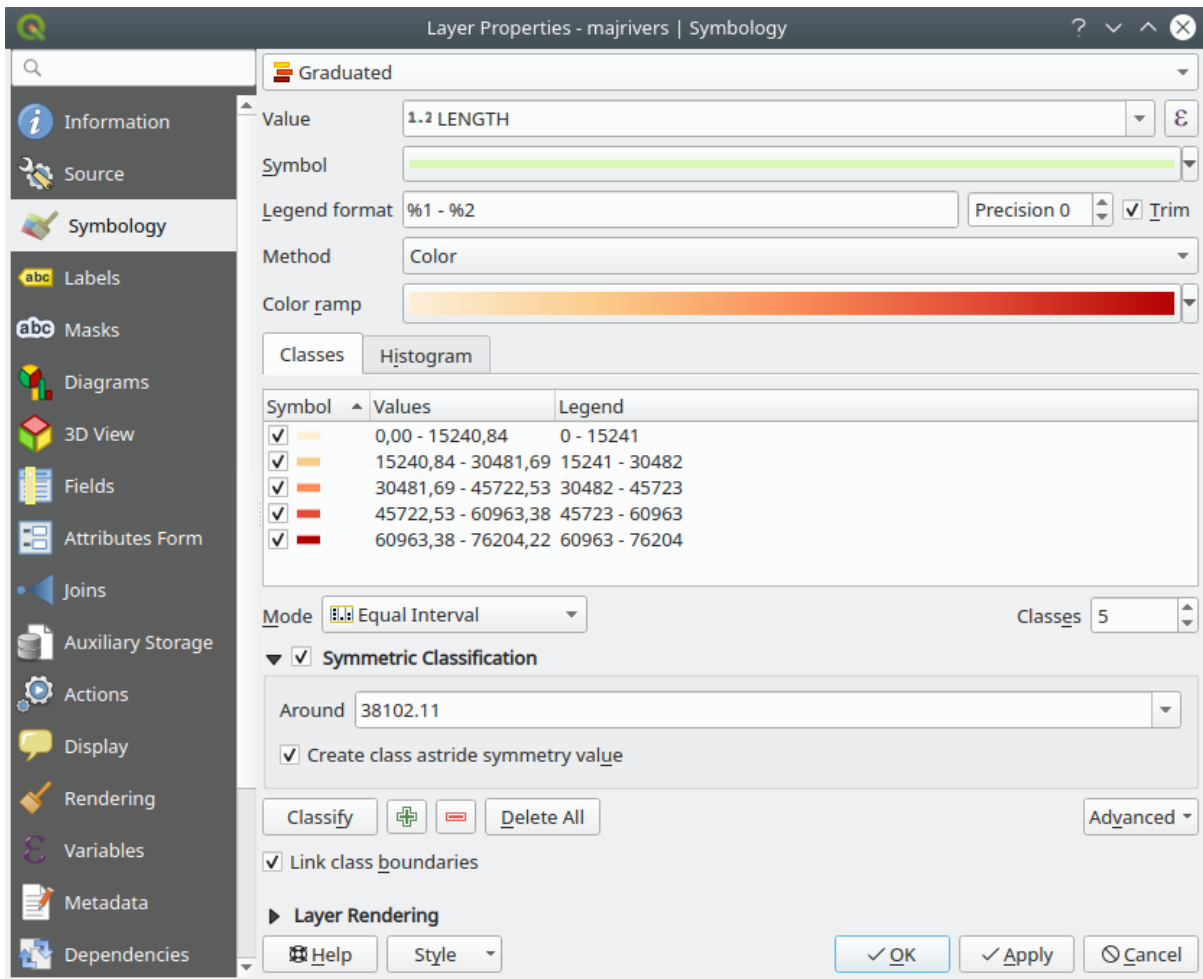



图 14.5: Graduated Symbolizing options

小技巧: Thematic maps using an expression


Categorized and graduated thematic maps can be created using the result of an expression. In the properties dialog for vector layers, the attribute chooser is extended with a  *Set column expression* function. So you don't need to write the classification attribute to a new column in your attribute table if you want the classification attribute to be a composite of multiple fields, or a formula of some sort.

Proportional Symbol and Multivariate Analysis

Proportional Symbol and Multivariate Analysis are not rendering types available from the Symbology rendering drop-down list. However with the *data-defined override* options applied over any of the previous rendering options, QGIS allows you to display your point and line data with such representation.

Creating proportional symbol

To apply a proportional rendering:

1. First apply to the layer the *single symbol renderer*.
2. Then set the symbol to apply to the features.
3. Select the item at the upper level of the symbol tree, and use the  *Data-defined override* button next to the *Size* (for point layer) or *Width* (for line layer) option.

4. Select a field or enter an expression, and for each feature, QGIS will apply the output value to the property and proportionally resize the symbol in the map canvas.

If need be, use the *Size assistant...* option of the  menu to apply some transformation (exponential, flannery...) to the symbol size rescaling (see *Using the data-defined assistant interface* for more details).

You can choose to display the proportional symbols in the *Layers panel* and the *print layout legend item*: unfold the *Advanced* drop-down list at the bottom of the main dialog of the *Symbology* tab and select **Data-defined size legend...** to configure the legend items (see *Data-defined size legend* for details).

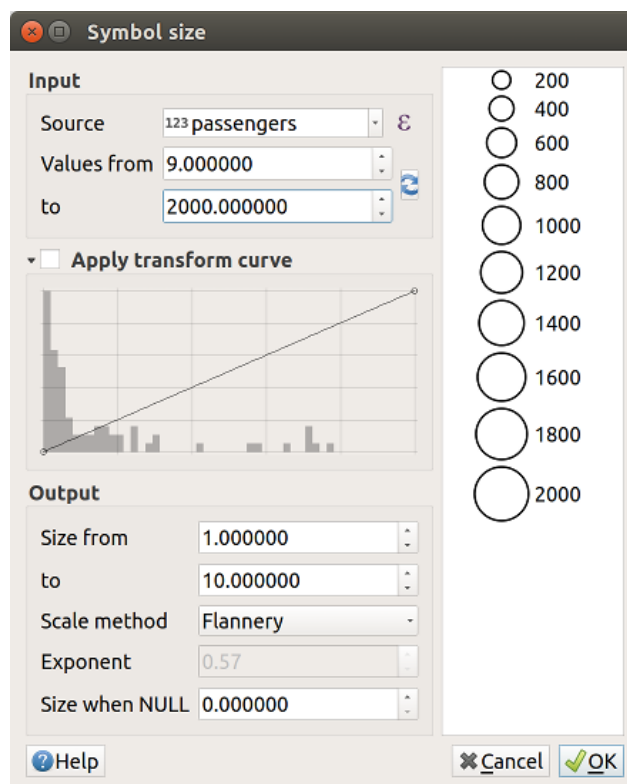



图 14.6: Scaling airports size based on elevation of the airport

Creating multivariate analysis

A multivariate analysis rendering helps you evaluate the relationship between two or more variables e.g., one can be represented by a color ramp while the other is represented by a size.

The simplest way to create multivariate analysis in QGIS is to:

1. First apply a categorized or graduated rendering on a layer, using the same type of symbol for all the classes.
2. Then, apply a proportional symbology on the classes:
 1. Click on the *Change* button above the classification frame: you get the *The Symbol Selector* dialog.
 2. Rescale the size or width of the symbol layer using the  *data defined override* widget as seen above.

Like the proportional symbol, the scaled symbology can be added to the layer tree, on top of the categorized or graduated classes symbols using the *data defined size legend* feature. And both representation are also available in the print layout legend item.

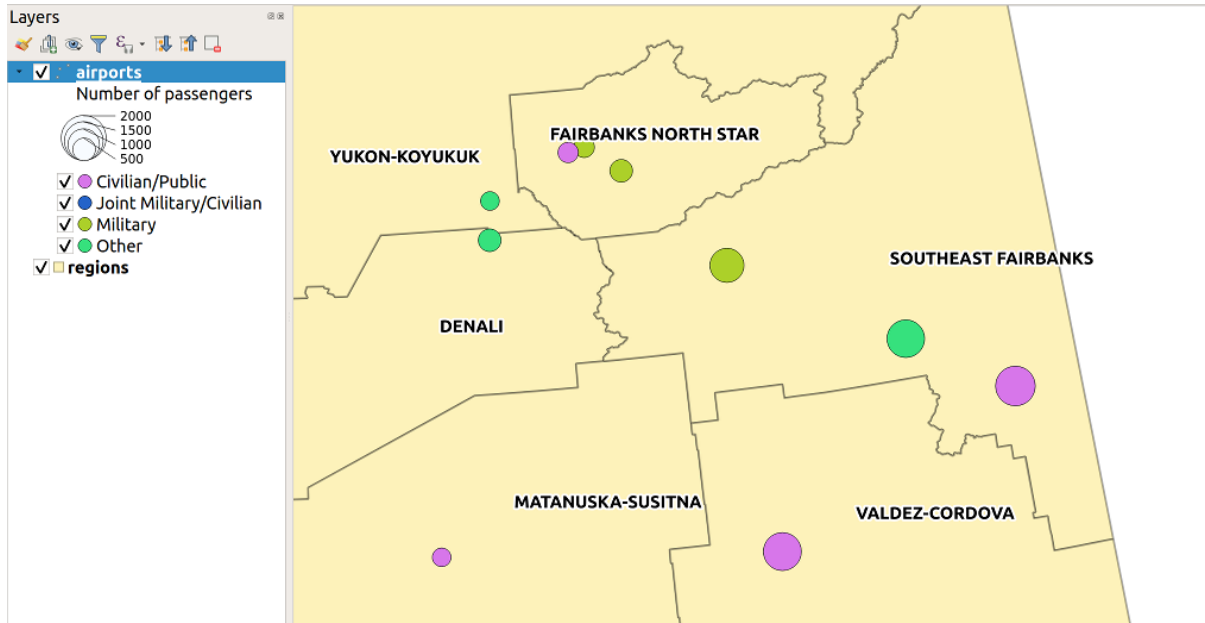








图 14.7: Multivariate example with scaled size legend

Rule-based Renderer

The  *Rule-based* renderer is used to render all the features from a layer, using rule-based symbols whose aspect reflects the assignment of a selected feature's attribute to a class. The rules are based on SQL statements and can be nested. The dialog allows rule grouping by filter or scale, and you can decide if you want to enable symbol levels or use only the first-matched rule.

To create a rule:

1. Activate an existing row by double-clicking it (by default, QGIS adds a symbol without a rule when the rendering mode is enabled) or click the  *Edit rule* or  *Add rule* button.
2. In the *Edit Rule* dialog that opens, you can define a label to help you identify each rule. This is the label that will be displayed in the *Layers Panel* and also in the print composer legend.
3. Manually enter an expression in the text box next to the  *Filter* option or press the  button next to it to open the expression string builder dialog.
4. Use the provided functions and the layer attributes to build an *expression* to filter the features you'd like to retrieve. Press the *Test* button to check the result of the query.
5. You can enter a longer label to complete the rule description.
6. You can use the  *Scale Range* option to set scales at which the rule should be applied.
7. Finally, configure the *symbol to use* for these features.
8. And press *OK*.

A new row summarizing the rule is added to the Layer Properties dialog. You can create as many rules as necessary following the steps above or copy pasting an existing rule. Drag-and-drop the rules on top of each other to nest them and refine the upper rule features in subclasses.

Selecting a rule, you can also organize its features in subclasses using the *Refine selected rules* drop-down menu. Automated rule refinement can be based on:

- **scales;**

- **categories:** applying a *categorized renderer*;
- or **ranges:** applying a *graduated renderer*.

Refined classes appear like sub-items of the rule, in a tree hierarchy and like above, you can set symbology of each class.

In the *Edit rule* dialog, you can avoid writing all the rules and make use of the *Else* option to catch all the features that do not match any of the other rules, at the same level. This can also be achieved by writing `ELSE` in the *Rule* column of the *Layer Properties* ► *Symbology* ► *Rule-based* dialog.

Right-clicking over selected item(s) shows a contextual menu to:

- *Copy* and *Paste*, a convenient way to create new item(s) based on existing item(s)
- *Copy Symbol* and *Paste Symbol*, a convenient way to apply the item’s representation to others
- *Change Color...* of the selected symbol(s)
- *Change Opacity...* of the selected symbol(s)
- *Change Output Unit...* of the selected symbol(s)
- *Change Width...* of the selected line symbol(s)
- *Change Size...* of the selected point symbol(s)
- *Change Angle...* of the selected point symbol(s)
- *Refine Current Rule:* open a submenu that allows to refine the current rule with **scales**, **categories** (categorized renderer) or **Ranges** (graduated renderer).

The created rules also appear in a tree hierarchy in the map legend. Double-click the rules in the map legend and the Symbology tab of the layer properties appears showing the rule that is the background for the symbol in the tree.

The example in 图 14.8 shows the rule-based rendering dialog for the rivers layer of the QGIS sample dataset.

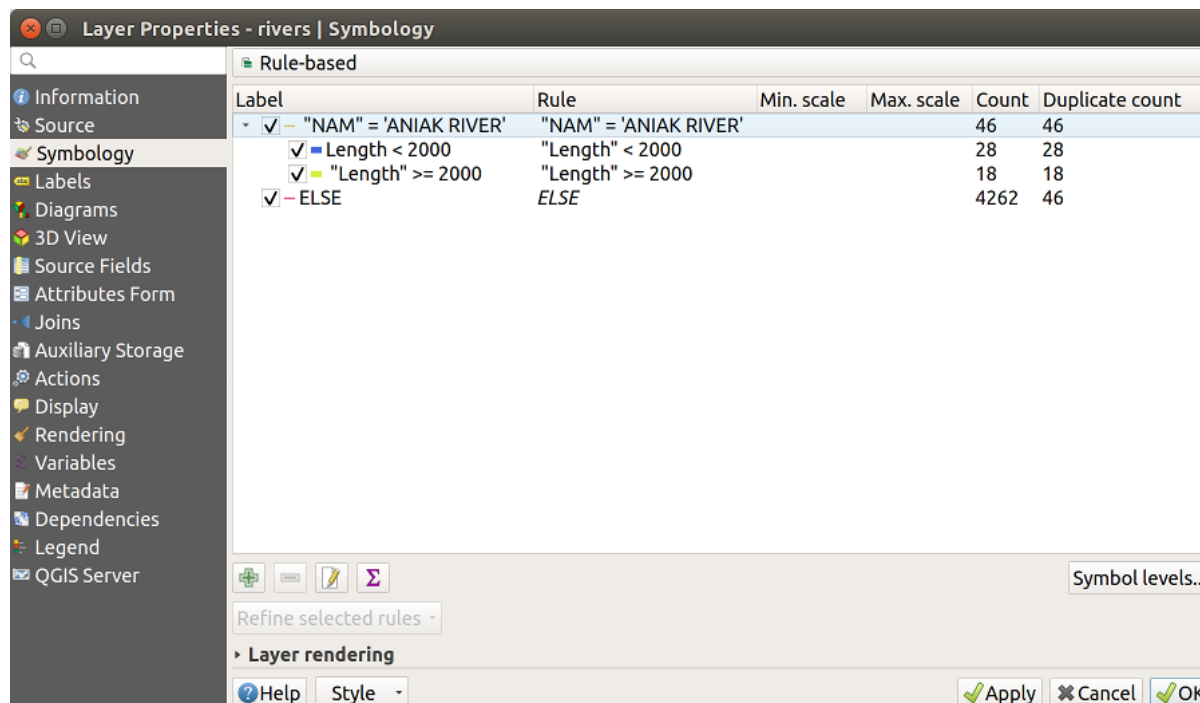



图 14.8: Rule-based Symbolizing options

Point displacement Renderer

The  *Point Displacement* renderer works to visualize all features of a point layer, even if they have the same location. To do this, the renderer takes the points falling in a given *Distance* tolerance from each other and places them around their barycenter following different *Placement methods*:

- **Ring:** places all the features on a circle whose radius depends on the number of features to display.
- **Concentric rings:** uses a set of concentric circles to show the features.
- **Grid:** generates a regular grid with a point symbol at each intersection.

The *Center symbol* widget helps you customize the symbol and color of the middle point. For the distributed points symbols, you can apply any of the *No symbols*, *Single symbol*, *Categorized*, *Graduated* or *Rule-based* renderer using the *Renderer* drop-down list and customize them using the *Renderer Settings...* button.

While the minimal spacing of the *Displacement lines* depends on the point symbol renderer's, you can still customize some of its settings such as the *Stroke width*, *Stroke color* and *Size adjustment* (eg, to add more spacing between the rendered points).

Use the *Labels* group options to perform points labeling: the labels are placed near the displaced position of the symbol, and not at the feature real position. Other than the *Label attribute*, *Label font* and *Label color*, you can set the *Minimum map scale* to display the labels.

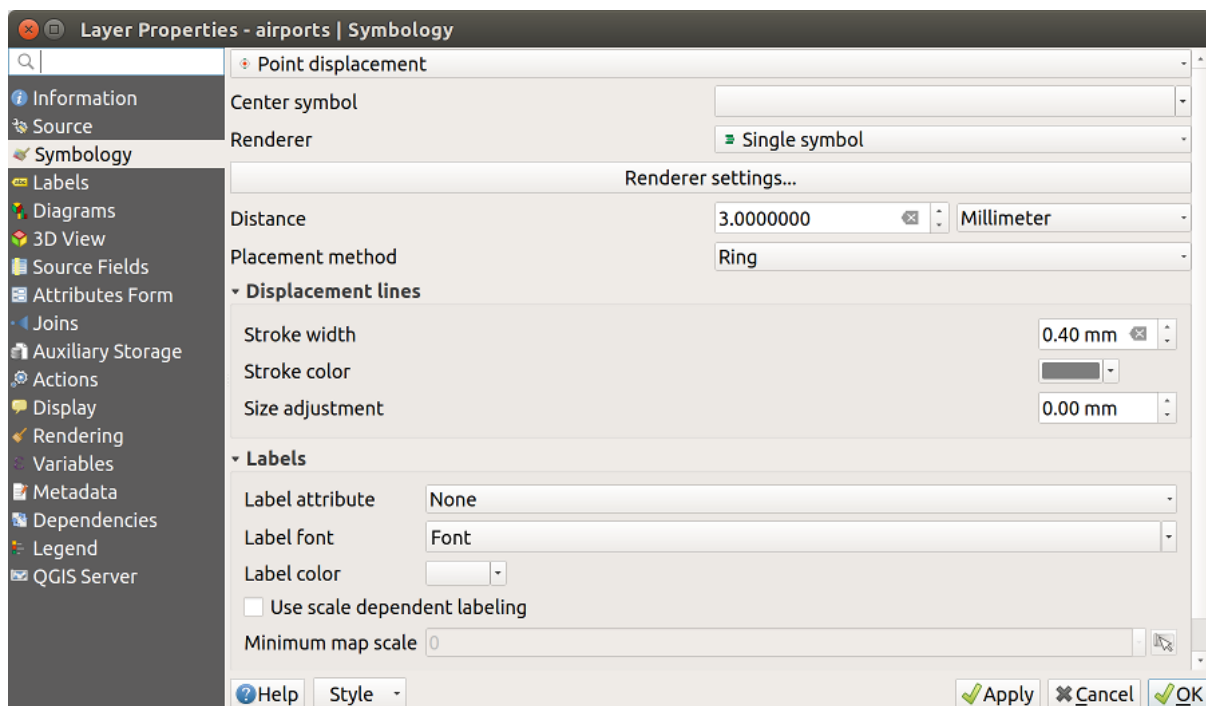



图 14.9: Point displacement dialog

注解: Point Displacement renderer does not alter feature geometry, meaning that points are not moved from their position. They are still located at their initial place. Changes are only visual, for rendering purpose. Use instead the Processing *Points displacement* algorithm if you want to create displaced features.

Point Cluster Renderer

Unlike the  *Point Displacement* renderer which blows up nearest or overlaid point features placement, the *Point Cluster* renderer groups nearby points into a single rendered marker symbol. Based on a specified *Distance*, points that fall within from each others are merged into a single symbol. Points aggregation is made based on the closest group being formed, rather than just assigning them the first group within the search distance.

From the main dialog, you can:

- set the symbol to represent the point cluster in the *Cluster symbol*; the default rendering displays the number of aggregated features thanks to the `@cluster_size` variable on Font marker symbol layer.
- use the *Renderer* drop-down list to apply any of the other feature rendering types to the layer (single, categorized, rule-based...). Then, push the *Renderer Settings...* button to configure features' symbology as usual. Note that this renderer is only visible on features that are not clustered. Also, when the symbol color is the same for all the point features inside a cluster, that color sets the `@cluster_color` variable of the cluster.

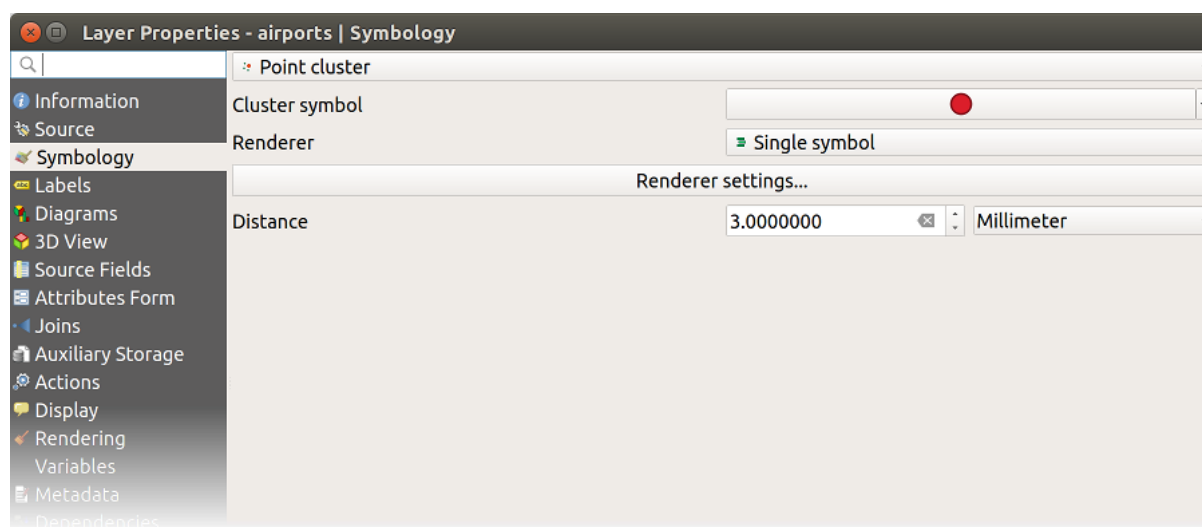



图 14.10: Point Cluster dialog

注解: Point Cluster renderer does not alter feature geometry, meaning that points are not moved from their position. They are still located at their initial place. Changes are only visual, for rendering purpose. Use instead the Processing *K-means clustering* or *DBSCAN clustering* algorithm if you want to create cluster-based features.

Inverted Polygon Renderer

The  *Inverted Polygon* renderer allows user to define a symbol to fill in outside of the layer's polygons. As above you can select subrenderers, namely Single symbol, Graduated, Categorized, Rule-Based or 2.5D renderer.

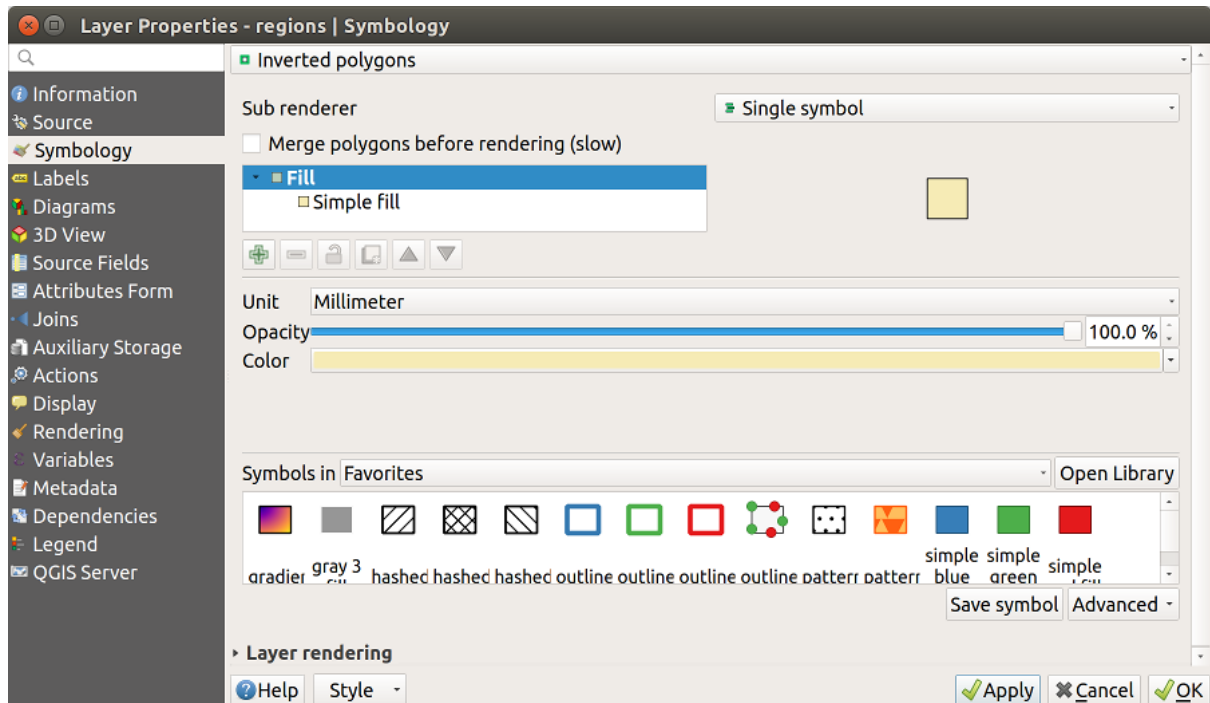



图 14.11: Inverted Polygon dialog

Heatmap Renderer

With the  *Heatmap* renderer you can create live dynamic heatmaps for (multi)point layers. You can specify the heatmap radius in millimeters, points, pixels, map units or inches, choose and edit a color ramp for the heatmap style and use a slider for selecting a trade-off between render speed and quality. You can also define a maximum value limit and give a weight to points using a field or an expression. When adding or removing a feature the heatmap renderer updates the heatmap style automatically.

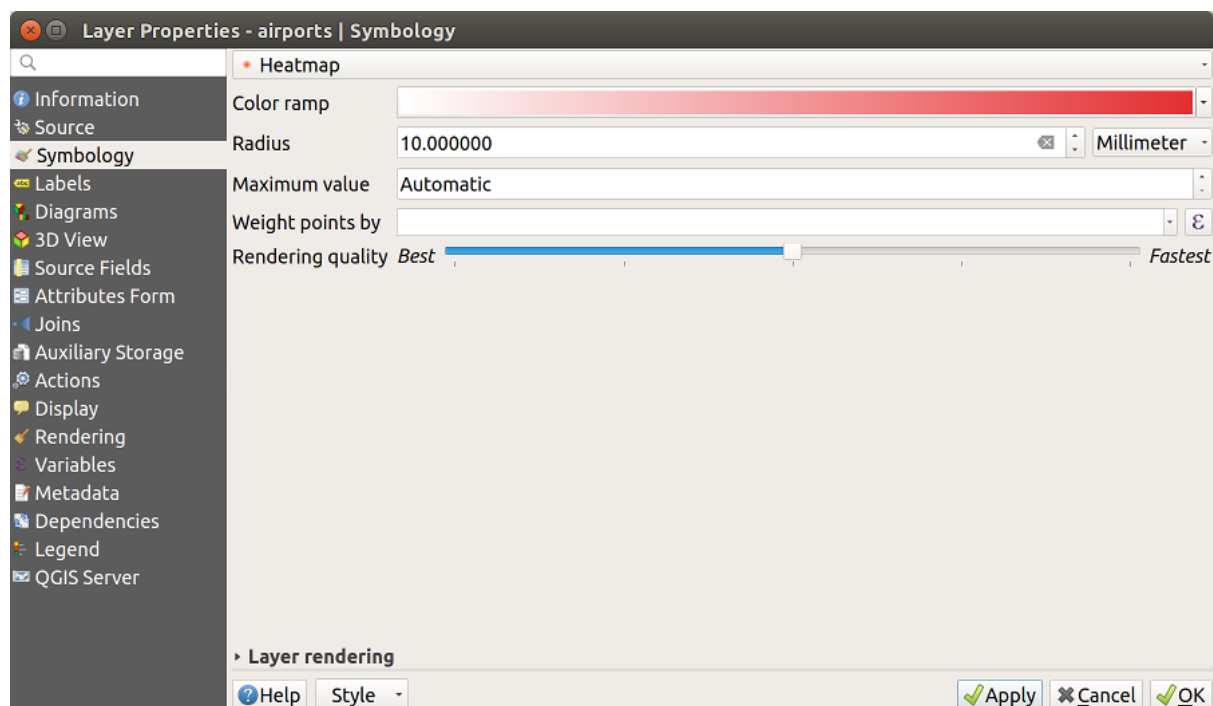



图 14.12: Heatmap dialog

2.5D Renderer

Using the  2.5D renderer it's possible to create a 2.5D effect on your layer's features. You start by choosing a *Height* value (in map units). For that you can use a fixed value, one of your layer's fields, or an expression. You also need to choose an *Angle* (in degrees) to recreate the viewer position (0° means west, growing in counter clock wise). Use advanced configuration options to set the *Roof Color* and *Wall Color*. If you would like to simulate solar radiation on the features walls, make sure to check the *Shade walls based on aspect* option. You can also simulate a shadow by setting a *Color* and *Size* (in map units).

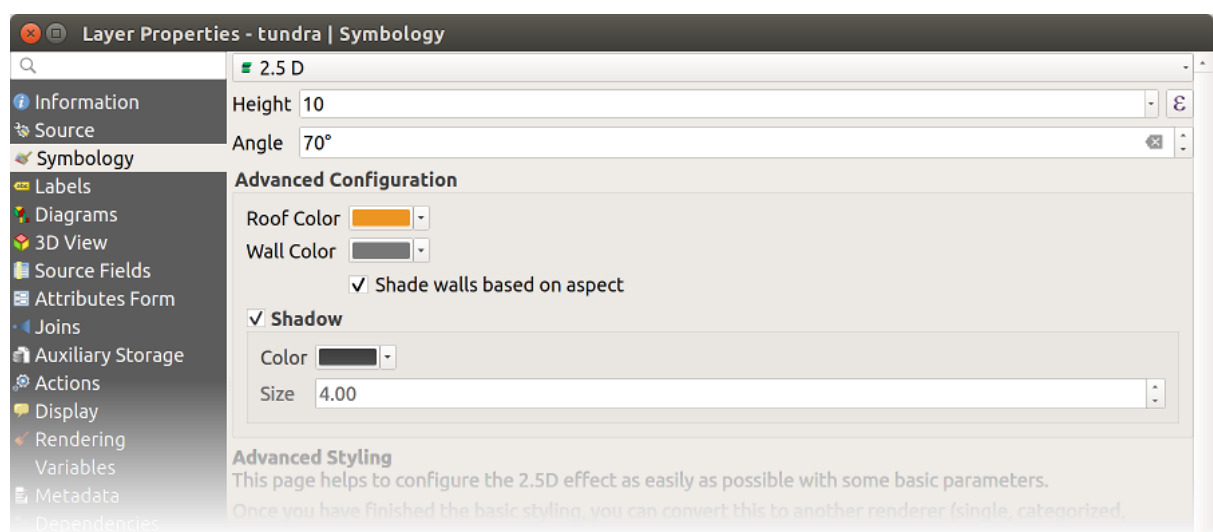




图 14.13: 2.5D dialog

小技巧: Using 2.5D effect with other renderers

Once you have finished setting the basic style on the 2.5D renderer, you can convert this to another renderer (single, categorized, graduated). The 2.5D effects will be kept and all other renderer specific options will be available for you to fine tune them (this way you can have for example categorized symbols with a nice 2.5D representation or add some extra styling to your 2.5D symbols). To make sure that the shadow and the "building" itself do not interfere with other nearby features, you may need to enable Symbols Levels (*Advanced* ► *Symbol levels...*). The 2.5D height and angle values are saved in the layer's variables, so you can edit it afterwards in the variables tab of the layer's properties dialog.

Layer rendering

From the Symbology tab, you can also set some options that invariably act on all features of the layer:

- *Opacity* : You can make the underlying layer in the map canvas visible with this tool. Use the slider to adapt the visibility of your vector layer to your needs. You can also make a precise definition of the percentage of visibility in the menu beside the slider.
- *Blending mode* at the *Layer* and *Feature* levels: You can achieve special rendering effects with these tools that you may previously only know from graphics programs. The pixels of your overlaying and underlying layers are mixed through the settings described in *Blending Modes*.
- Apply *paint effects* on all the layer features with the *Draw Effects* button.
- *Control feature rendering order* allows you, using features attributes, to define the z-order in which they shall be rendered. Activate the checkbox and click on the  button beside. You then get the *Define Order* dialog in which you:
 1. Choose a field or build an expression to apply to the layer features.
 2. Set in which order the fetched features should be sorted, i.e. if you choose **Ascending** order, the features with lower value are rendered under those with higher value.
 3. Define when features returning NULL value should be rendered: **first** (bottom) or **last** (top).
 4. Repeat the above steps as many times as rules you wish to use.

The first rule is applied to all the features in the layer, z-ordering them according to their returned value. Then, within each group of features with the same value (including those with NULL value) and thus the same z-level, the next rule is applied to sort them. And so on...

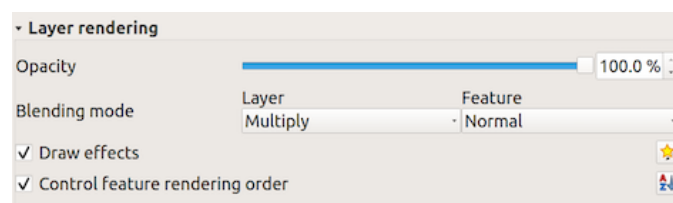


图 14.14: Layer rendering options

Other Settings

Symbol levels

For renderers that allow stacked symbol layers (only heatmap doesn't) there is an option to control the rendering order of each symbol's levels.

For most of the renderers, you can access the Symbols levels option by clicking the *Advanced* button below the saved symbols list and choosing *Symbol levels*. For the *Rule-based Renderer* the option is directly available through *Symbols Levels...* button, while for *Point displacement Renderer* renderer the same button is inside the *Rendering settings* dialog.

To activate symbols levels, select the *Enable symbol levels*. Each row will show up a small sample of the combined symbol, its label and the individual symbols layer divided into columns with a number next to it. The numbers represent the rendering order level in which the symbol layer will be drawn. Lower values levels are drawn first, staying at the bottom, while higher values are drawn last, on top of the others.

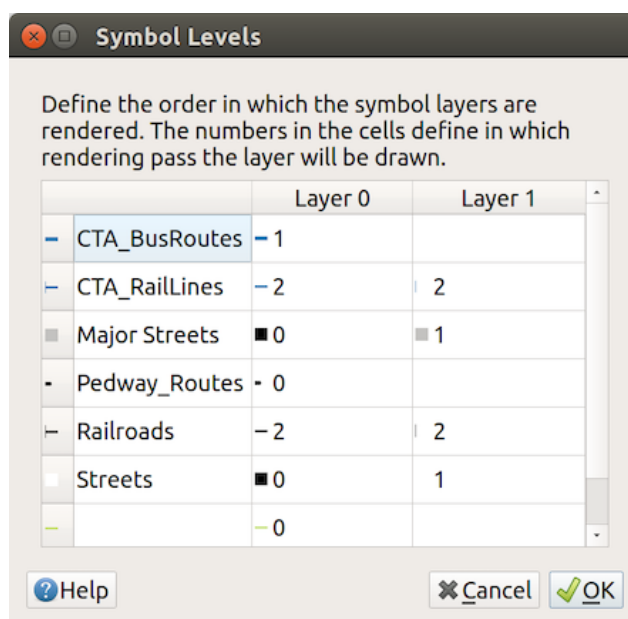


图 14.15: Symbol levels dialog

注解: If symbols levels are deactivated, the complete symbols will be drawn according to their respective features order. Overlapping symbols will simply obfuscate to other below. Besides, similar symbols won't "merge" with each other.

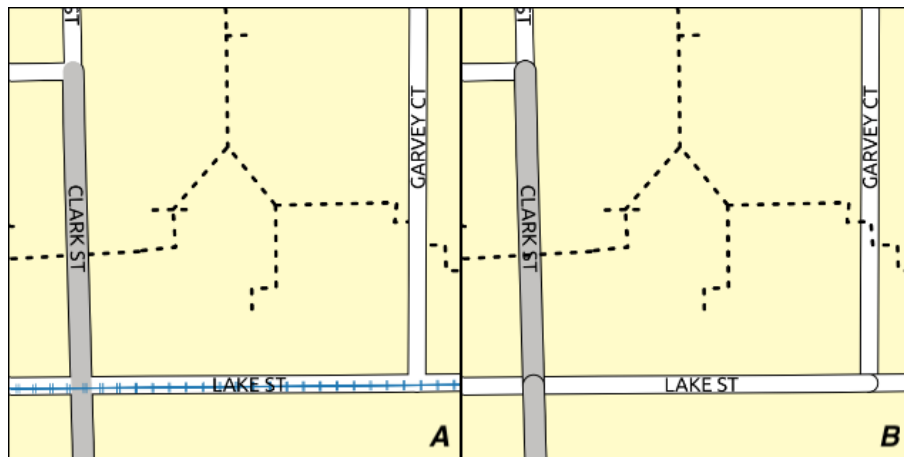




图 14.16: Symbol levels activated (A) and deactivated (B) difference

Data-defined size legend

When a layer is rendered with the *proportional symbol* or the *multivariate rendering* or when a *scaled size diagram* is applied to the layer, you can allow the display of the scaled symbols in both the *Layers panel* and the *print layout legend*.

To enable the *Data-defined Size Legend* dialog to render symbology, select the eponym option in the *Advanced* button below the saved symbols list. For diagrams, the option is available under the *Legend* tab. The dialog provides the following options to:

- select the type of legend: *Legend not enabled*, *Separated legend items* and *Collapsed legend*. For the latter option, you can select whether the legend items are aligned at the **Bottom** or at the **Center**;
- set the *symbol to use* for legend representation;
- insert the title in the legend;
- resize the classes to use: by default, QGIS provides you with a legend of five classes (based on natural pretty breaks) but you can apply your own classification using the *Manual size classes* option. Use the  and  buttons to set your custom classes values and labels.

A preview of the legend is displayed in the right panel of the dialog and updated as you set the parameters. For collapsed legend, a leader line from the horizontal center of the symbol to the corresponding legend text is drawn.

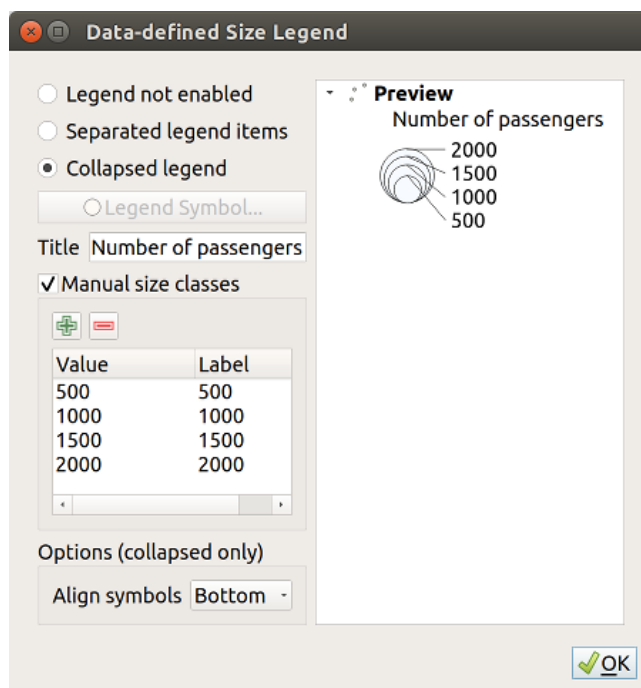





图 14.17: Setting size scaled legend

注解: Currently, data-defined size legend for layer symbology can only be applied to point layer using single, categorized or graduated symbology.

Draw effects

In order to improve layer rendering and avoid (or at least reduce) the resort to other software for final rendering of maps, QGIS provides another powerful functionality: the  *Draw Effects* options, which adds paint effects for customizing the visualization of vector layers.

The option is available in the *Layer Properties* ► *Symbology* dialog, under the *Layer rendering* group (applying to the whole layer) or in *symbol layer properties* (applying to corresponding features). You can combine both usage.

Paint effects can be activated by checking the  *Draw effects* option and clicking the  *Customize effects* button. That will open the *Effect Properties* Dialog (see 图 14.18). The following effect types, with custom options are available:

- **Source:** Draws the feature's original style according to the configuration of the layer's properties. The *Opacity* of its style can be adjusted as well as the *Blend mode* and *Draw mode*. These are common properties for all types of effects.

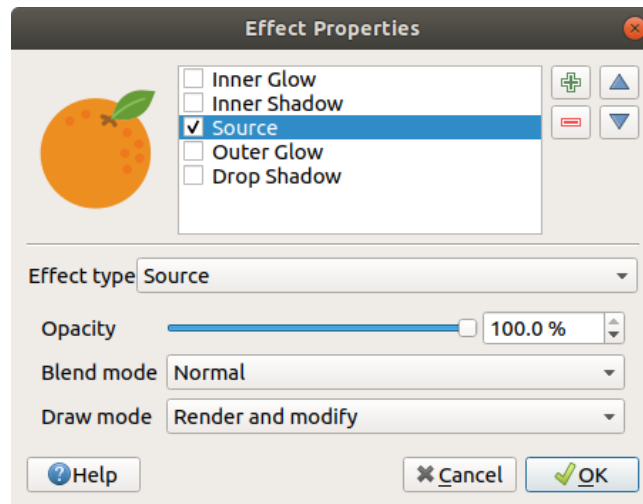


图 14.18: Draw Effects: Source dialog

- **Blur:** Adds a blur effect on the vector layer. The custom options that you can change are the *Blur type* (*Stack blur (fast)* or *Gaussian blur (quality)*) and the *Blur strength*.

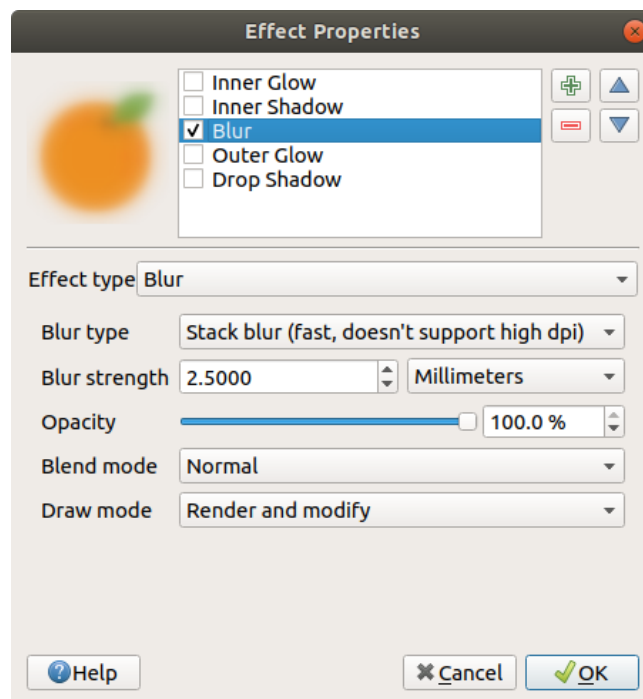




图 14.19: Draw Effects: Blur dialog

- **Colorise:** This effect can be used to make a version of the style using one single hue. The base will always be a grayscale version of the symbol and you can:
 - Use the  *Grayscale* to select how to create it: options are 'By lightness', 'By luminosity', 'By average' and 'Off'.
 - If  *Colorise* is selected, it will be possible to mix another color and choose how strong it should be.
 - Control the *Brightness*, *Contrast* and *Saturation* levels of the resulting symbol.

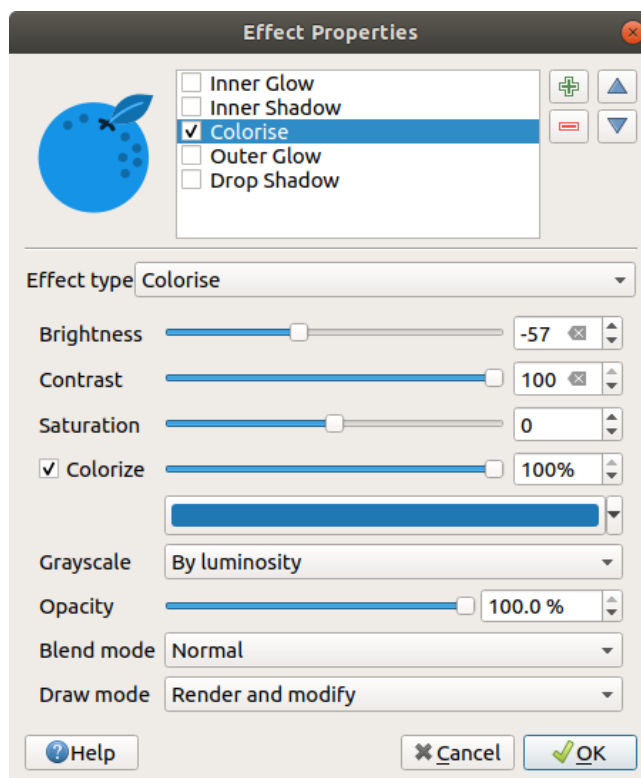


图 14.20: Draw Effects: Colorize dialog

- **Drop Shadow:** Using this effect adds a shadow on the feature, which looks like adding an extra dimension. This effect can be customized by changing the *Offset* angle and distance, determining where the shadow shifts towards to and the proximity to the source object. *Drop Shadow* also has the option to change the *Blur radius* and the *Color* of the effect.

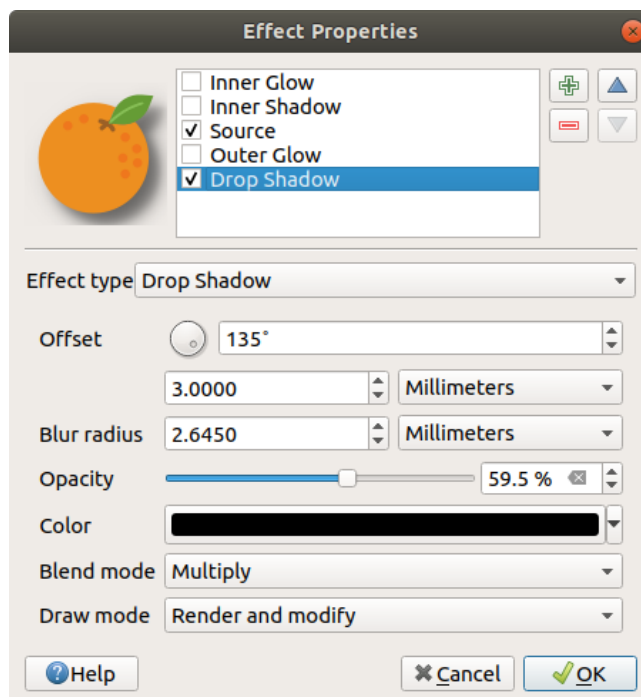


图 14.21: Draw Effects: Drop Shadow dialog

- **Inner Shadow:** This effect is similar to the *Drop Shadow* effect, but it adds the shadow effect on the inside of the edges of the feature. The available options for customization are the same as the *Drop Shadow* effect.

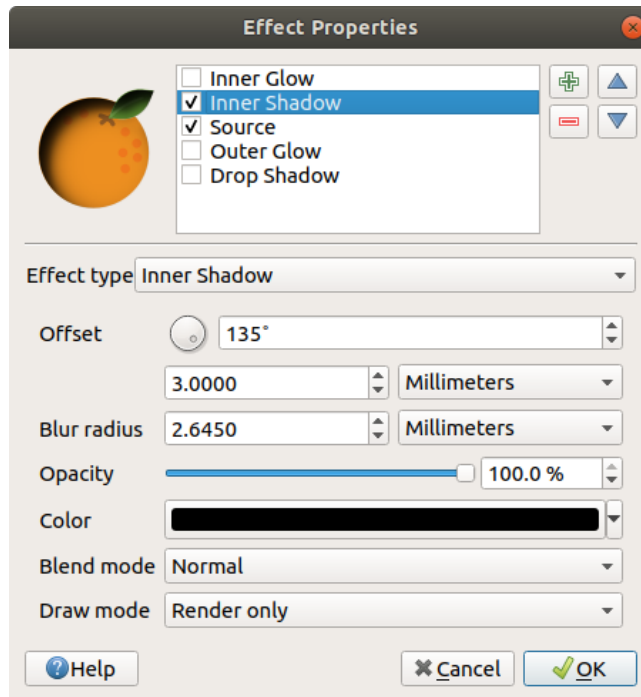


图 14.22: Draw Effects: Inner Shadow dialog

- **Inner Glow:** Adds a glow effect inside the feature. This effect can be customized by adjusting the *Spread* (width) of the glow, or the *Blur radius*. The latter specifies the proximity from the edge of the feature where you want any blurring to happen. Additionally, there are options to customize the color of the glow using a *Single color* or a *Color ramp*.

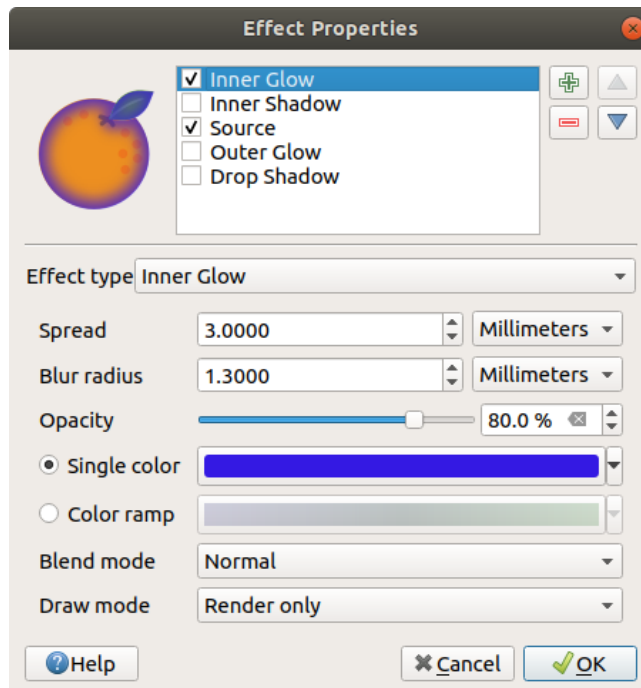


图 14.23: Draw Effects: Inner Glow dialog

- **Outer Glow:** This effect is similar to the *Inner Glow* effect, but it adds the glow effect on the outside of the edges of the feature. The available options for customization are the same as the *Inner Glow* effect.

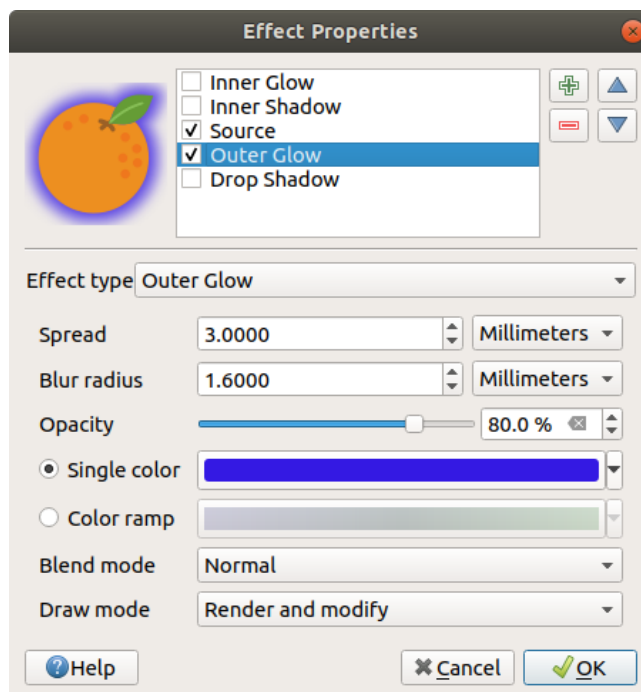


图 14.24: Draw Effects: Outer Glow dialog

- **Transform:** Adds the possibility of transforming the shape of the symbol. The first options available for customization are the *Reflect horizontal* and *Reflect vertical*, which actually create a reflection on the horizontal and/or vertical axes. The other options are:
 - *Shear X,Y:* Slants the feature along the X and/or Y axis.
 - *Scale X,Y:* Enlarges or minimizes the feature along the X and/or Y axis by the given percentage.
 - *Rotation:* Turns the feature around its center point.
 - and *Translate X,Y* changes the position of the item based on a distance given on the X and/or Y axis.

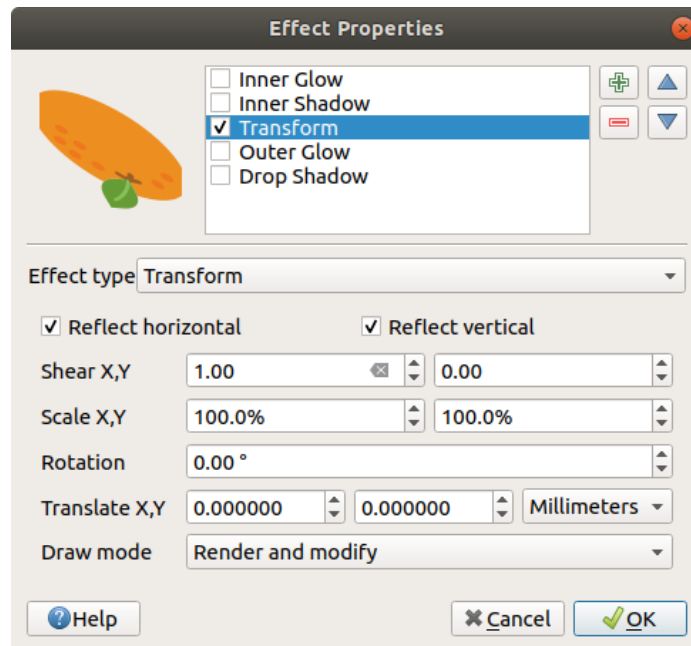


图 14.25: Draw Effects: Transform dialog

One or more effect types can be used at the same time. You (de)activate an effect using its checkbox in the effects list. You can change the selected effect type by using the *Effect type* option. You can reorder the effects using Move up and Move down buttons, and also add/remove effects using the Add new effect and Remove effect buttons.

There are some common options available for all draw effect types. *Opacity* and *Blend mode* options work similar to the ones described in *Layer rendering* and can be used in all draw effects except for the transform one.

There is also a *Draw mode* option available for every effect, and you can choose whether to render and/or modify the symbol, following some rules:


- Effects render from top to bottom.
- *Render only* mode means that the effect will be visible.
- *Modifier only* mode means that the effect will not be visible but the changes that it applies will be passed to the next effect (the one immediately below).
- The *Render and Modify* mode will make the effect visible and pass any changes to the next effect. If the effect is at the top of the effects list or if the immediately above effect is not in modify mode, then it will use the original source symbol from the layers properties (similar to source).

14.1.4 Labels Properties

The *Labels* properties provides you with all the needed and appropriate capabilities to configure smart labeling on vector layers. This dialog can also be accessed from the *Layer Styling* panel, or using the Layer Labeling Options button of the **Labels toolbar**.

The first step is to choose the labeling method from the drop-down list. Available methods are:

- *No labels*: the default value, showing no labels from the layer
- *Single labels*: Show labels on the map using a single attribute or an expression
- *Rule-based labeling*

- and  *Blocking*: allows to set a layer as just an obstacle for other layer's labels without rendering any labels of its own.

The next steps assume you select the  *Single labels* option, opening the following dialog.

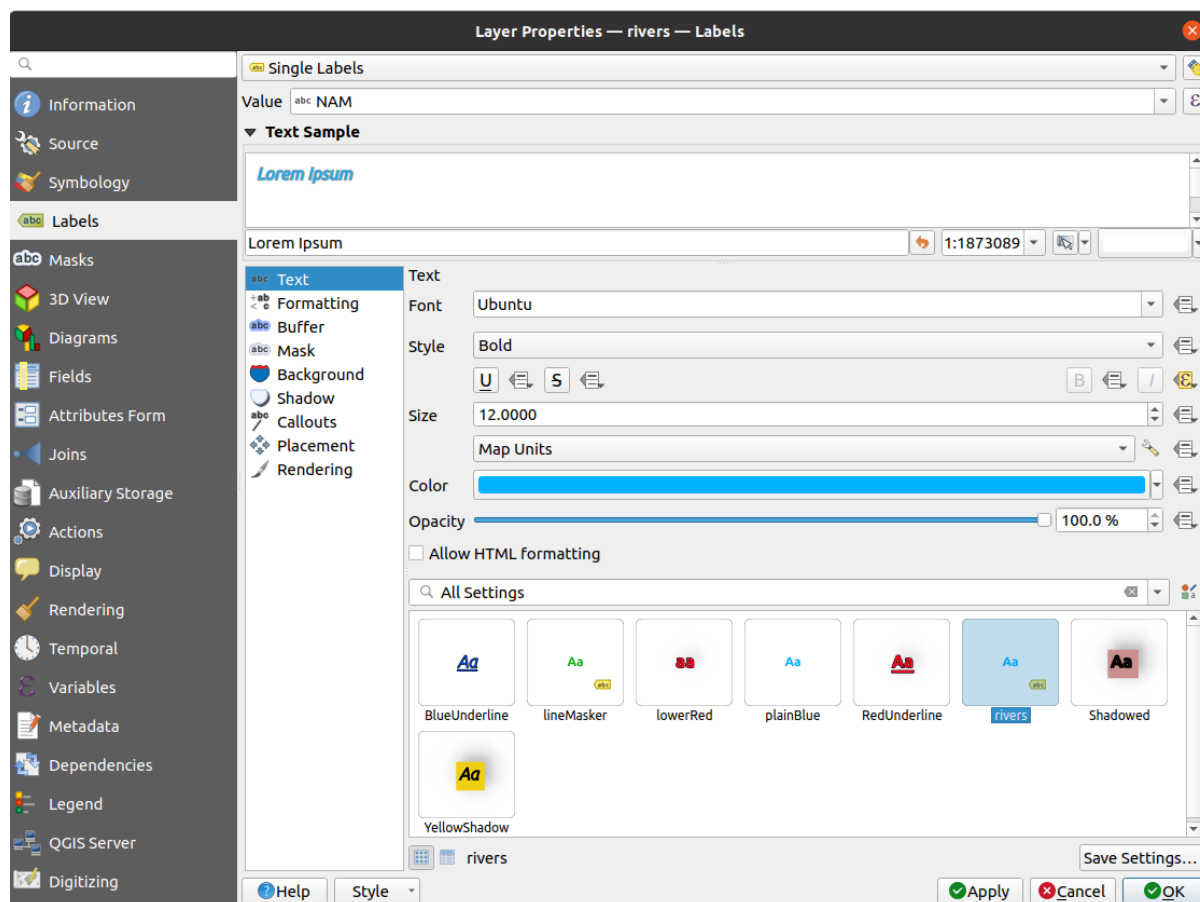








图 14.26: Layer labeling settings - Single labels


At the top of the dialog, a *Value* drop-down list is enabled. You can select an attribute column to use for labeling. By default, the *display field* is used. Click  if you want to define labels based on expressions - See *Define labels based on expressions*.

Below are displayed options to customize the labels, under various tabs:

-  *Text*
-  *Formatting*
-  *Buffer*
-  *Mask*
-  *Background*
-  *Shadow*
-  *Callouts*
-  *Placement*
-  *Rendering*

Description of how to set each property is exposed at *Setting a label*.

Setting the automated placement engine

You can use the automated placement settings to configure a project-level automated behavior of the labels. In the top right corner of the *Labels* tab, click the  Automated placement settings (applies to all layers) button, opening a dialog with the following options:

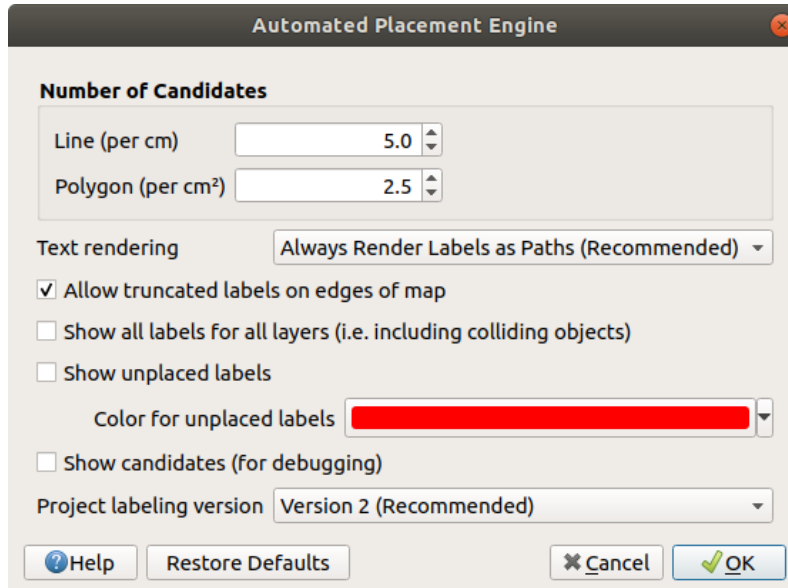


图 14.27: The labels automated placement engine



- *Number of candidates*: calculates and assigns to line and polygon features the number of possible labels placement based on their size. The longer or wider a feature is, the more candidates it has, and its labels can be better placed with less risk of collision.
- *Text rendering*: sets the default value for label rendering widgets when *exporting a map canvas* or *a layout* to PDF or SVG. If *Always render labels as text* is selected then labels can be edited in external applications (e.g. Inkscape) as normal text. BUT the side effect is that the rendering quality is decreased, and there are issues with rendering when certain text settings like buffers are in place. That's why *Always render labels as paths (recommended)* which exports labels as outlines, is recommended.
- *Allow truncated labels on edges of map*: controls whether labels which fall partially outside of the map extent should be rendered. If checked, these labels will be shown (when there's no way to place them fully within the visible area). If unchecked then partially visible labels will be skipped. Note that this setting has no effects on labels' display in the *layout map item*.
- *Show all labels for all layers (i.e. including colliding objects)*. Note that this option can be also set per layer (see *Rendering tab*)
- *Show unplaced labels*: allows to determine whether any important labels are missing from the maps (e.g. due to overlaps or other constraints). They are displayed using a customizable color.
- *Show candidates (for debugging)*: controls whether boxes should be drawn on the map showing all the candidates generated for label placement. Like the label says, it's useful only for debugging and testing the effect different labeling settings have. This could be handy for a better manual placement with tools from the *label toolbar*.
- *Project labeling version*: QGIS supports two different versions of label automatic placement:
 - *Version 1*: the old system (used by QGIS versions 3.10 and earlier, and when opening projects created in these versions in QGIS 3.12 or later). Version 1 treats label and obstacle priorities as "rough guides" only, and it's possible that a low-priority label will be placed over a high-priority obstacle in this version.

Accordingly, it can be difficult to obtain the desired labeling results when using this version and it is thus recommended only for compatibility with older projects.

- *Version 2 (recommended)*: this is the default system in new projects created in QGIS 3.12 or later. In version 2, the logic dictating when labels are allowed to overlap *obstacles* has been reworked. The newer logic forbids any labels from overlapping any obstacles with a greater obstacle weight compared to the label's priority. As a result, this version results in much more predictable and easier to understand labeling results.

Rule-based labeling

With rule-based labeling multiple label configurations can be defined and applied selectively on the base of expression filters and scale range, as in *Rule-based rendering*.

To create a rule, select the  **Rule-based labeling** option in the main drop-down list from the *Labels* tab and click the  button at the bottom of the dialog. Then fill the new dialog with a description and an expression to filter features. You can also set a *scale range* in which the label rule should be applied. The other options available in this dialog are the *common settings* seen beforehand.

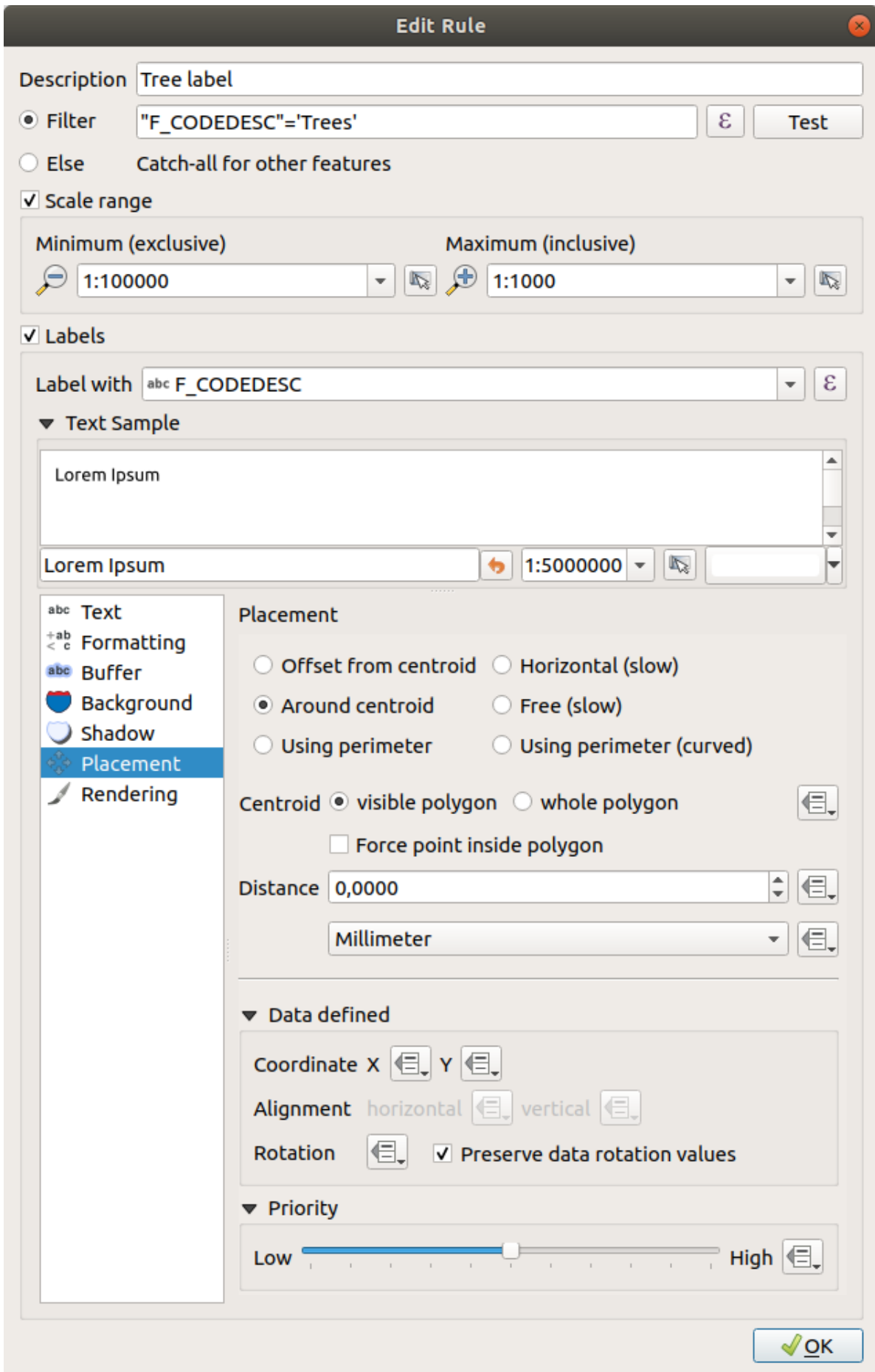




图 14.28: Rule settings

A summary of existing rules is shown in the main dialog (see 图 14.29). You can add multiple rules, reorder or imbricate them with a drag-and-drop. You can as well remove them with the  button or edit them with  button or a double-click.

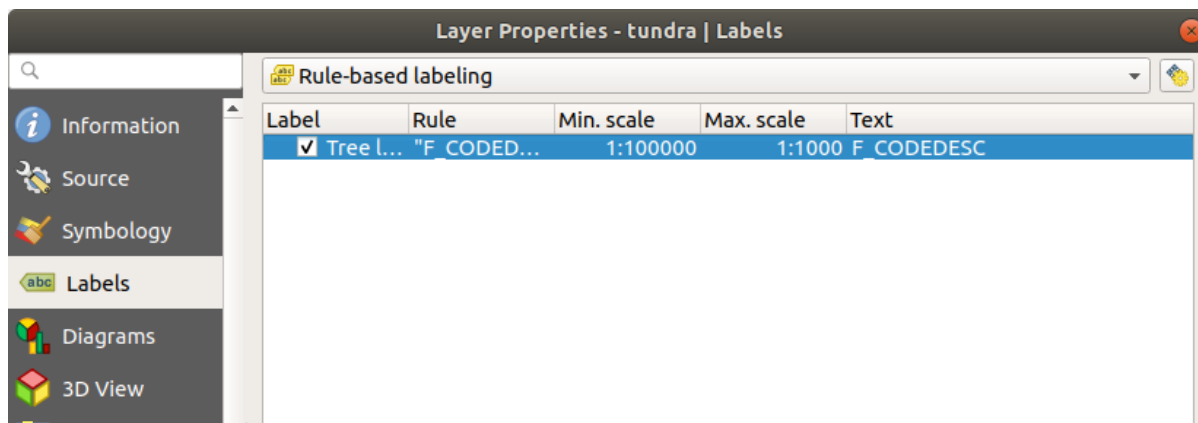




图 14.29: Rule based labeling panel

Define labels based on expressions

Whether you choose single or rule-based labeling type, QGIS allows using expressions to label features.

Assuming you are using the *Single labels* method, click the  button near the *Value* drop-down list in the  *Labels* tab of the properties dialog.

In 图 14.30, you see a sample expression to label the alaska trees layer with tree type and area, based on the field 'VEGDESC', some descriptive text, and the function \$area in combination with `format_number()` to make it look nicer.

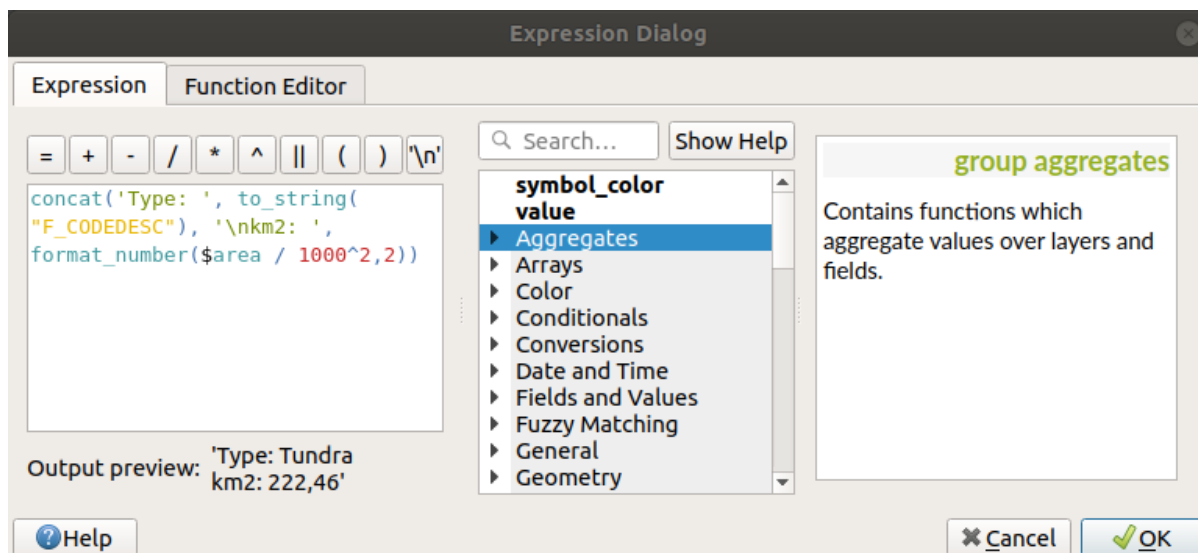


图 14.30: Using expressions for labeling

Expression based labeling is easy to work with. All you have to take care of is that:

- You may need to combine all elements (strings, fields, and functions) with a string concatenation function such as `concat`, `+` or `||`. Be aware that in some situations (when null or numeric value are involved) not all of these tools will fit your need.

- Strings are written in 'single quotes'.
- Fields are written in "double quotes" or without any quote.

Let's have a look at some examples:

1. Label based on two fields 'name' and 'place' with a comma as separator:

```
"name" || ', ' || "place"
```

Returns:

```
John Smith, Paris
```

2. Label based on two fields 'name' and 'place' with other texts:

```
'My name is ' + "name" + 'and I live in ' + "place"
'My name is ' || "name" || 'and I live in ' || "place"
concat('My name is ', name, ' and I live in ', "place")
```

Returns:

```
My name is John Smith and I live in Paris
```

3. Label based on two fields 'name' and 'place' with other texts combining different concatenation functions:

```
concat('My name is ', name, ' and I live in ' || place)
```

Returns:

```
My name is John Smith and I live in Paris
```

Or, if the field 'place' is NULL, returns:

```
My name is John Smith
```

4. Multi-line label based on two fields 'name' and 'place' with a descriptive text:

```
concat('My name is ', "name", '\n', 'I live in ', "place")
```

Returns:

```
My name is John Smith
I live in Paris
```

5. Label based on a field and the \$area function to show the place's name and its rounded area size in a converted unit:

```
'The area of ' || "place" || ' has a size of '
|| round($area/10000) || ' ha'
```

Returns:

```
The area of Paris has a size of 10500 ha
```

6. Create a CASE ELSE condition. If the population value in field *population* is <= 50000 it is a town, otherwise it is a city:

```
concat('This place is a ',
CASE WHEN "population" <= 50000 THEN 'town' ELSE 'city' END)
```

Returns:

```
This place is a town
```

7. Display name for the cities and no label for the other features (for the "city" context, see example above):


```
CASE WHEN "population" > 50000 THEN "NAME" END
```

Returns:

```
Paris
```

As you can see in the expression builder, you have hundreds of functions available to create simple and very complex expressions to label your data in QGIS. See [Expressions](#) chapter for more information and examples on expressions.


Using data-defined override for labeling

With the  Data defined override function, the settings for the labeling are overridden by entries in the attribute table or expressions based on them. This feature can be used to set values for most of the labeling options described above.



For example, using the Alaska QGIS sample dataset, let's label the `airports` layer with their name, based on their military USE, i.e. whether the airport is accessible to :

- military people, then display it in gray color, size 8;
- others, then show in blue color, size 10.


To do this, after you enabled the labeling on the NAME field of the layer (see [Setting a label](#)):

1. Activate the *Text* tab.
2. Click on the  icon next to the *Size* property.
3. Select *Edit...* and type:

```
CASE
  WHEN "USE" like '%Military%' THEN 8 -- because compatible values are
  ↳ 'Military'                               -- and 'Joint Military/Civilian'
  ELSE 10
END
```

4. Press *OK* to validate. The dialog closes and the  button becomes  meaning that an rule is being run.
5. Then click the button next to the color property, type the expression below and validate:

```
CASE
  WHEN "USE" like '%Military%' THEN '150, 150, 150'
  ELSE '0, 0, 255'
END
```

Likewise, you can customize any other property of the label, the way you want. See more details on the  Data-define override widget's description and manipulation in [Data defined override setup](#) section.

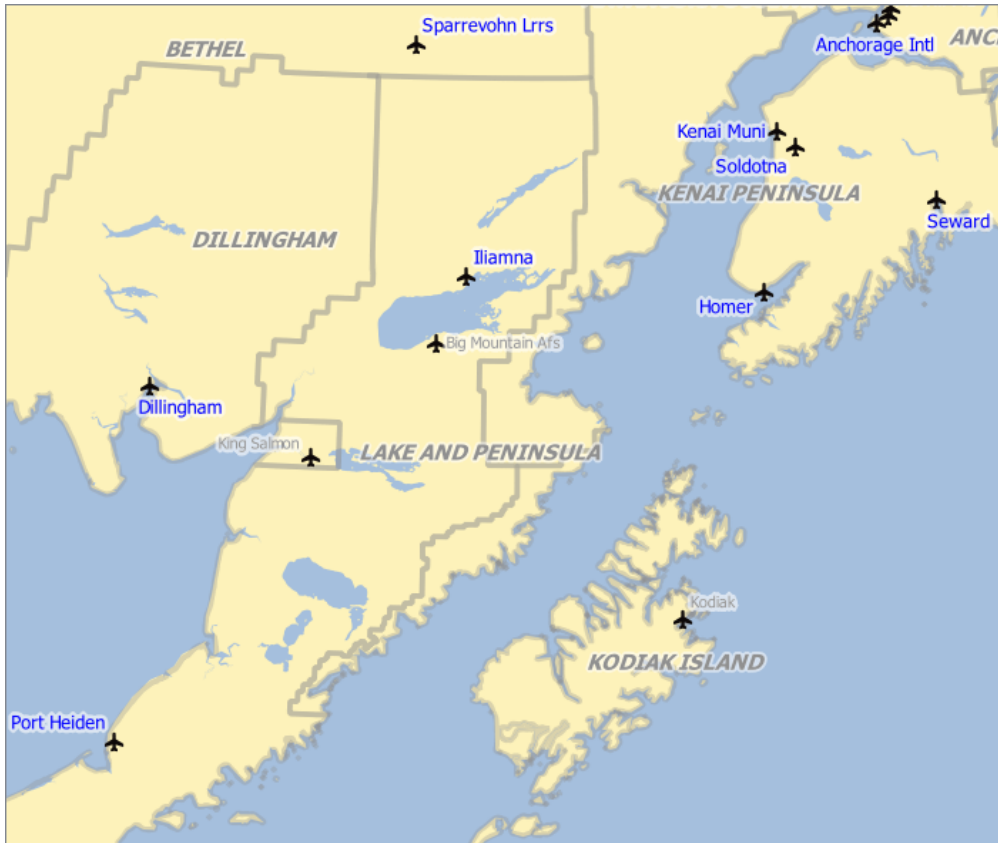


图 14.31: Airports labels are formatted based on their attributes

小技巧: Use the data-defined override to label every part of multi-part features

There is an option to set the labeling for multi-part features independently from your label properties. Choose the *Rendering*, *Feature options*, go to the *Data-define override* button next to the checkbox *Label every part of multipart-features* and define the labels as described in *Data defined override setup*.

The Label Toolbar










The *Label Toolbar* provides some tools to manipulate  *label* or  *diagram* properties.



图 14.32: The Label toolbar

While for readability, *label* has been used below to describe the Label toolbar, note that when mentioned in their name, the tools work almost the same way with diagrams:

-  **Highlight Pinned Labels and Diagrams**: If the vector layer of the label is editable, then the highlighting is green, otherwise it's blue.
-  **Toggles Display of Unplaced Labels**: Allows to determine whether any important labels are missing from the maps (e.g. due to overlaps or other constraints). They are displayed with a customizable color (see *Setting the automated placement engine*).


-  Pin/Unpin Labels and Diagrams . By clicking or dragging an area, you pin label(s). If you click or drag an area holding `Shift`, label(s) are unpinned. Finally, you can also click or drag an area holding `Ctrl` to toggle the pin status of label(s).
-  Show/Hide Labels and Diagrams . If you click on the labels, or click and drag an area holding `Shift`, they are hidden. When a label is hidden, you just have to click on the feature to restore its visibility. If you drag an area, all the labels in the area will be restored.
-  Moves a Label or Diagram . You just have to drag the label to the desired place.
-  Rotates a Label . Click the label and move around and you get the text rotated.
-  Change Label Properties . It opens a dialog to change the clicked label properties; it can be the label itself, its coordinates, angle, font, size, multiline alignment ... as long as this property has been mapped to a field. Here you can set the option to *Label every part of a feature*.



警告: Label tools overwrite current field values

Using the *Label toolbar* to customize the labeling actually writes the new value of the property in the mapped field. Hence, be careful to not inadvertently replace data you may need later!

注解: The *Auxiliary Storage Properties* mechanism may be used to customize labeling (position, and so on) without modifying the underlying data source.

Customize the labels from the map canvas

Combined with the *Label Toolbar*, the data defined override setting helps you manipulate labels in the map canvas (move, edit, rotate). We now describe an example using the data-defined override function for the  Move label function (see 图 14.33).

1. Import `lakes.shp` from the QGIS sample dataset.
2. Double-click the layer to open the Layer Properties. Click on *Labels and Placement*. Select  *Offset from centroid*.
3. Look for the *Data defined* entries. Click the  icon to define the field type for the *Coordinate*. Choose `xlabel` for X and `ylabel` for Y. The icons are now highlighted in yellow.

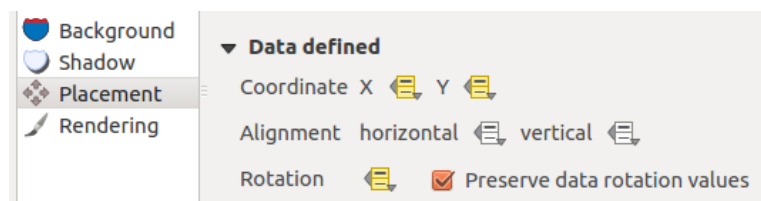




图 14.33: Labeling of vector polygon layers with data-defined override

4. Zoom into a lake.
5. Set editable the layer using the  Toggle Editing button.
6. Go to the Label toolbar and click the  icon. Now you can shift the label manually to another position (see 图 14.34). The new position of the label is saved in the `xlabel` and `ylabel` columns of the attribute table.

7. It's also possible to add a line connecting each lake to its moved label using:

- the label's *callout property*
- or the *geometry generator symbol layer* with the expression below:

```
make_line( centroid( $geometry ), make_point( "xlabel", "ylabel" ) )
```

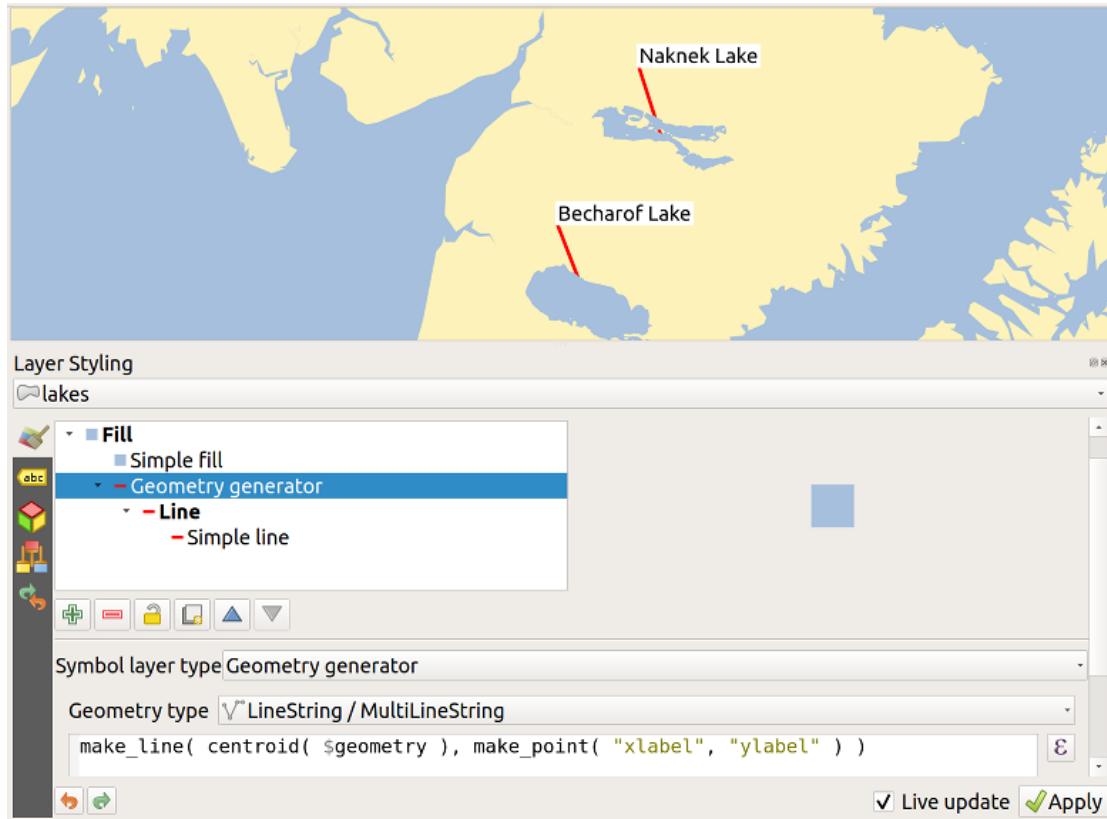


图 14.34: Moved labels






注解: The *Auxiliary Storage Properties* mechanism may be used with data-defined properties without having an editable data source.


14.1.5 Diagrams Properties



The *Diagrams* tab allows you to add a graphic overlay to a vector layer (see 图 14.35).

The current core implementation of diagrams provides support for:

-  *No diagrams*: the default value with no diagram displayed over the features;
-  *Pie chart*, a circular statistical graphic divided into slices to illustrate numerical proportion. The arc length of each slice is proportional to the quantity it represents;
-  *Text diagram*, a horizontally divided circle showing statistics values inside;
-  *Histogram*, bars of varying colors for each attribute aligned next to each other
-  *Stacked bars*, Stacks bars of varying colors for each attribute on top of each other vertically or horizontally

In the top right corner of the *Diagrams* tab, the  Automated placement settings (applies to all layers) button provides means to control diagram *labels placement* on the map canvas.


小技巧: Switch quickly between types of diagrams

Given that the settings are almost common to the different types of diagram, when designing your diagram, you can easily change the diagram type and check which one is more appropriate to your data without any loss.

For each type of diagram, the properties are divided into several tabs:

- *Attributes*
- *Rendering*
- *Size*
- *Placement*
- *Options*
- *Legend*

Attributes

Attributes defines which variables to display in the diagram. Use  add item button to select the desired fields into the 'Assigned Attributes' panel. Generated attributes with *Expressions* can also be used.

You can move up and down any row with click and drag, sorting how attributes are displayed. You can also change the label in the 'Legend' column or the attribute color by double-clicking the item.

This label is the default text displayed in the legend of the print layout or of the layer tree.

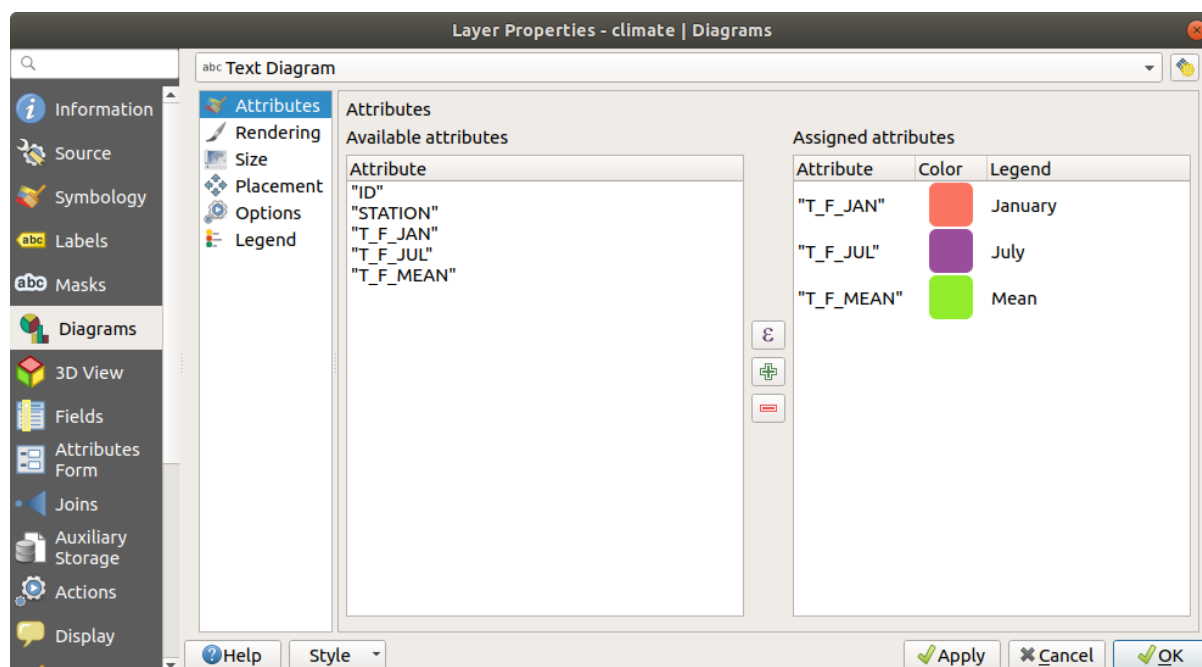


图 14.35: Diagram properties - Attributes tab

Rendering

Rendering defines how the diagram looks like. It provides general settings that do not interfere with the statistic values such as:

- the graphic's opacity, its outline width and color;
- depending on the type of diagram:
 - for histogram and stacked bars, the width of the bar and the spacing between the bars. You may want to set the spacing to 0 for stacked bars. Moreover, the *Axis line symbol* can be made visible on the map canvas and customized using *line symbol properties*.
 - for text diagram, the circle background color and the *font* used for texts;
 - for pie charts, the *Start angle* of the first slice and their *Direction* (clockwise or not).
- the use of *paint effects* on the graphics.

In this tab, you can also manage and fine tune the diagram visibility with different options:

- *Diagram z-index*: controls how diagrams are drawn on top of each other and on top of labels. A diagram with a high index is drawn over diagrams and labels;
- *Show all diagrams*: shows all the diagrams even if they overlap each other;
- *Show diagram*: allows only specific diagrams to be rendered;
- *Always Show*: selects specific diagrams to always render, even when they overlap other diagrams or map labels;
- setting the *Scale dependent visibility*;

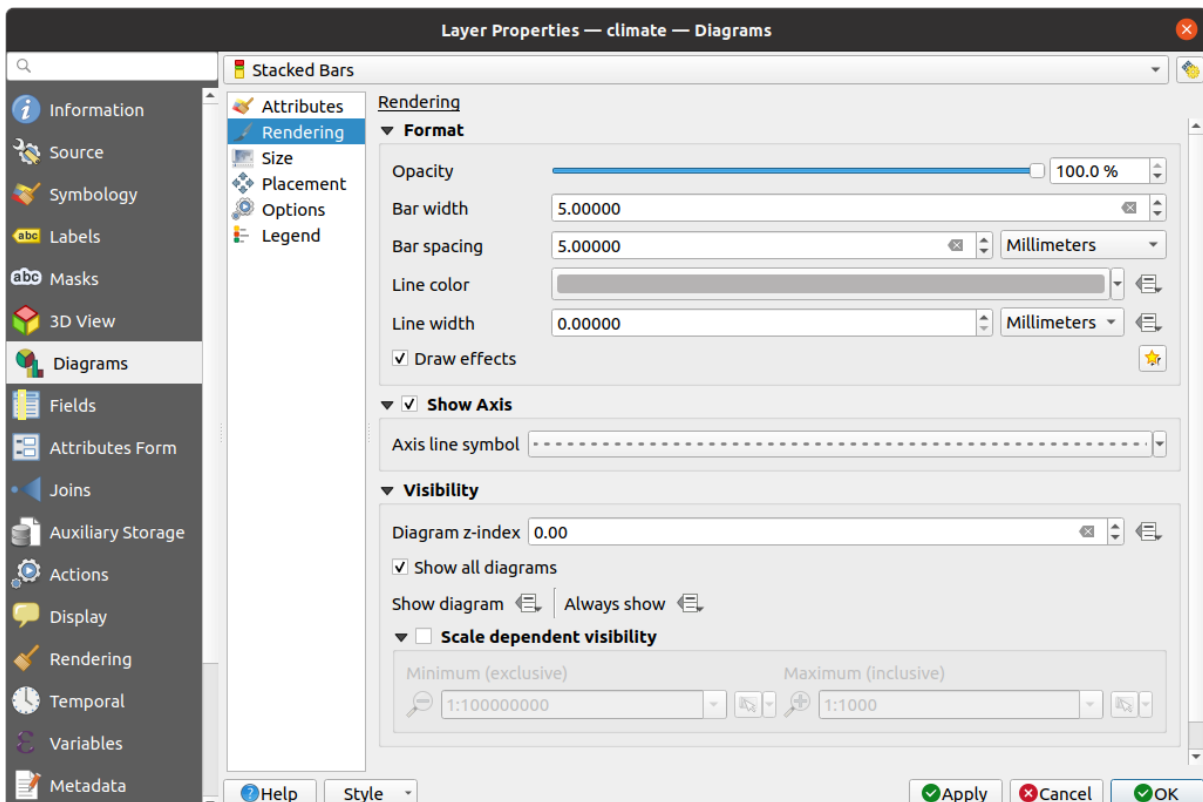


图 14.36: Diagram properties - Rendering tab

Size

Size is the main tab to set how the selected statistics are represented. The diagram size *units* can be 'Millimeters', 'Points', 'Pixels', 'Map Units' or 'Inches'. You can use:

- *Fixed size*, a unique size to represent the graphic of all the features (not available for histograms)
- or *Scaled size*, based on an expression using layer attributes:
 1. In *Attribute*, select a field or build an expression
 2. Press *Find* to return the *Maximum value* of the attribute or enter a custom value in the widget.
 3. For histogram and stacked bars, enter a *Bar length* value, used to represent the *Maximum value* of the attributes. For each feature, the bar length will then be scaled linearly to keep this matching.
 4. For pie chart and text diagram, enter a *Size* value, used to represent the *Maximum value* of the attributes. For each feature, the circle area or diameter will then be scaled linearly to keep this matching (from 0). A *Minimum size* can however be set for small diagrams.

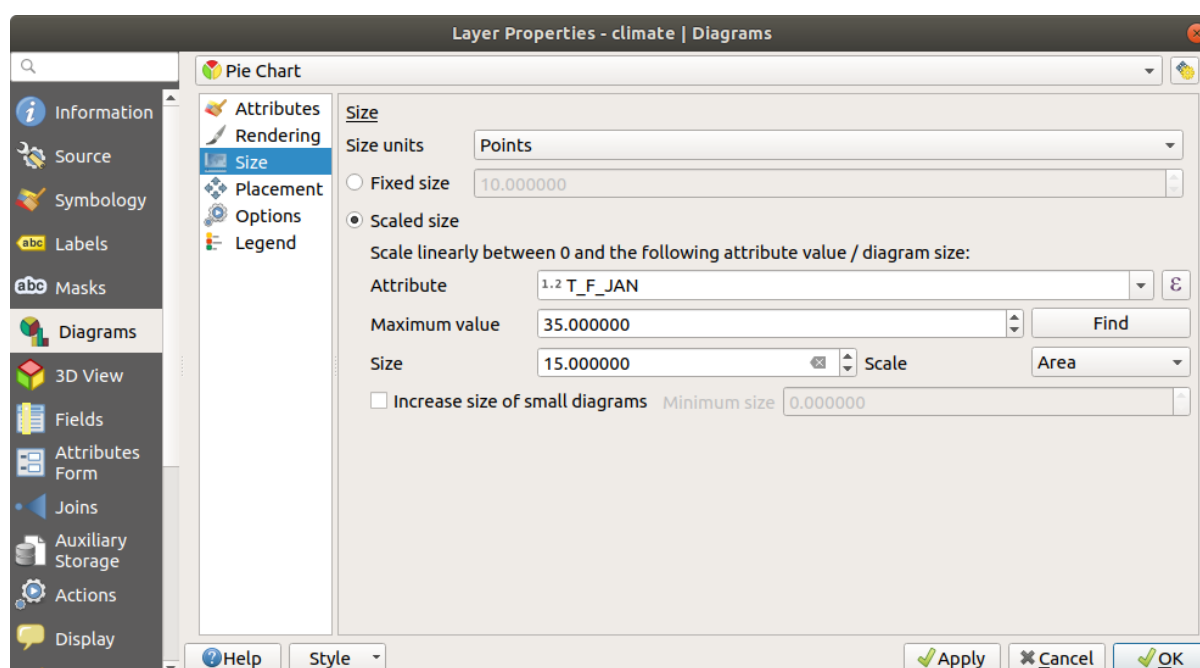


图 14.37: Diagram properties - Size tab

Placement

Placement defines the diagram position. Depending on the layer geometry type, it offers different options for the placement (more details at [Placement](#)):

- *Around point* or *Over point* for point geometry. The former variable requires a radius to follow.
- *Around line* or *Over line* for line geometry. Like point feature, the first variable requires a distance to respect and you can specify the diagram placement relative to the feature ('above', 'on' and/or 'below' the line) It's possible to select several options at once. In that case, QGIS will look for the optimal position of the diagram. Remember that you can also use the line orientation for the position of the diagram.
- *Around centroid* (at a set *Distance*), *Over centroid*, *Using perimeter* and *Inside polygon* are the options for polygon features.

The *Coordinate* group provides direct control on diagram placement, on a feature-by-feature basis, using their attributes or an expression to set the *X* and *Y* coordinate. The information can also be filled using the [Move labels and diagrams](#) tool.

In the *Priority* section, you can define the placement priority rank of each diagram, ie if there are different diagrams or labels candidates for the same location, the item with the higher priority will be displayed and the others could be left out.

Discourage diagrams and labels from covering features defines features to use as *obstacles*, ie QGIS will try to not place diagrams nor labels over these features. The priority rank is then used to evaluate whether a diagram could be omitted due to a greater weighted obstacle feature.

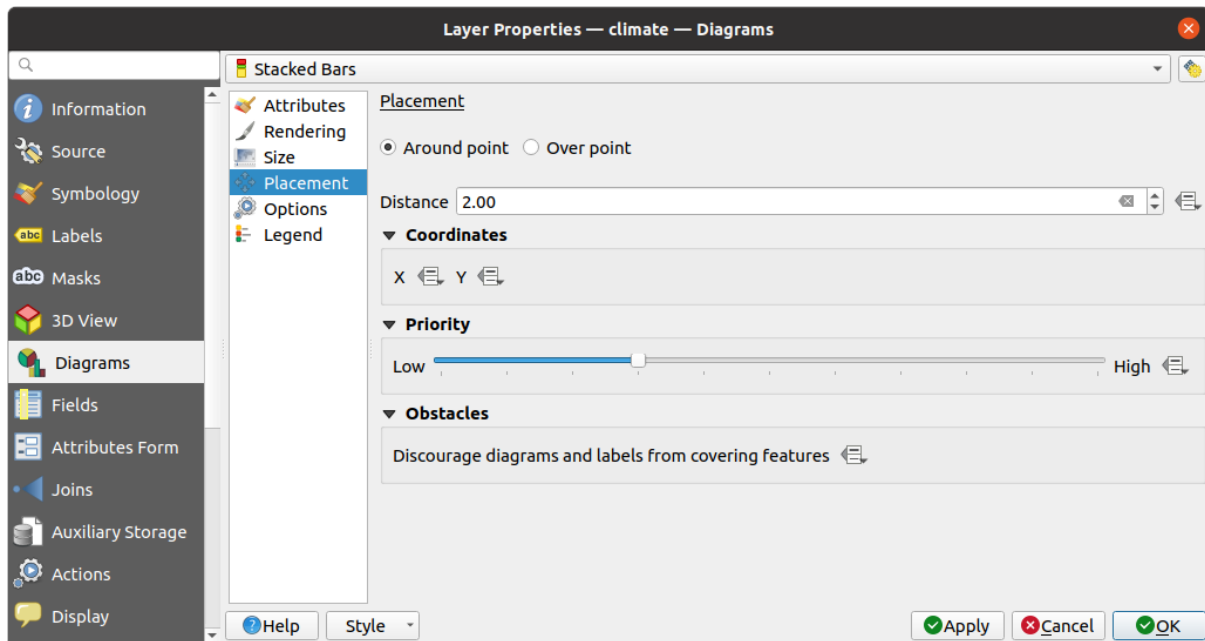


图 14.38: Vector properties dialog with diagram properties, Placement tab

Options

The *Options* tab has settings for histograms and stacked bars. You can choose whether the *Bar orientation* should be *Up*, *Down*, *Right* or *Left*, for horizontal and vertical diagrams.


Legend

From the *Legend* tab, you can choose to display items of the diagram in the *Layers panel*, and in the *print layout legend*, next to the layer symbology:

- check *Show legend entries for diagram attributes* to display in the legends the *Color* and *Legend* properties, as previously assigned in the *Attributes* tab;
- and, when a *scaled size* is being used for the diagrams, push the *Legend Entries for Diagram Size...* button to configure the diagram symbol aspect in the legends. This opens the *Data-defined Size Legend* dialog whose options are described in *Data-defined size legend*.

When set, the diagram legend items (attributes with color and diagram size) are also displayed in the print layout legend, next to the layer symbology.

14.1.6 Masks Properties

 The *Masks* tab helps you configure the current layer symbols overlay with other symbol layers or labels, from any layer. This is meant to improve the readability of symbols and labels whose colors are close and can be hard to decipher when overlapping; it adds a custom and transparent mask around the items to "hide" parts of the symbol layers of the current layer.

To apply masks on the active layer, you first need to enable in the project either *mask symbol layers* or *mask labels*. Then, from the *Masks* tab, check:

- the *Masked symbol layers*: lists in a tree structure all the symbol layers of the current layer. There you can select the symbol layer item you would like to transparently "cut out" when they overlap the selected mask sources
- the *Mask sources* tab: list all the mask labels and mask symbol layers defined in the project. Select the items that would generate the mask over the selected masked symbol layers

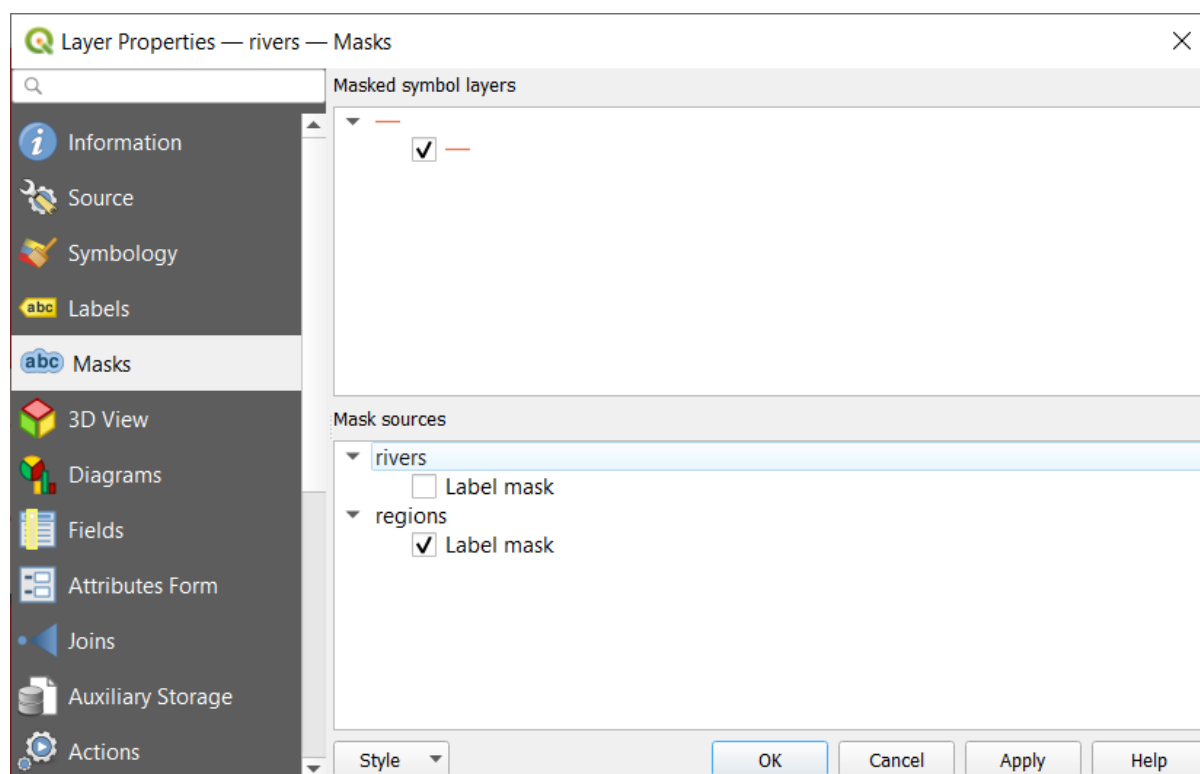



图 14.39: Layer properties - Masks tab

14.1.7 3D View Properties

 The *3D View* tab provides settings for vector layers that should be depicted in the *3D Map view* tool.

For better performance, data from vector layers are loaded in the background, using multithreading, and rendered in tiles whose size can be controlled from the *Layer rendering* section of the tab:

- *Zoom levels count*: determines how deep the quadtree will be. For example, one zoom level means there will be a single tile for the whole layer. Three zoom levels means there will be 16 tiles at the leaf level (every extra zoom level multiplies that by 4). The default is 3 and the maximum is 8.
-  *Show bounding boxes of tiles*: especially useful if there are issues with tiles not showing up when they should

To display a layer in 3D, select from the combobox at the top of the tab, either:

- *Single symbol*: features are rendered using a common 3D symbol whose properties can be *data-defined* or not. Read details on *setting a 3D symbol* for each layer geometry type.
- *Rule-based*: multiple symbol configurations can be defined and applied selectively based on expression filters and scale range. More details on how-to at *Rule-based rendering*.

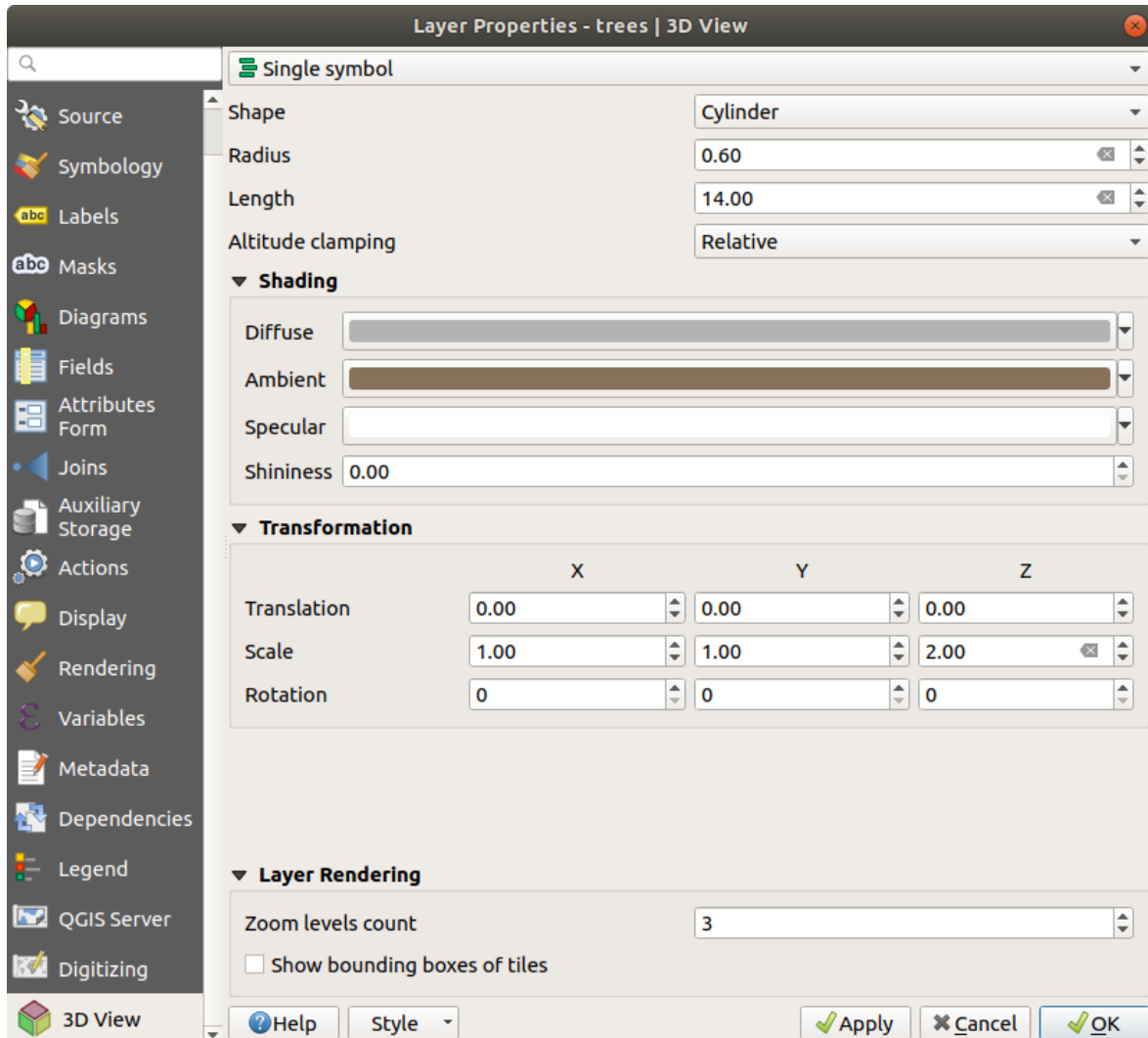





图 14.40: 3D properties of a point layer

14.1.8 Fields Properties

The *Fields* tab provides information on fields related to the layer and helps you organize them.

The layer can be made *editable* using the  *Toggle editing mode*. At this moment, you can modify its structure using the  *New field* and  *Delete field* buttons.

You can also rename fields by double-clicking its name. This is only supported for data providers like PostgreSQL, Oracle, Memory layer and some OGR layer depending on the OGR data format and version.

If set in the underlying data source or in the *forms properties*, the field's alias is also displayed. An alias is a human readable field name you can use in the feature form or the attribute table. Aliases are saved in the project file.

Depending on the data provider, you can associate a comment with a field, for example at its creation. This information

is retrieved and shown in the *Comment* column and is later displayed when hovering over the field label in a feature form.

Other than the fields contained in the dataset, virtual fields and *Auxiliary Storage* included, the *Fields* tab also lists fields from any *joined layers*. Depending on the origin of the field, a different background color is applied to it.

For each listed field, the dialog also lists read-only characteristics such as its *type*, *type name*, *length* and *precision*. When serving the layer as WMS or WFS, you can also check here which fields could be retrieved.

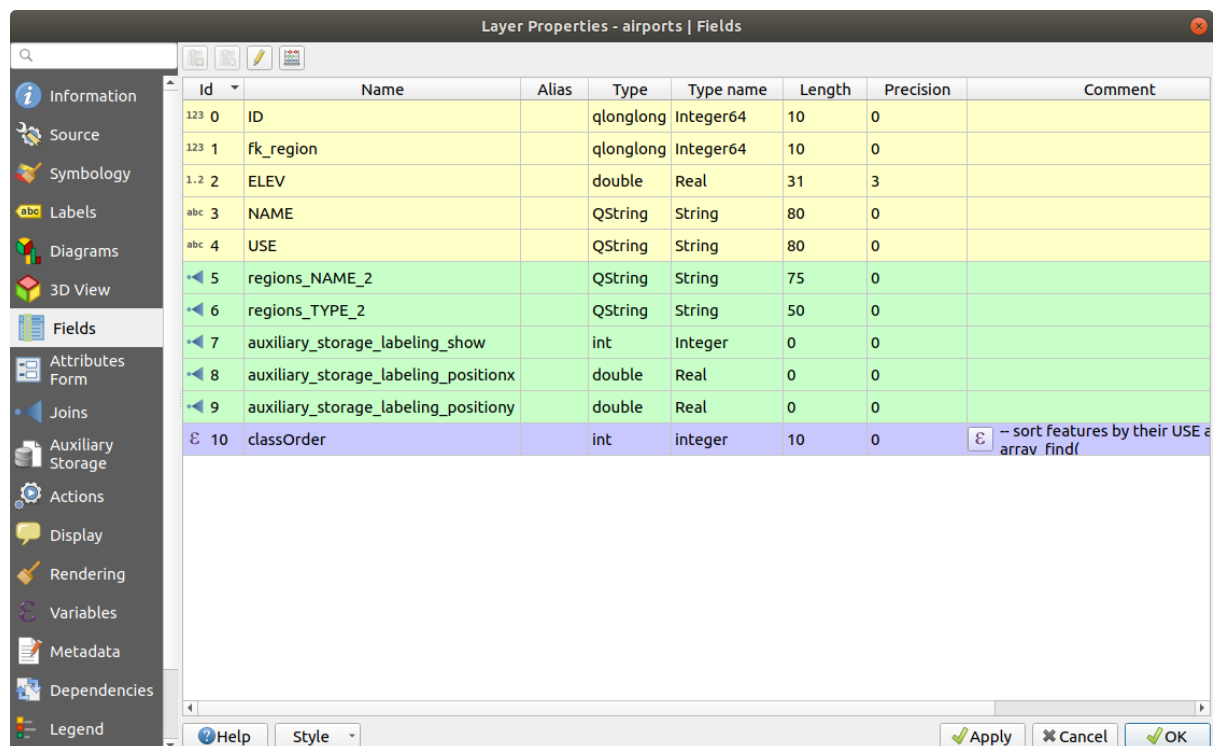


图 14.41: Fields properties tab


14.1.9 Attributes Form Properties

The *Attributes Form* tab helps you set up the form to display when creating new features or querying existing one. You can define:

- the look and the behavior of each field in the feature form or the attribute table (label, widget, constraints...);
- the form's structure (custom or autogenerated);
- extra logic in Python to handle interaction with the form or field widgets.

At the top right of the dialog, you can set whether the form is opened by default when creating new features. This can be configured per layer or globally with the *Suppress attribute form pop-up after feature creation* option in the *Settings ► Options ► Digitizing* menu.

Customizing a form for your data

By default, when you click on a feature with the  Identify Features tool or switch the attribute table to the *form view* mode, QGIS displays a basic form with predefined widgets (generally spinboxes and textboxes --- each field is represented on a dedicated row by its label next to the widget). If *relations* are set on the layer, fields from the referencing layers are shown in an embedded frame at the bottom of the form, following the same basic structure.

This rendering is the result of the default `Autogenerate` value of the *Attribute editor layout* setting in the *Layer properties* ► *Attributes Form* tab. This property holds three different values:

- `Autogenerate`: keeps the basic structure of "one row - one field" for the form but allows to customize each corresponding widget.
- `Drag-and-drop designer`: other than widget customization, the form structure can be made more complex eg, with widgets embedded in groups and tabs.
- `Provide ui file`: allows to use a Qt designer file, hence a potentially more complex and fully featured template, as feature form.

The autogenerated form

When the `Autogenerate` option is on, the *Available widgets* panel shows lists of fields (from the layer and its relations) that would be shown in the form. Select a field and you can configure its appearance and behavior in the right panel:

- adding *custom label and automated checks* to the field;
- setting a *particular widget* to use.

The drag and drop designer

The drag and drop designer allows you to create a form with several containers (tabs or groups) to present the attribute fields, as shown for example in [图 14.42](#).

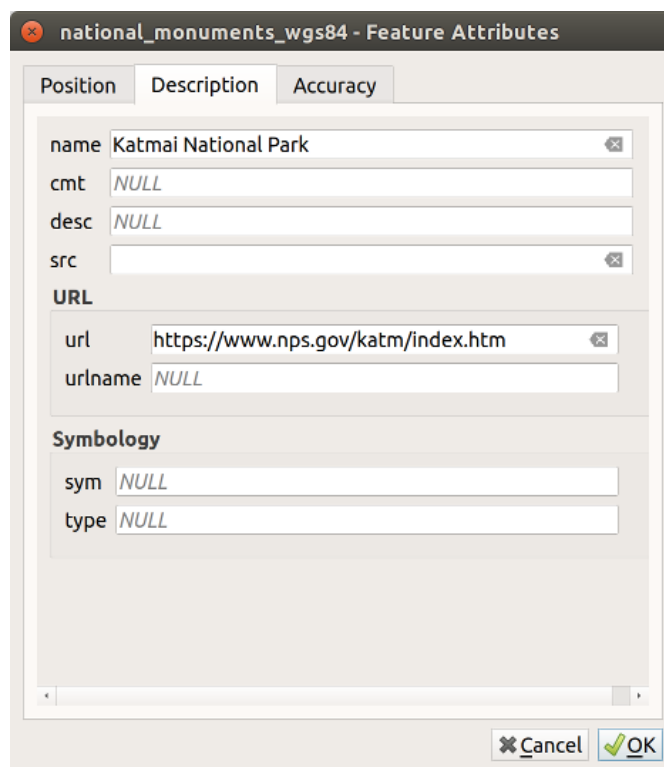




图 14.42: Resulting built-in form with tabs and named groups

1. Choose Drag and drop designer from the *Select attribute layout editor* combobox. This enables the *Form Layout* panel next to the *Available widgets* panel, filled with existing fields. The selected field displays its *properties* in a third panel.
2. Select fields you do not want to use in your *Form Layout* panel and hit the  button to remove them. Drag and drop fields from the other panel to re-add them. The same field can be added multiple times.
3. Drag and drop fields within the *Form Layout* panel to reorder their position.
4. Add containers (tab or group frames) to associate fields that belong to the same category and better structure the form.
 1. The first step is to use the  icon to create a tab in which fields and groups will be displayed
 2. Then set the properties of the container, ie:
 - the name
 - the type, ie a *tab* or a *group in container* (a group inside a tab or another group)
 - and the *number of columns* the embedded fields should be distributed over

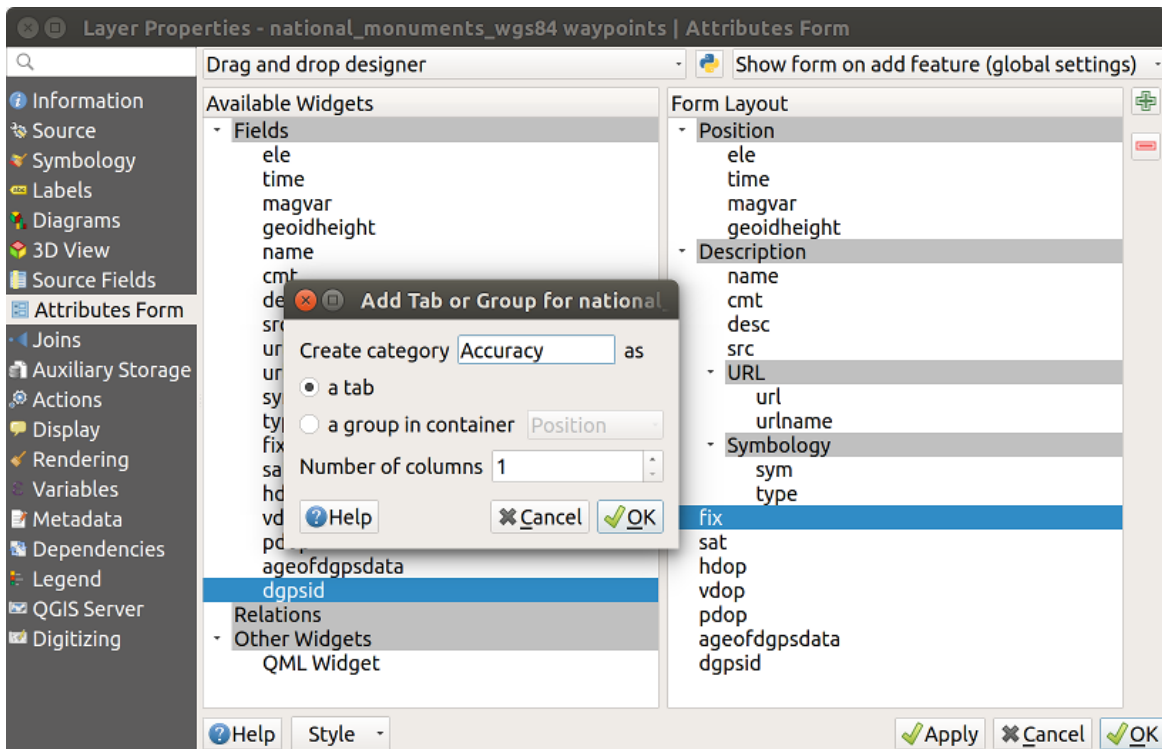




图 14.43: Dialog to create containers with the **Attribute editor layout**

These, and other properties can later be updated by selecting the item and, from the third panel:

- hide or show the container’s label
 - display the container as a group box (only available for tabs).
 - rename the container
 - set the number of columns
 - enter an expression to control the container’s visibility. The expression will be re-evaluated every time values in the form change, and the tab or group box shown/hidden accordingly
 - add a background color
3. You can create as many containers as you want; press the  icon again to create another tab or a group frame under an existing tab.
 5. The next step is to assign the relevant fields to each container, by simple drag and drop. Groups and tabs can also be moved in the same way.
 6. *Customize the widget* of the fields in use
 7. In case the layer is involved in a *one or many to many relation*, drag-and-drop the relation name from the *Available widgets* panel to the *Form Layout* panel. The associated layer attribute form will be embedded at the chosen place in the current layer’s form. As for the other items, select the relation label to configure some properties:
 - hide or show the relation label
 - show the link button
 - show the unlink button
 8. Apply the layer’s properties dialog
 9. Open a feature attribute form (eg, using the  Identify features tool) and it should display the new form.

Using custom ui-file

The `Provide ui-file` option allows you to use complex dialogs made with Qt-Designer. Using a UI-file allows a great deal of freedom in creating a dialog. Note that, in order to link the graphical objects (textbox, combobox...) to the layer's fields, you need to give them the same name.

Use the *Edit UI* to define the path to the file to use.

UI-files can also be hosted on a remote server. In this case, you provide the URL of the form instead of the file path in *Edit UI*.

You'll find some example in the *Creating a new form* lesson of the *QGIS-training-manual-index-reference*. For more advanced information, see <https://woostuff.wordpress.com/2011/09/05/qgis-tips-custom-feature-forms-with-python-logic/>.

Enhance your form with custom functions

QGIS forms can have a Python function that is called when the dialog is opened. Use this function to add extra logic to your dialogs. The form code can be specified in three different ways:

- `load from the environment`: use a function, for example in `startup.py` or from an installed plugin
- `load from an external file`: a file chooser will let you select a Python file from your filesystem or enter a URL for a remote file.
- `provide code in this dialog`: a Python editor will appear where you can directly type the function to use.

In all cases you must enter the name of the function that will be called (`open` in the example below).

An example is (in module `MyForms.py`):

```
def open(dialog, layer, feature):  
    geom = feature.geometry()  
    control = dialog.findChild(QWidget, "My line edit")
```

Reference in Python Init Function like so: `open`

Configure the field behavior

The main part of the *Attributes Form* tab helps you set the type of widget used to fill or display values of the field, in the attribute table or the feature form: you can define how user interacts with each field and the values or range of values that are allowed to be added to each.

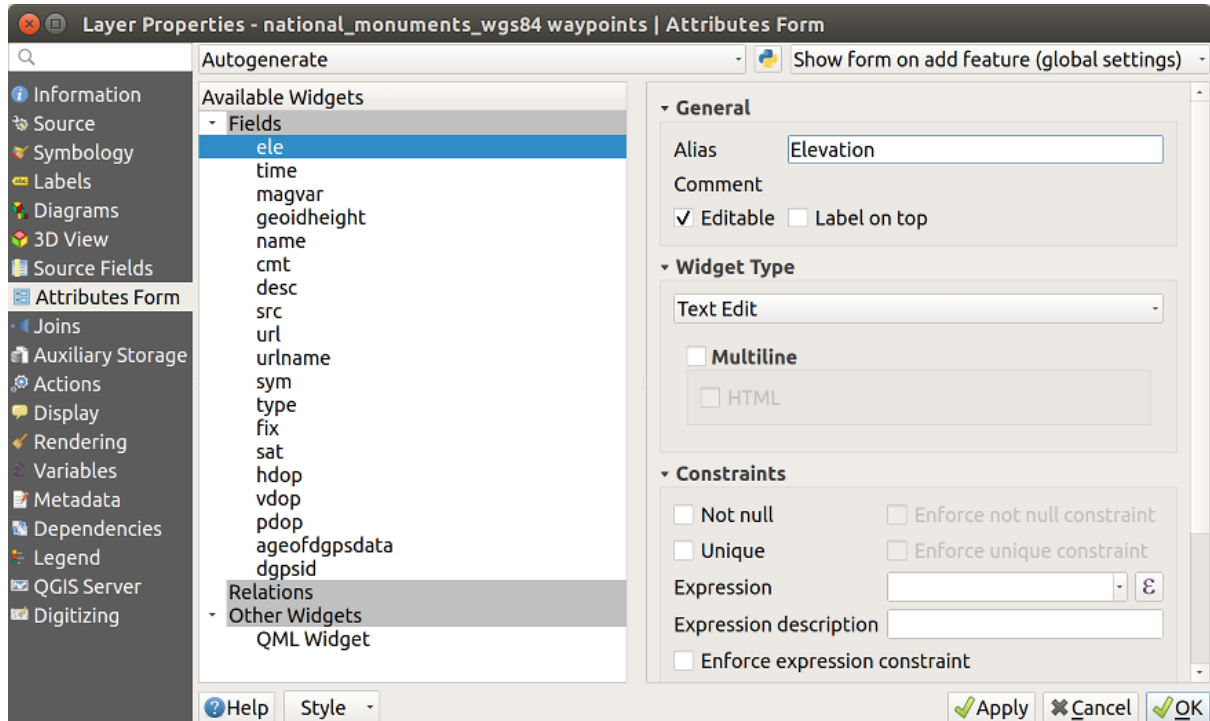


图 14.44: Dialog to select an edit widget for an attribute column

Common settings

Regardless the type of widget applied to the field, there are some common properties you can set to control whether and how a field can be edited.

Widget display

Show label: indicates whether the field name should be displayed in the form (only in the *Drag and drop* designer mode).

General options


- *Alias*: a human readable name to use for fields. The alias will be displayed in the feature form, the attribute table, or in the *Identify results* panel. It can also be used as field name replacement in the *expression builder*, easing expressions understanding and reviews. Aliases are saved in project file.
- *Comment*: displays the field's comment as shown in the *Fields* tab, in a read-only state. This information is shown as tooltip when hovering over the field label in a feature form.
- *Editable*: uncheck this option to set the field read-only (not manually modifiable) even when the layer is in edit mode. Note that checking this setting doesn't override any edit limitation from the provider.
- *Label on top*: places the field name above or beside the widget in the feature form.

Default values

- *Default value*: for new features, automatically populates by default the field with a predefined value or an *expression-based one*. For example, you can:
 - use `$x`, `$length`, `$area` to automatically populate a field with the feature's X coordinate, length, area or any geometric information at its creation;
 - increment a field by 1 for each new feature using `maximum("field")+1`;
 - save the feature creation datetime using `now()`;
 - use *variables* in expressions, making it easier to e.g. insert the operator name (`@user_full_name`), the project file path (`@project_path`), ...



A preview of the resulting default value is displayed at the bottom of the widget.

注解: The `Default value` option is not aware of the values in any other field of the feature being created so it won't be possible to use an expression combining any of those values i.e using an expression like `concat(field1, field2)` may not work.


-  *Apply default value on update*: whenever the feature attribute or geometry is changed, the default value is recalculated. This could be handy to save values like last user that modifies data, last time it was changed...

Constraints

You can constrain the value to insert in the field. This constraint can be:

-  *Not null*: requires the user to provide a value;
-  *Unique*: guarantee the inserted value to be unique throughout the field;
- based on a custom *expression*: e.g. `not regexp_match(col0, '^[A-Za-z]')` will ensure that the value of the field `col0` has only alphabet letters. A short description can be added to help you remember the constraint.

Whenever a value is added or edited in a field, it's submitted to the existing constraints and:

- if it meets all the requirements, a green check is shown beside the field in the form;
- if it does not meet all the requirements, then the field is colored in yellow or orange and a corresponding cross is displayed next to the widget. You can hover over the cross to remind which constraints are applied to the field and fix the value:
 - A yellow cross appears when the unmet constraint is an unenforced one and it does not prevent you to save the changes with the "wrong" values;
 - An orange cross can not be ignored and does not allow you to save your modifications until they meet the constraints. It appears when the  *Enforce constraint* option is checked.

Edit widgets

Based on the field type, QGIS automatically determines and assigns a default widget type to it. You can then replace the widget with any other compatible with the field type. The available widgets are:

- **Checkbox:** Displays a checkbox whose state defines the value to insert.
- **Classification:** Only available when a *categorized symbology* is applied to the layer, displays a combo box with the values of the classes.
- **Color:** Displays a *color widget* allowing to select a color; the color value is stored as a html notation in the attribute table.
- **Date/Time:** Displays a line field which can open a calendar widget to enter a date, a time or both. Column type must be text. You can select a custom format, pop-up a calendar, etc.
- **Enumeration:** Opens a combo box with predefined values fetched from the database. This is currently only supported by the PostgreSQL provider, for fields of `enum` type.
- **Attachment:** Uses a "Open file" dialog to store file path in a relative or absolute mode. It can also be used to display a hyperlink (to document path), a picture or a web page.
- **Hidden:** A hidden attribute column is invisible. The user is not able to see its contents.
- **Key/Value:** Displays a two-columns table to store sets of key/value pairs within a single field. This is currently supported by the PostgreSQL provider, for fields of `hstore` type.
- **List:** Displays a single column table to add different values within a single field. This is currently supported by the PostgreSQL provider, for fields of `array` type.
- **Range:** Allows you to set numeric values from a specific range. The edit widget can be either a slider or a spin box.
- **Relation Reference:** This is the default widget assigned to the referencing field (i.e., the foreign key in the child layer) when a *relation* is set. It provides direct access to the parent feature's form which in turn embeds the list and form of its children.
- **Text Edit (default):** This opens a text edit field that allows simple text or multiple lines to be used. If you choose multiple lines you can also choose html content.
- **Unique Values:** You can select one of the values already used in the attribute table. If 'Editable' is activated, a line edit is shown with autocompletion support, otherwise a combo box is used.
- **Uuid Generator:** Generates a read-only UUID (Universally Unique Identifiers) field, if empty.
- **Value Map:** A combo box with predefined items. The value is stored in the attribute, the description is shown in the combo box. You can define values manually or load them from a layer or a CSV file.
- **Value Relation:** Offers values from a related table in a combobox. You can select layer, key column and value column. Several options are available to change the standard behaviors: allow null value, order by value, allow multiple selections and use of auto-completer. The forms will display either a drop-down list or a line edit field when completer checkbox is enabled.

小技巧: Relative Path in Attachment widget


If the path which is selected with the file browser is located in the same directory as the `.qgs` project file or below, paths are converted to relative paths. This increases portability of a `.qgs` project with multimedia information attached.

14.1.10 Joins Properties



The *Joins* tab allows you to associate features of the current layer (called *Target layer*) to features from another loaded vector layer (or table). The join is based on an attribute that is shared by the layers. The layers can be geometryless (tables) or not but their join attribute should be of the same type.

To create a join:

1. Click the  *Add new join* button. The *Add vector join* dialog appears.
2. Select the *Join layer* you want to connect with the target vector layer
3. Specify the *Join field* and the *Target field* that are common to both the join layer and the target layer
4. Press *OK* and a summary of selected parameters is added to the *Join* panel.

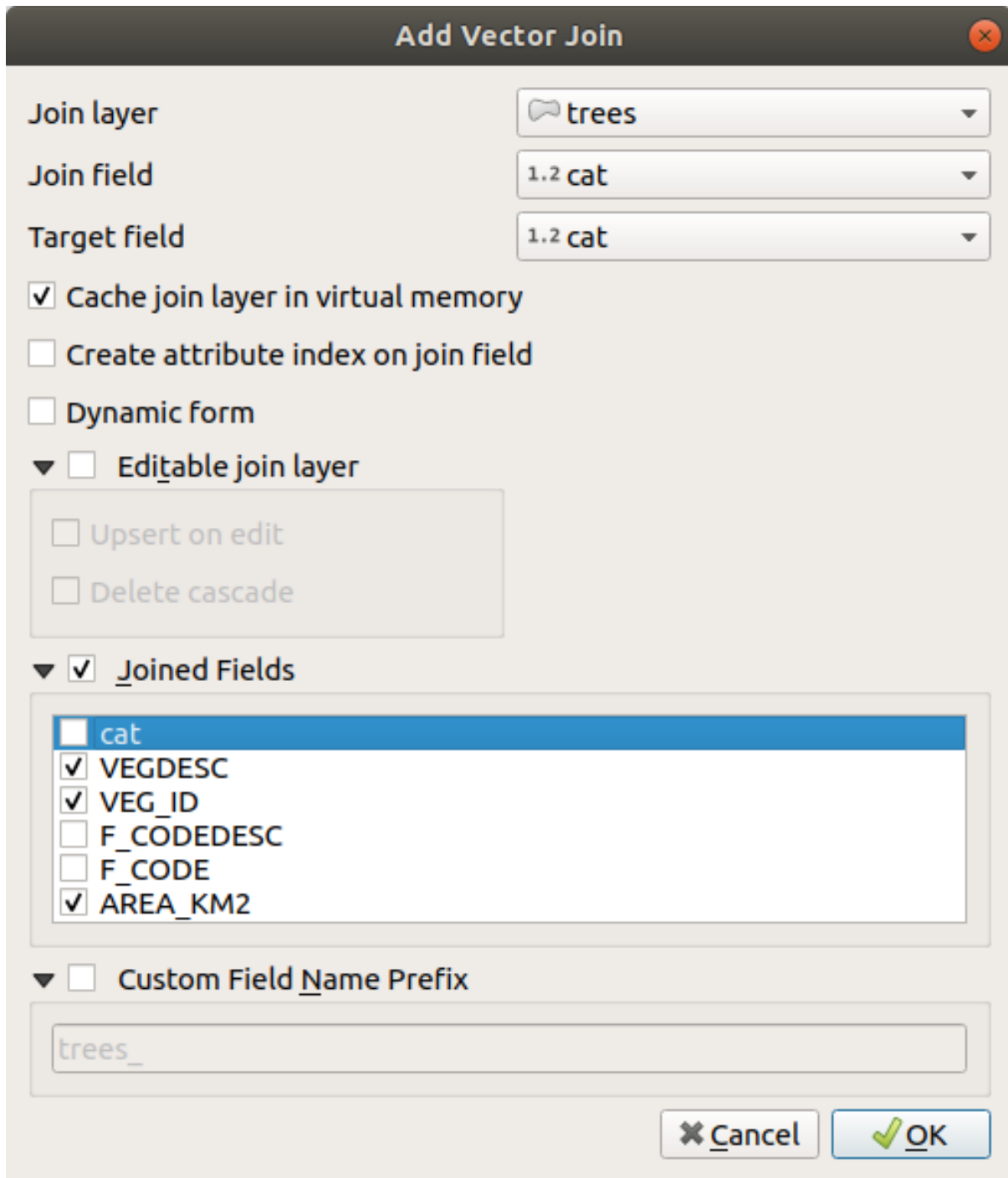





图 14.45: Join an attribute table to an existing vector layer

The steps above will create a join, where **ALL** the attributes of the first matching feature in the join layer is added to the target layer's feature. QGIS provides more options to tweak the join:

- *Cache join layer in virtual memory*: allows you to cache values in memory (without geometries) from the joined layer in order to speed up lookups.
- *Create attribute index on the join field*
- *Dynamic form*: helps to synchronize join fields on the fly, according to the *Target field*. This way, constraints for join fields are also correctly updated. Note that it's deactivated by default because it may be very time consuming if you have a lot of features or a myriad of joins.

- If the target layer is editable, then some icons will be displayed in the attribute table next to fields, in order to inform about their status:
 - : the join layer is not configured to be editable. If you want to be able to edit join features from the target attribute table, then you have to check the option *Editable join layer*.
 - : the join layer is well configured to be editable, but its current status is read only.
 - : the join layer is editable, but synchronization mechanisms are not activated. If you want to automatically add a feature in the join layer when a feature is created in the target layer, then you have to check the option *Upsert on edit*. Symmetrically, the option *Delete cascade* may be activated if you want to automatically delete join features.
- *Joined fields*: instead of adding all the fields from the joined layer, you can specify a subset.
- *Custom field name prefix* for joined fields, in order to avoid name collision

QGIS currently has support for joining non-spatial table formats supported by OGR (e.g., CSV, DBF and Excel), delimited text and the PostgreSQL provider.

14.1.11 Auxiliary Storage Properties

The regular way to customize styling and labeling is to use data-defined properties as described in *Data defined override setup*. However, it may not be possible if the underlying data is read only. Moreover, configuring these data-defined properties may be very time consuming or not desirable! For example, if you want to fully use map tools coming with *The Label Toolbar*, then you need to add and configure more than 20 fields in your original data source (X and Y positions, rotation angle, font style, color and so on).

The Auxiliary Storage mechanism provides the solution to these limitations and awkward configurations. Auxiliary fields are a roundabout way to automatically manage and store these data-defined properties (labels, diagram, symbology...) in a SQLite database thanks to editable joins. This allows you to store properties for layers that aren't editable.

A tab is available in vector layer properties dialog to manage auxiliary storage:

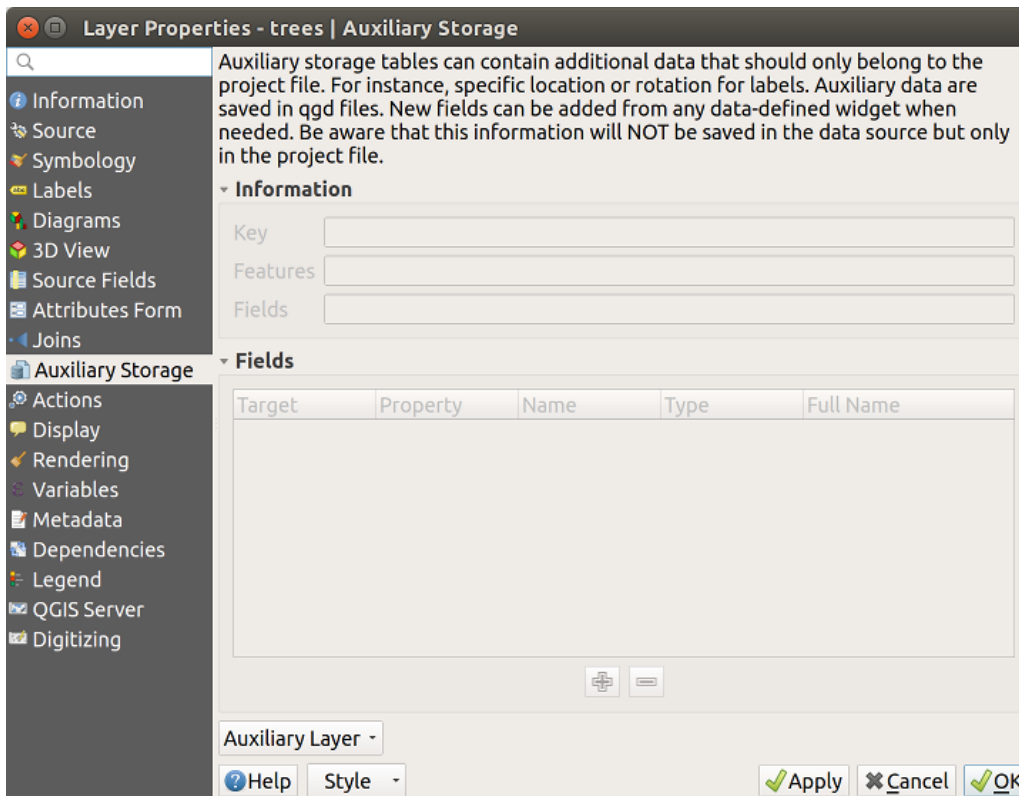


图 14.46: Auxiliary Storage tab

Labeling

Considering that the data source may be customized thanks to data-defined properties without being editable, labeling map tools described in *The Label Toolbar* are always available as soon as labeling is activated.

Actually, the auxiliary storage system needs an auxiliary layer to store these properties in a SQLite database (see *Auxiliary storage database*). Its creation process is run the first time you click on the map while a labeling map tool is currently activated. Then, a window is displayed, allowing you to select the primary key to use for joining (to ensure that features are uniquely identified):

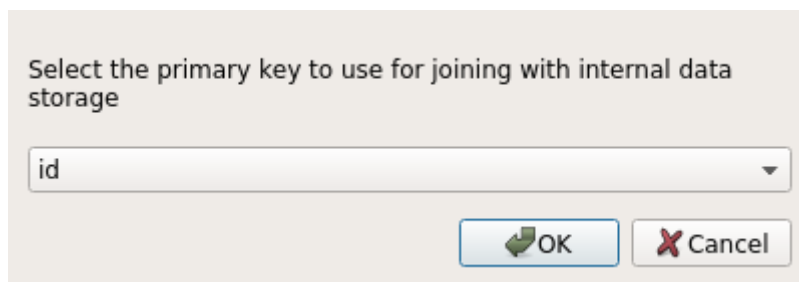


图 14.47: Auxiliary Layer creation dialog

As soon as an auxiliary layer is configured for the current data source, you can retrieve its information in the tab:

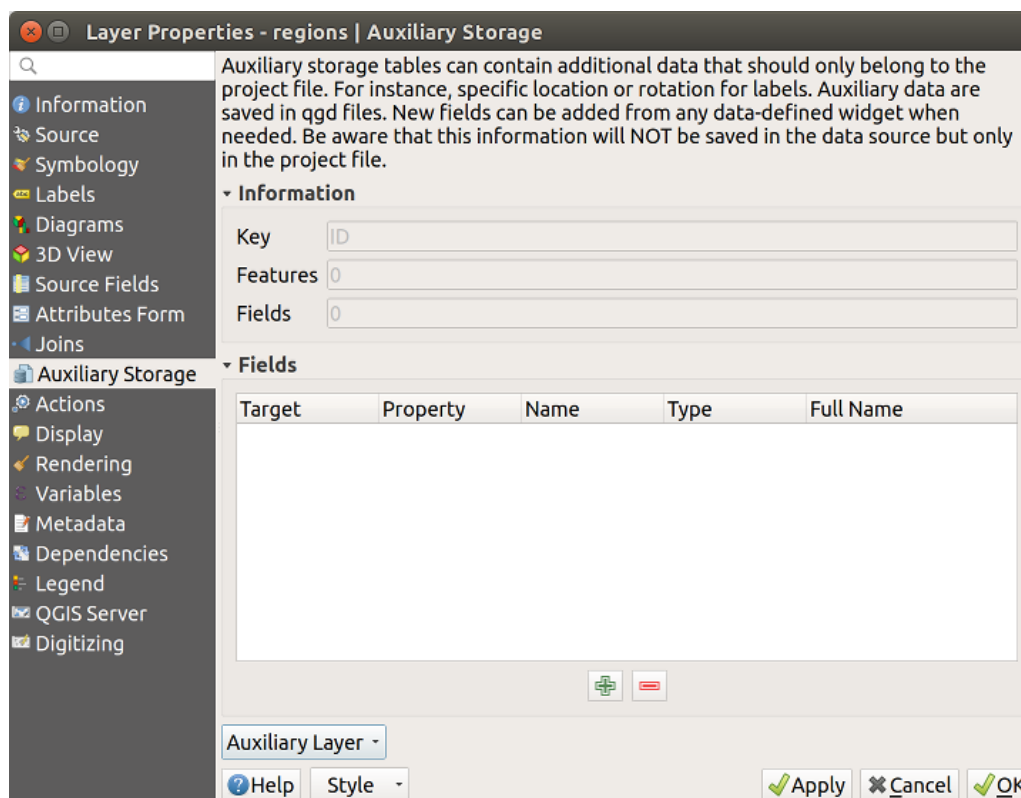



图 14.48: Auxiliary Layer key

The auxiliary layer now has these characteristics:

- the primary key is ID,
- there are 0 features using an auxiliary field,
- there are 0 auxiliary fields.

Now that the auxiliary layer is created, you can edit the layer labels. Click on a label while the  Change Label map tool is activated, then you can update styling properties like sizes, colors, and so on. The corresponding data-defined properties are created and can be retrieved:

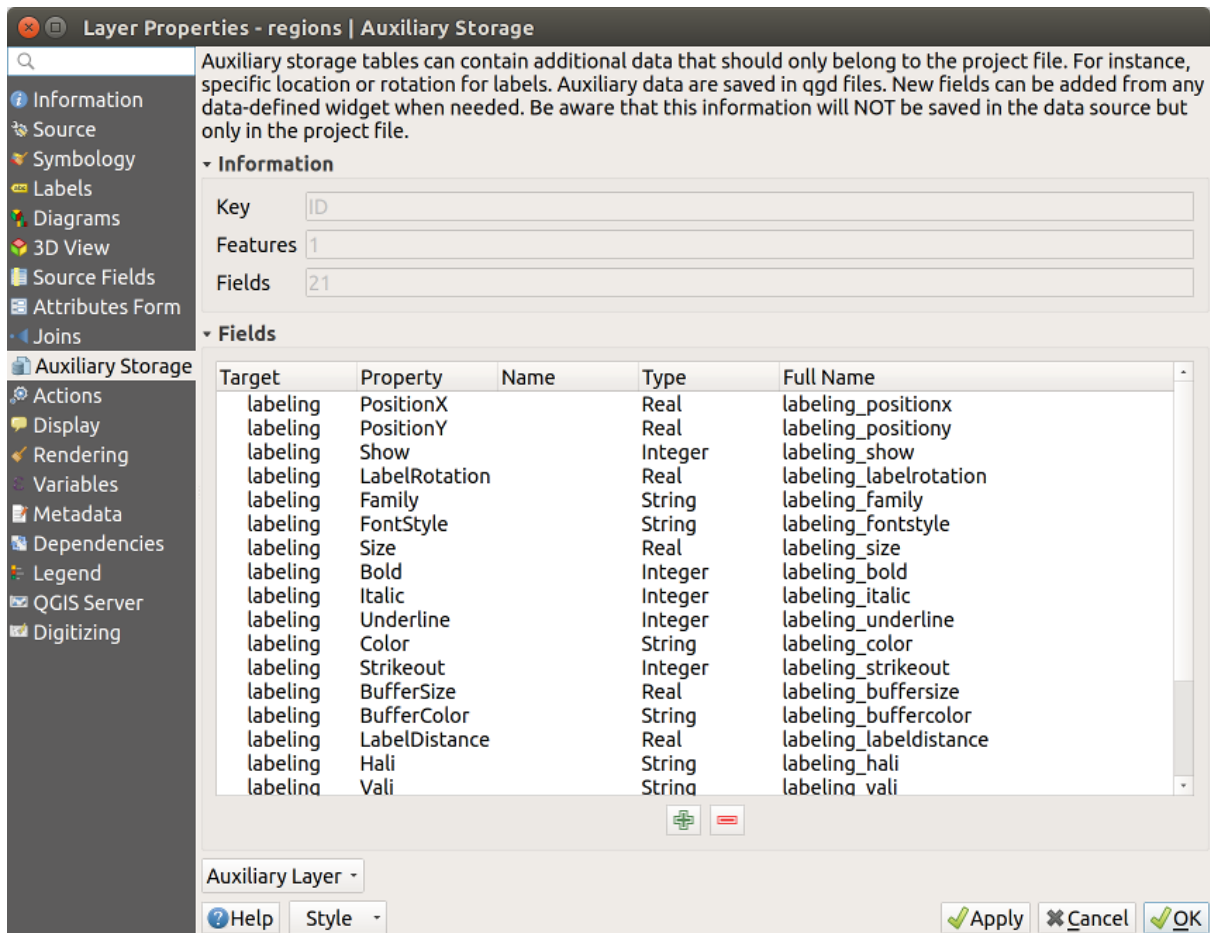



图 14.49: Auxiliary Fields

As you can see in the figure above, 21 fields are automatically created and configured for labeling. For example, the `FontStyle` auxiliary field type is a `String` and is named `labeling_fontstyle` in the underlying SQLite database. There is also 1 feature which is currently using these auxiliary fields.

Notice that the icon  is displayed in the *Labels* properties tab indicating that the data-defined override options are set correctly:

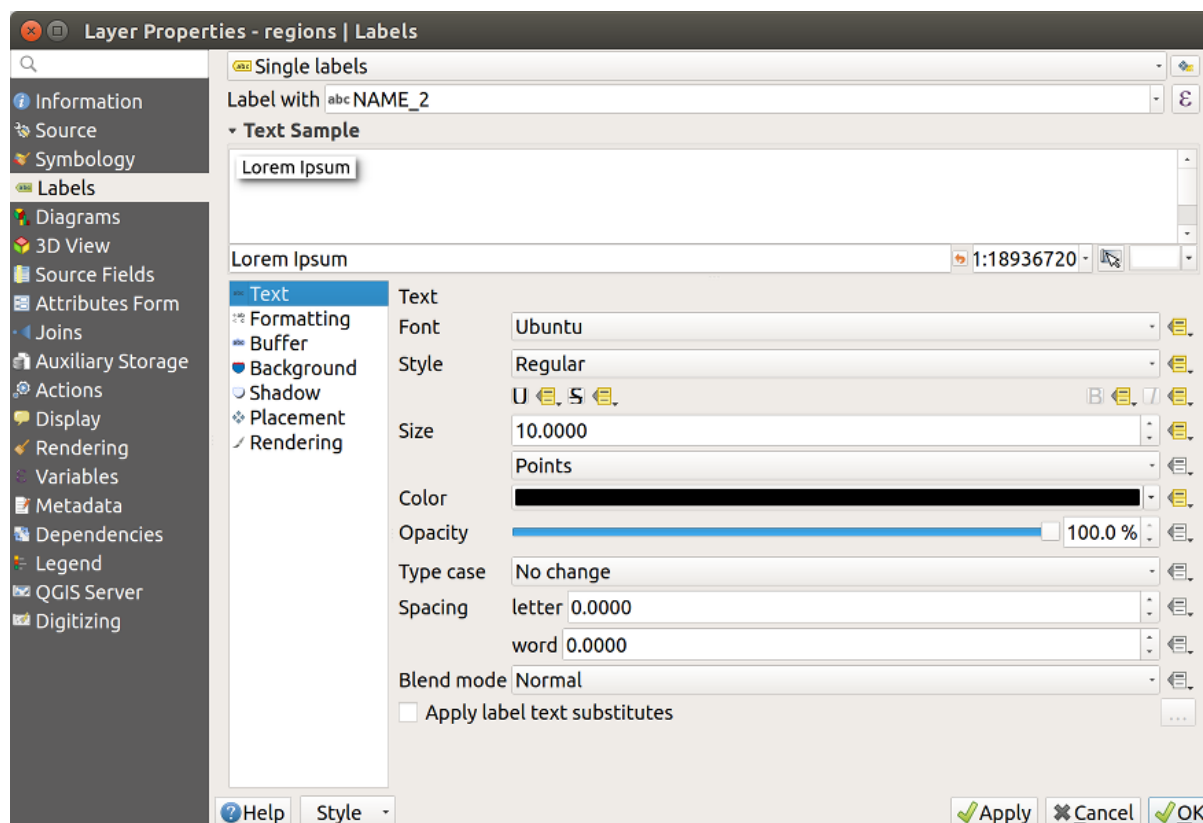




图 14.50: Data-defined properties automatically created

Otherwise, there's another way to create an auxiliary field for a specific property thanks to the  data-defined override button. By clicking on *Store data in the project*, an auxiliary field is automatically created for the *Opacity* field. If you click on this button and the auxiliary layer is not created yet, a window (图 14.47) is first displayed to select the primary key to use for joining.

Symbology

Like the method described above for customizing labels, auxiliary fields can also be used to stylize symbols and diagrams. To do this, click on  Data-defined override and select *Store data in the project* for a specific property. For example, the *Fill color* field:

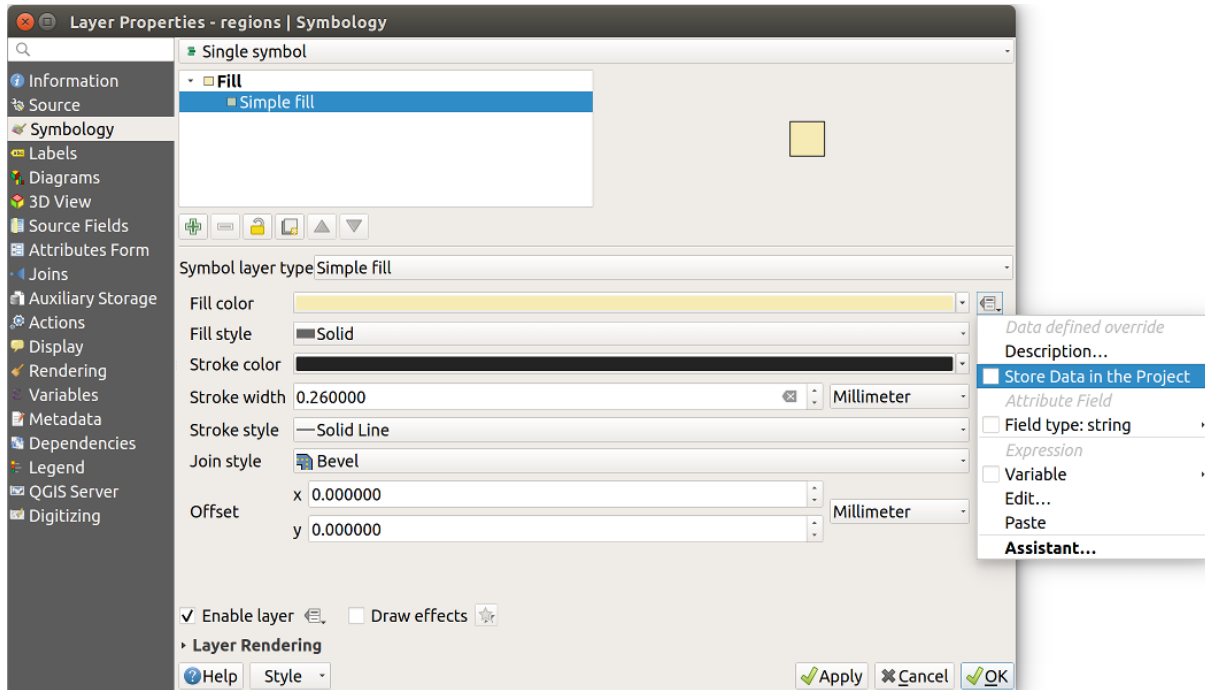


图 14.51: Data-defined property menu for symbol

There are different attributes for each symbol (e.g. fill style, fill color, stroke color, etc...), so each auxiliary field representing an attribute requires a unique name to avoid conflicts. After selecting *Store data in the project*, a window opens and displays the *Type* of the field and prompts you to enter a unique name for the auxiliary field. For example, when creating a *Fill color* auxiliary field the following window opens:

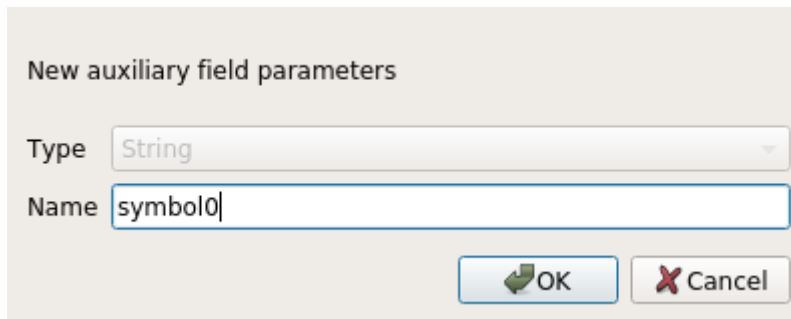


图 14.52: Name of the auxiliary field for a symbol

Once created, the auxiliary field can be retrieved in the auxiliary storage tab:

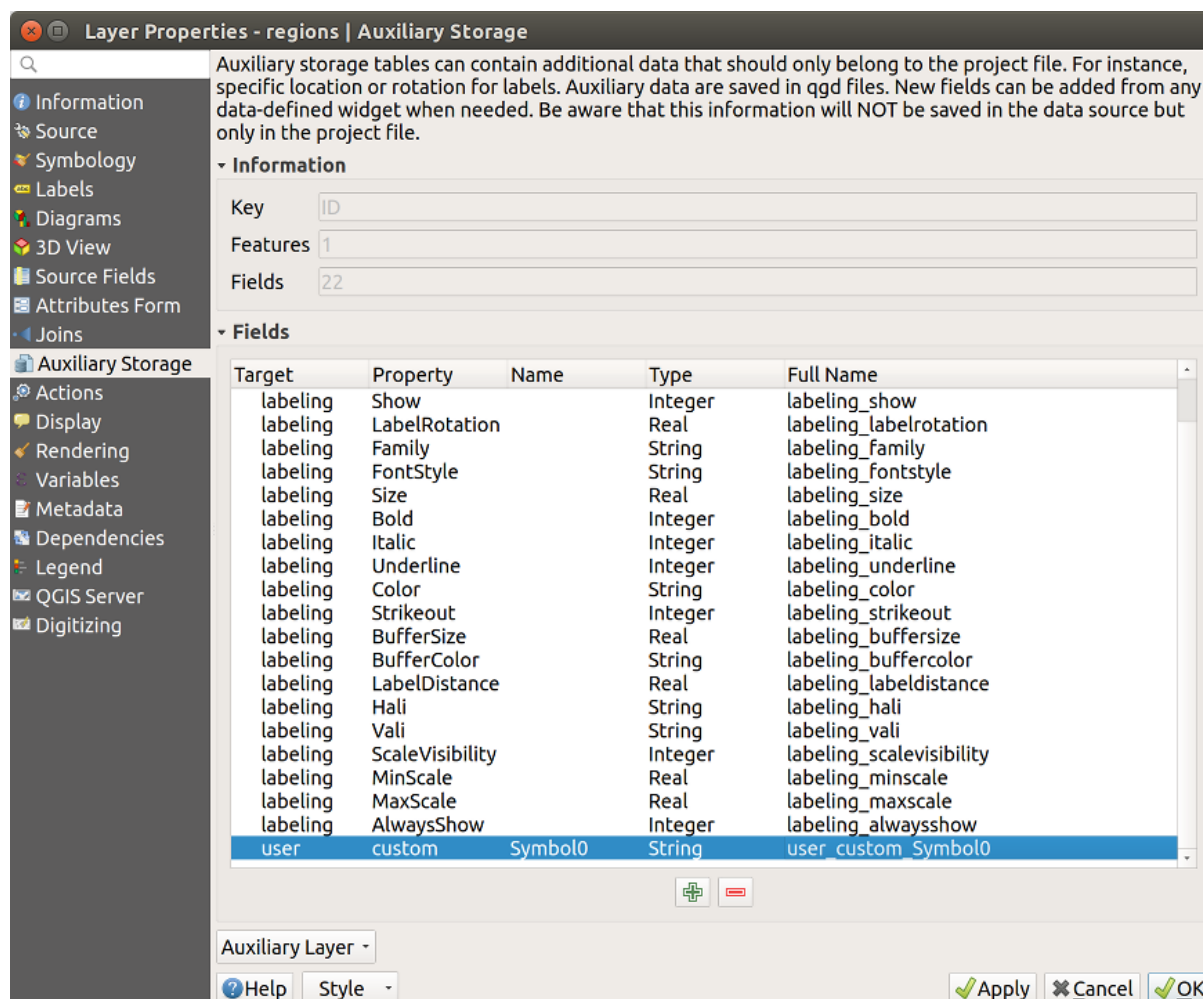


图 14.53: Auxiliary field symbol

Attribute table and widgets

Auxiliary fields can be edited using the *attribute table*. However, not all auxiliary fields are initially visible in the attribute table.

Auxiliary fields representing attributes of a layer’s symbology, labeling, appearance, or diagrams will appear automatically in the attribute table. The exception are attributes that can be modified using the *Label Toolbar* which are hidden by default. Auxiliary fields representing a `Color` have a widget **Color** set by default, otherwise auxiliary fields default to the **Text Edit** widget.

Auxiliary fields that represent attributes that can be modified using the *Label toolbar* are **Hidden** in the attribute table by default. To make a field visible, open the *Attribute Form properties tab* and change the value of an auxiliary field *Widget Type* from **Hidden** to another relevant value. For example, change the `auxiliary_storage_labeling_size` to **Text Edit** or change `auxiliary_storage_labeling_color` to the **Color** widget. Those fields will now be visible in the attribute table.

Auxiliary fields in the attribute table will appear like the following image:

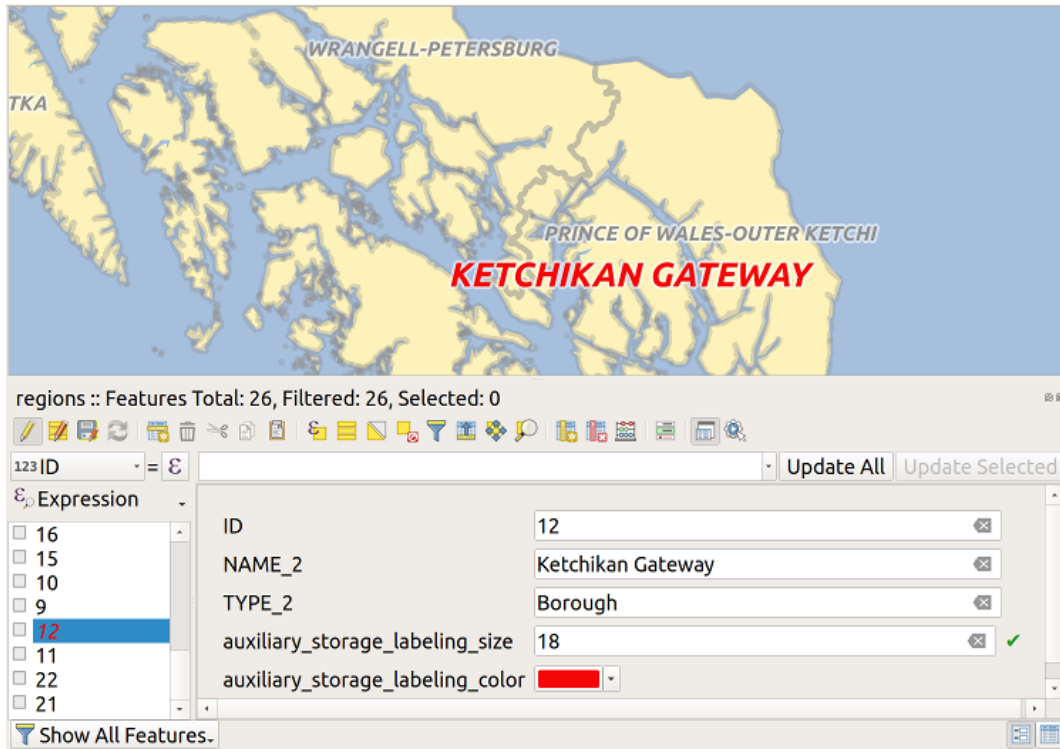


图 14.54: Form with auxiliary fields

Management

The *Auxiliary Layer* menu allows you to manage the auxiliary fields:

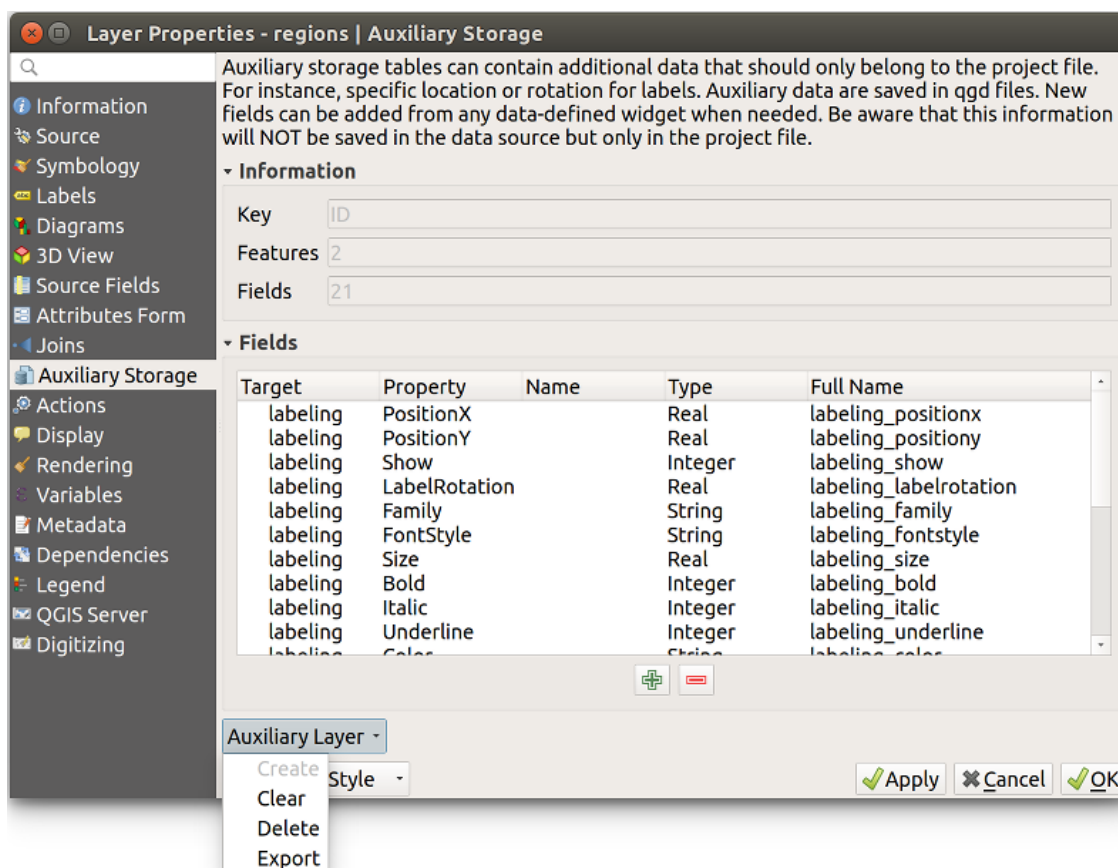


图 14.55: Auxiliary layer management

The first item *Create* is disabled in this case because the auxiliary layer is already created. But in case of a fresh work, you can use this action to create an auxiliary layer. As explained in [Labeling](#), a primary key will be needed then.

The *Clear* action allows to keep all auxiliary fields, but remove their contents. This way, the number of features using these fields will fall to 0.

The *Delete* action completely removes the auxiliary layer. In other words, the corresponding table is deleted from the underlying SQLite database and properties customization are lost.

Finally, the *Export* action allows to save the auxiliary layer as a *new vector layer*. Note that geometries are not stored in auxiliary storage. However, in this case, geometries are exported from the original data source too.

Auxiliary storage database

When you save your project with the `.qgs` format, the SQLite database used for auxiliary storage is saved at the same place but with the extension `.qgd`.

For convenience, an archive may be used instead thanks to the `.qgz` format. In this case, `.qgd` and `.qgs` files are both embedded in the archive.

14.1.12 Actions Properties



QGIS provides the ability to perform an action based on the attributes of a feature. This can be used to perform any number of actions, for example, running a program with arguments built from the attributes of a feature or passing parameters to a web reporting tool.

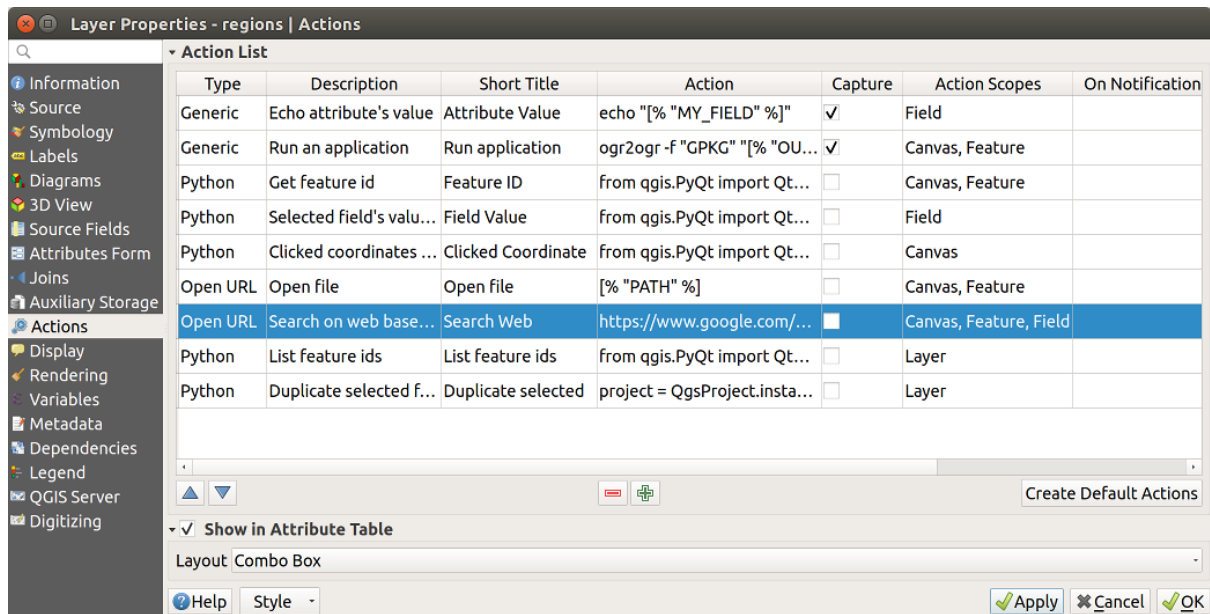


图 14.56: Overview action dialog with some sample actions


Actions are useful when you frequently want to run an external application or view a web page based on one or more values in your vector layer. They are divided into six types and can be used like this:

- Generic, Mac, Windows and Unix actions start an external process.
- Python actions execute a Python expression.
- Generic and Python actions are visible everywhere.
- Mac, Windows and Unix actions are visible only on the respective platform (i.e., you can define three 'Edit' actions to open an editor and the users can only see and execute the one 'Edit' action for their platform to run the editor).

There are several examples included in the dialog. You can load them by clicking on *Create Default Actions*. To edit any of the examples, double-click its row. One example is performing a search based on an attribute value. This concept is used in the following discussion.

The *Show in Attribute Table* allows you to display in the attribute table dialog the checked feature-scoped actions, either as *Combo Box* or as *Separate Buttons* (see *Configuring the columns*).

Defining Actions

To define an attribute action, open the vector *Layer Properties* dialog and click on the *Actions* tab. In the *Actions* tab, click the  Add a new action to open the *Edit Action* dialog.

Select the action *Type* and provide a descriptive name for the action. The action itself must contain the name of the application that will be executed when the action is invoked. You can add one or more attribute field values as arguments to the application. When the action is invoked, any set of characters that start with a % followed by the name of a field will be replaced by the value of that field. The special characters %% will be replaced by the value of the field that was selected from the identify results or attribute table (see *using_actions* below). Double quote marks can be used to group text into a single argument to the program, script or command. Double quotes will be ignored if preceded by a backslash.

The *Action Scopes* allows you to define *where* the action should be available. You have 4 different choices:

1. *Feature Scope*: action is available when right click in the cell within the attribute table.
2. *Field Scope*: action is available when right click in the cell within the attribute table, in the feature form and in the default action button of the main toolbar.
3. *Layer Scope*: action is available in the action button in the attribute table toolbar. Be aware that this type of action involves the entire layer and not the single features.
4. *Canvas*: action is available in the main action button in the toolbar.

If you have field names that are substrings of other field names (e.g., `col1` and `col10`), you should indicate that by surrounding the field name (and the % character) with square brackets (e.g., `[%col10]`). This will prevent the `%col10` field name from being mistaken for the `%col1` field name with a 0 on the end. The brackets will be removed by QGIS when it substitutes in the value of the field. If you want the substituted field to be surrounded by square brackets, use a second set like this: `[[%col10]]`.

Using the *Identify Features* tool, you can open the *Identify Results* dialog. It includes a (*Derived*) item that contains information relevant to the layer type. The values in this item can be accessed in a similar way to the other fields by preceding the derived field name with (*Derived*) .. For example, a point layer has an X and Y field, and the values of these fields can be used in the action with `%(Derived) .X` and `%(Derived) .Y`. The derived attributes are only available from the *Identify Results* dialog box, not the *Attribute Table* dialog box.

Two example actions are shown below:



- `konqueror https://www.google.com/search?q=%nam`
- `konqueror https://www.google.com/search?q=%%`

In the first example, the web browser `konqueror` is invoked and passed a URL to open. The URL performs a Google search on the value of the `nam` field from our vector layer. Note that the application or script called by the action must be in the path, or you must provide the full path. To be certain, we could rewrite the first example as: `/opt/kde3/bin/konqueror https://www.google.com/search?q=%nam`. This will ensure that the `konqueror` application will be executed when the action is invoked.



The second example uses the `%%` notation, which does not rely on a particular field for its value. When the action is invoked, the `%%` will be replaced by the value of the selected field in the identify results or attribute table.

Using Actions

QGIS offers many ways to execute actions you enabled on a layer. Depending on their settings, they can be available:

- in the drop-down menu of  Run Feature Action button from the *Attributes toolbar* or *Attribute table* dialog;
- when right-clicking a feature with the  Identify Features tool (see *Identifying Features* for more information);
- from the *Identify Results* panel, under the *Actions* section;
- as items of an *Actions* column in the *Attribute Table* dialog.

If you are invoking an action that uses the `%%` notation, right-click on the field value in the *Identify Results* dialog or the *Attribute Table* dialog that you wish to pass to the application or script.


Here is another example that pulls data out of a vector layer and inserts it into a file using `bash` and the `echo` command (so it will only work on  or perhaps ). The layer in question has fields for a species name `taxon_name`, latitude `lat` and longitude `long`. We would like to be able to make a spatial selection of localities and export these field values to a text file for the selected record (shown in yellow in the QGIS map area). Here is the action to achieve this:

```
bash -c "echo \"%taxon_name %lat %long\" >> /tmp/species_localities.txt"
```

After selecting a few localities and running the action on each one, opening the output file will show something like this:

```
Acacia mearnsii -34.0800000000 150.0800000000
Acacia mearnsii -34.9000000000 150.1200000000
Acacia mearnsii -35.2200000000 149.9300000000
Acacia mearnsii -32.2700000000 150.4100000000
```

As an exercise, we can create an action that does a Google search on the `lakes` layer. First, we need to determine the URL required to perform a search on a keyword. This is easily done by just going to Google and doing a simple search, then grabbing the URL from the address bar in your browser. From this little effort, we see that the format is <https://www.google.com/search?q=QGIS>, where `QGIS` is the search term. Armed with this information, we can proceed:

1. Make sure the `lakes` layer is loaded.
2. Open the *Layer Properties* dialog by double-clicking on the layer in the legend, or right-click and choose *Properties* from the pop-up menu.
3. Click on the *Actions* tab.
4. Click  Add a new action.
5. Choose the *Open* action type,
6. Enter a name for the action, for example `Google Search`.
7. Additionally you can add a *Short Name* or even an *Icon*.
8. Choose the action *Scope*. See *Defining Actions* for further information. Leave the default settings for this example.
9. For the action, we need to provide the name of the external program to run. In this case, we can use Firefox. If the program is not in your path, you need to provide the full path.
10. Following the name of the external application, add the URL used for doing a Google search, up to but not including the search term: `https://www.google.com/search?q=`
11. The text in the *Action* field should now look like this: `https://www.google.com/search?q=`
12. Click on the drop-down box containing the field names for the `lakes` layer. It's located just to the left of the *Insert* button.
13. From the drop-down box, select 'NAMES' and click *Insert*.
14. Your action text now looks like this:


```
https://www.google.com/search?q=[%NAMES%]
```
15. To finalize and add the action, click the *OK* button.

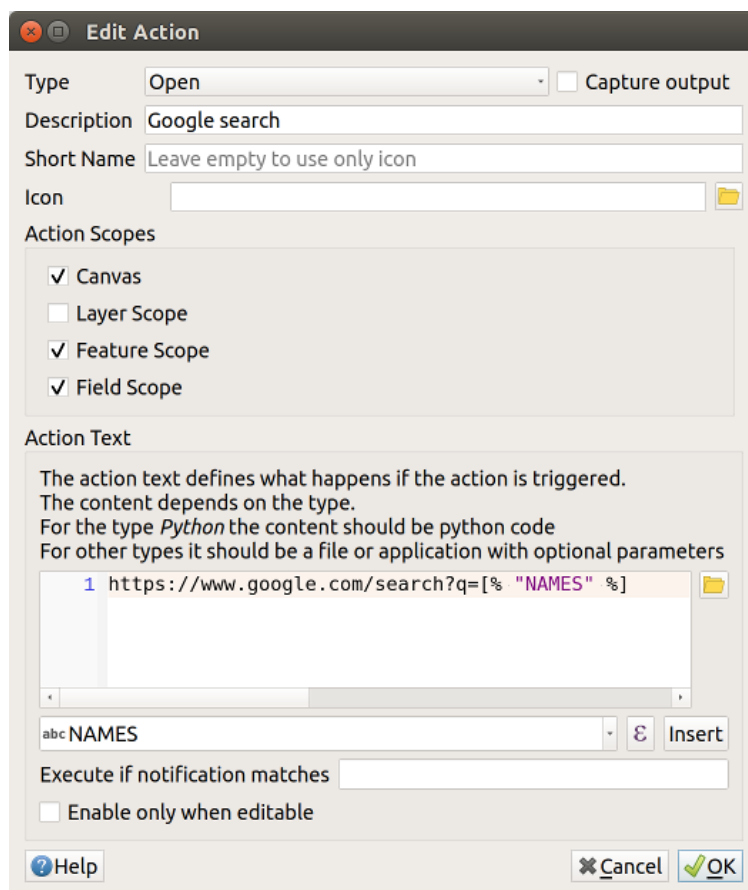


图 14.57: Edit action dialog configured with the example

This completes the action, and it is ready to use. The final text of the action should look like this:

```
https://www.google.com/search?q=[%NAMES%]
```

We can now use the action. Close the *Layer Properties* dialog and zoom in to an area of interest. Make sure the `lakes` layer is active and identify a lake. In the result box you'll now see that our action is visible:

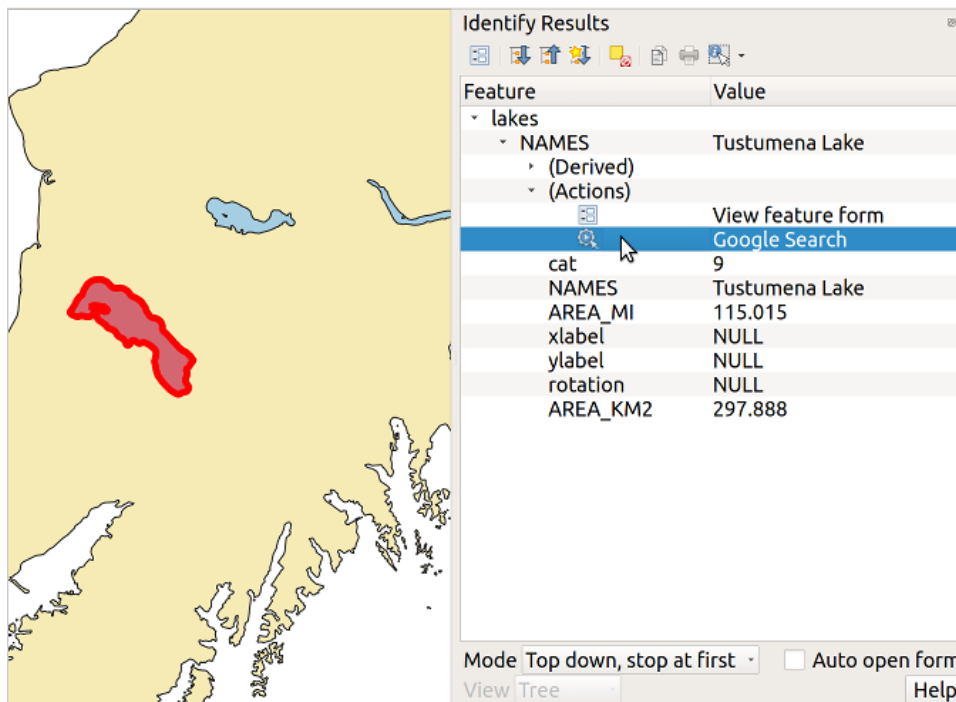


图 14.58: Select feature and choose action

When we click on the action, it brings up Firefox and navigates to the URL <https://www.google.com/search?q=Tustumena>. It is also possible to add further attribute fields to the action. Therefore, you can add a + to the end of the action text, select another field and click on *Insert Field*. In this example, there is just no other field available that would make sense to search for.

You can define multiple actions for a layer, and each will show up in the *Identify Results* dialog.

You can also invoke actions from the attribute table by selecting a row and right-clicking, then choosing the action from the pop-up menu.

There are all kinds of uses for actions. For example, if you have a point layer containing locations of images or photos along with a file name, you could create an action to launch a viewer to display the image. You could also use actions to launch web-based reports for an attribute field or combination of fields, specifying them in the same way we did in our Google search example.

We can also make more complex examples, for instance, using **Python** actions.

Usually, when we create an action to open a file with an external application, we can use absolute paths, or eventually relative paths. In the second case, the path is relative to the location of the external program executable file. But what about if we need to use relative paths, relative to the selected layer (a file-based one, like Shapefile or SpatialLite)? The following code will do the trick:

```
command = "firefox"
imagerelpath = "images_test/test_image.jpg"
layer = qgis.utils.iface.activeLayer()
import os.path
layerpath = layer.source() if layer.providerType() == 'ogr'
else (qgis.core.QgsDataSourceURI(layer.source()).database()
      if layer.providerType() == 'spatialite' else None)
path = os.path.dirname(str(layerpath))
image = os.path.join(path, imagerelpath)
import subprocess
subprocess.Popen( [command, image ] )
```

We just have to remember that the action is one of type *Python* and the *command* and *imagerelpath* variables must be changed to fit our needs.

But what about if the relative path needs to be relative to the (saved) project file? The code of the Python action would be:

```
command = "firefox"
imagerelpath = "images_test/test_image.jpg"
projectpath = qgis.core.QgsProject.instance().fileName()
import os.path
path = os.path.dirname(str(projectpath)) if projectpath != '' else None
image = os.path.join(path, imagerelpath)
import subprocess
subprocess.Popen( [command, image ] )
```

Another Python action example is the one that allows us to add new layers to the project. For instance, the following examples will add to the project respectively a vector and a raster. The names of the files to be added to the project and the names to be given to the layers are data driven (*filename* and *layername* are column names of the table of attributes of the vector where the action was created):

```
qgis.utils.iface.addVectorLayer('/yourpath/[% "filename" %].shp',
    '[% "layername" %]', 'ogr')
```


To add a raster (a TIF image in this example), it becomes:

```
qgis.utils.iface.addRasterLayer('/yourpath/[% "filename" %].tif',
    '[% "layername" %]')
```

14.1.13 Display Properties



The *Display* tab helps you configure fields to use for feature identification:

- The *Display name*: based on a field or an *expression*. This is:
 - the label shown on top of the feature information in the *Identify tool* results
 - the field used in the *locator bar* when looking for features in all layers
 - the feature identifier in the attribute table *form view*
 - the feature identifier when the map or layout is exported to a layered output format such as GeoPDF
 - the map tip information, i.e. the message displayed in the map canvas when hovering over a feature of the active layer with the  Show Map Tips icon pressed. Applicable when no *HTML Map Tip* is set.
- The *HTML Map Tip* is specifically created for the map tips: it's a more complex and full HTML text mixing fields, expressions and html tags (multiline, fonts, images, hyperlink...).

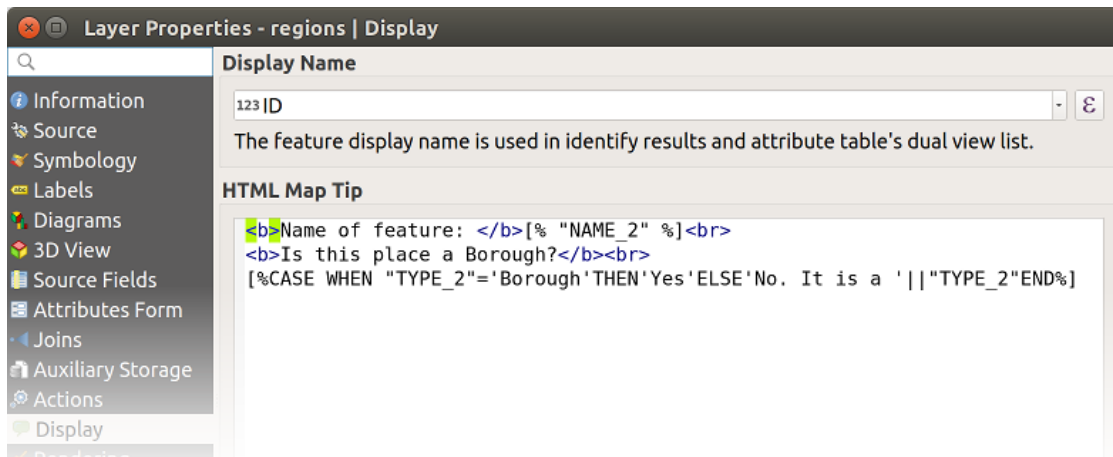


图 14.59: HTML code for map tip



To activate map tips, select the menu option *View ► Show Map Tips* or click on the  Show Map Tips icon of the *Attributes Toolbar*. Map tip is a cross-session feature meaning that once activated, it stays on and apply to any layer in any project, even in future QGIS sessions until it's toggled off.




图 14.60: Map tip made with HTML code

14.1.14 Rendering Properties

Scale dependent visibility

You can set the *Maximum (inclusive)* and *Minimum (exclusive)* scale, defining a range of scale in which features will be visible. Out of this range, they are hidden. The  Set to current canvas scale button helps you use the current map canvas scale as boundary of the range visibility. See [Scale Dependent Rendering](#) for more information.

Simplify geometry

QGIS offers support for on-the-fly feature generalisation. This can improve rendering times when drawing many complex features at small scales. This feature can be enabled or disabled in the layer settings using the  *Simplify geometry* option. There is also a global setting that enables generalisation by default for newly added layers (see [global simplification](#) for more information).

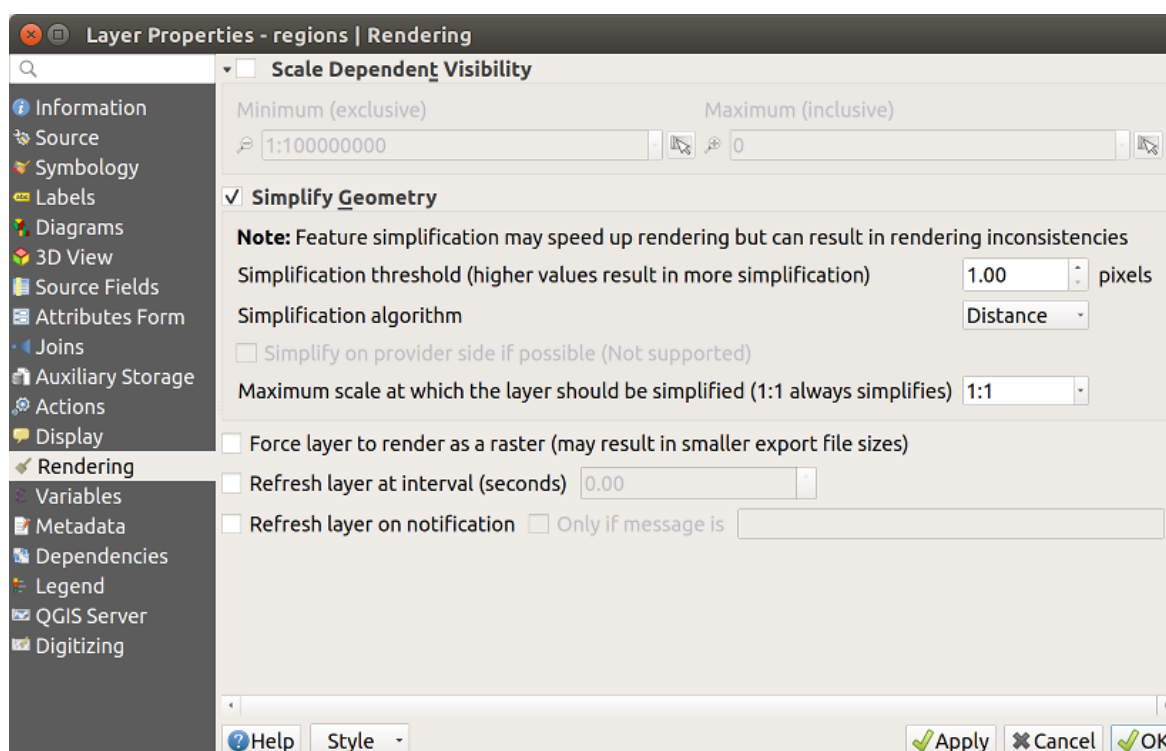



图 14.61: Layer Geometry Simplification dialog

注解: Feature generalisation may introduce artefacts into your rendered output in some cases. These may include slivers between polygons and inaccurate rendering when using offset-based symbol layers.


While rendering extremely detailed layers (e.g. polygon layers with a huge number of nodes), this can cause layout exports in PDF/SVG format to be huge as all nodes are included in the exported file. This can also make the resultant file very slow to work with/open in other programs.



Checking  *Force layer to render as raster* forces these layers to be rasterised so that the exported files won't have to include all the nodes contained in these layers and the rendering is therefore sped up.

You can also do this by forcing the layout to export as a raster, but that is an all-or-nothing solution, given that the rasterisation is applied to all layers.


Refresh layer at interval (seconds): set a timer to automatically refresh individual layers at a matching interval. Canvas updates are deferred in order to avoid refreshing multiple times if more than one layer has an auto update interval set. Depending on the data provider (e.g. PostgreSQL), notifications can be sent to QGIS when changes are applied to the data source, out of QGIS. Use the *Refresh layer on notification* option to trigger an update. You can also limit the layer refresh to a specific message set in the *Only if message is* text box.

14.1.15 Variables Properties

 The *Variables* tab lists all the variables available at the layer's level (which includes all global and project's variables).

It also allows the user to manage layer-level variables. Click the  button to add a new custom layer-level variable. Likewise, select a custom layer-level variable from the list and click the  button to remove it. More information on variables usage in the General Tools *Storing values in Variables* section.

14.1.16 Metadata Properties


 The *Metadata* tab provides you with options to create and edit a metadata report on your layer. Information to fill concern:

- the data *Identification*: basic attribution of the dataset (parent, identifier, title, abstract, language...);
- the *Categories* the data belongs to. Alongside the **ISO** categories, you can add custom ones;
- the *Keywords* to retrieve the data and associated concepts following a standard based vocabulary;
- the *Access* to the dataset (licenses, rights, fees, and constraints);
- the *Extent* of the dataset, either spatial one (CRS, map extent, altitudes) or temporal;
- the *Contact* of the owner(s) of the dataset;
- the *Links* to ancillary resources and related information;
- the *History* of the dataset.

A summary of the filled information is provided in the *Validation* tab and helps you identify potential issues related to the form. You can then either fix them or ignore them.

Metadata are currently saved in the project file. They can also be saved in a `.qmd` file alongside file based layers or in a local `.sqlite` database for remote layers (e.g. PostGIS).

14.1.17 Dependencies Properties

 The *Dependencies* tab allows to declare data dependencies between layers. A data dependency occurs when a data modification in a layer, not by direct user manipulation, may modify data of other layers. This is the case for instance when geometry of a layer is updated by a database trigger or custom PyQGIS scripting after modification of another layer's geometry.

In the *Dependencies* tab, you can select any layers which may externally alter the data in the current layer. Correctly specifying dependent layers allows QGIS to invalidate caches for this layer when the dependent layers are altered.

14.1.18 Legend Properties

The *Legend* properties tab provides you with advanced settings for the *Layers panel* and/or the *print layout legend*. These options include:

- Text on symbols*: In some cases it can be useful to add extra information to the symbols in the legend. With this frame, you can affect to any of the symbols used in the layer symbology a text that is displayed over the symbol, in both *Layers* panel and print layout legend. This mapping is done by typing each text next to the symbol in the table widget or filling the table using the *Set Labels from Expression* button. Text appearance is handled through the font and color selector widgets of the *Text Format* button.

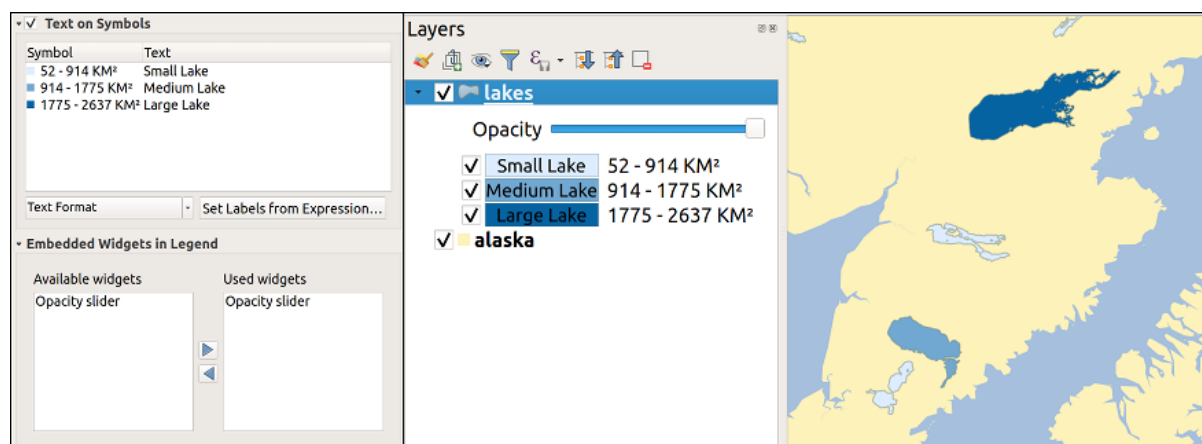


图 14.62: Setting text on symbols (left) and its rendering in the *Layers* panel (right)

- a list of widgets you can embed within the layer tree in the *Layers* panel. The idea is to have a way to quickly access some actions that are often used with the layer (setup transparency, filtering, selection, style or other stuff...).

By default, QGIS provides transparency widget but this can be extended by plugins registering their own widgets and assign custom actions to layers they manage.

14.1.19 QGIS Server Properties

The *QGIS Server* tab consists of *Description*, *Attribution*, *MetadataURL*, and *LegendUrl* sections.

From the *Description* section, you can change the *Short name* used to reference the layer in requests (to learn more about short names, read `server_short_name`). You can also add or edit a *Title* and *Abstract* for the layer, or define a *Keyword list* here. These keyword lists can be used in a metadata catalog. If you want to use a title from an XML metadata file, you have to fill in a link in the *DataUrl* field.

Use *Attribution* to get attribute data from an XML metadata catalog.

In *MetadataUrl*, you can define the general path to the XML metadata catalog. This information will be saved in the QGIS project file for subsequent sessions and will be used for QGIS server.

In the *LegendUrl* section, you can provide the url of a legend image in the url field. You can use the *Format* drop-down option to apply the appropriate format of the image. Currently png, jpg and jpeg image formats are supported.

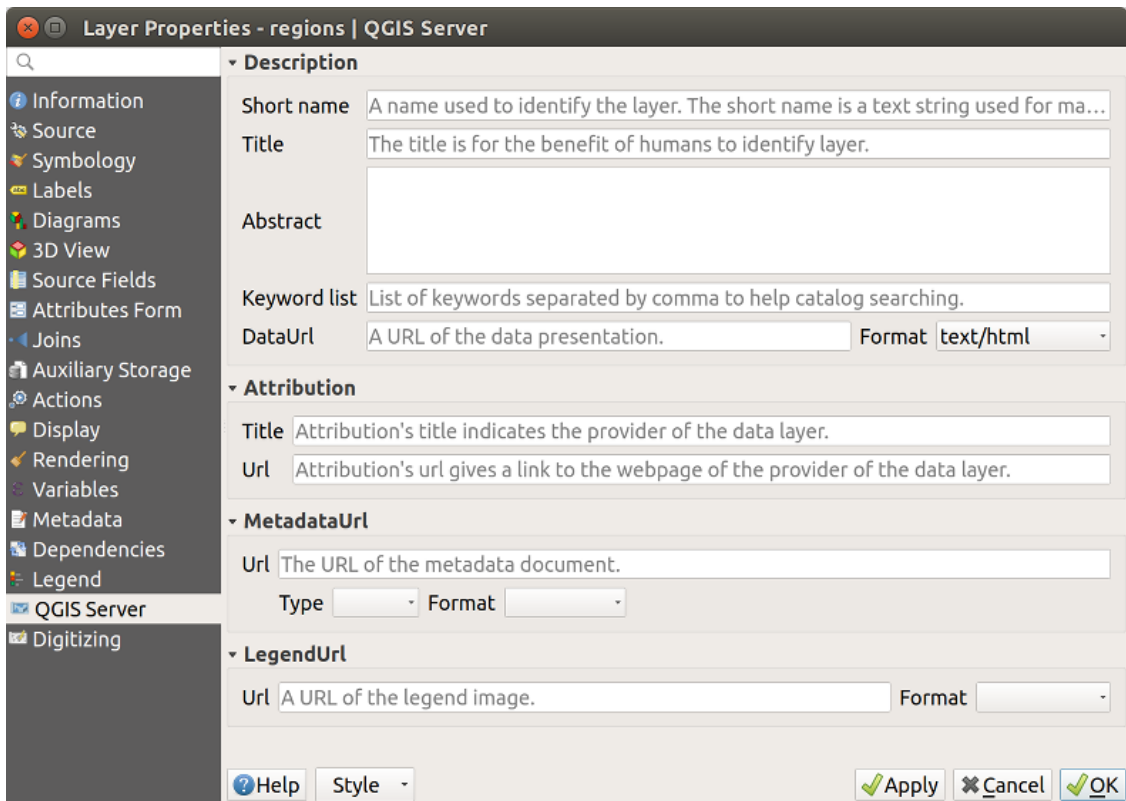



图 14.63: QGIS Server tab in vector layers properties dialog

To learn more about QGIS Server, read the QGIS-Server-manual.

14.1.20 Digitizing Properties

 The *Digitizing* tab gives access to options that help to ensure the quality of digitized geometries.

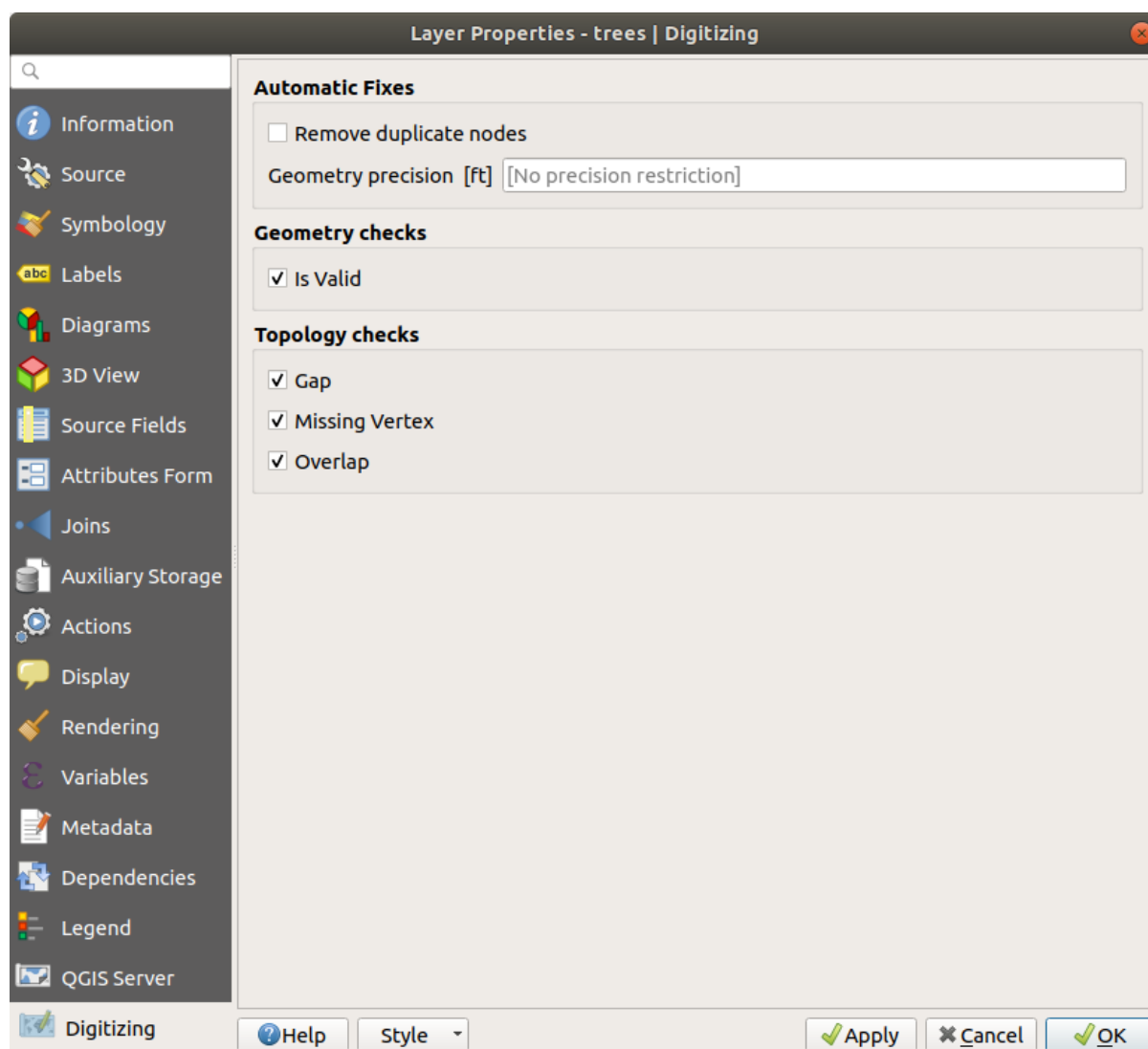


图 14.64: The QGIS Digitizing tab in the vector layers properties dialog

Automatic Fixes

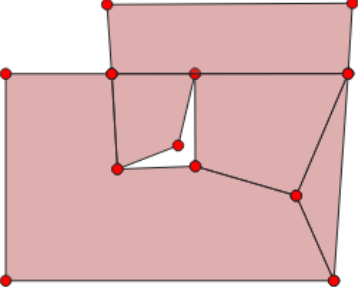
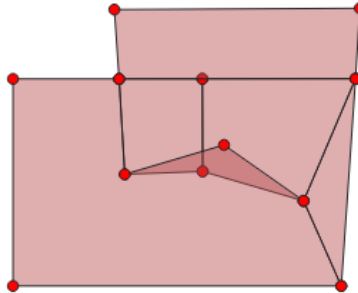
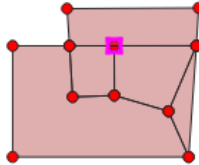
Options in the *Automatic Fixes* section will directly affect the vertices of any geometry which is added or modified. If the *Remove duplicate nodes* option is checked, any two subsequent vertices with exactly the same coordinates will be removed. If the *Geometry precision* is set, all vertices will be rounded to the closest multiple of the configured geometry precision. The rounding will happen in the layer coordinate reference system. Z and M values are not rounded. With many map tools, a grid is shown on the canvas while digitizing.

Geometry Checks

In the *Geometry checks* section, additional validations on a per geometry basis can be activated. Immediately after any geometry modification, failures in these checks are reported to the user in the geometry validation panel. As long as a check is failing, it is not possible to save the layer. The *Is valid* check will run basic validity checks like self intersection on geometries.

Topology Checks

In the *Topology checks* section, additional topology validation checks can be activated. Topology checks will be executed when the user saves the layer. Check errors will be reported in the geometry validation panel. As long as validation errors are present, the layer can not be saved. Topology checks are executed in the area of the bounding box of the modified features. Since other features may be present in the same area, topological errors concerning these features are reported as well as errors introduced in the current edit session.

Topology check option	Illustration
<p>The <input checked="" type="checkbox"/> <i>Gap</i> check will check for gaps between neighbouring polygons.</p>	
<p>The <input checked="" type="checkbox"/> <i>Overlap</i> check will check for overlaps between neighbouring polygons.</p>	
<p>The <input checked="" type="checkbox"/> <i>Missing vertex</i> check will check for shared boundaries of neighbouring polygons where one border misses a vertex which is present on the other one.</p>	

Gap check exceptions

Sometimes it is desirable to keep gaps inside an area in a polygon layer that otherwise is fully covered by polygons. For example, a land use layer may have acceptable holes for lakes. It is possible to define areas that are ignored in the gap check. Since gaps inside these areas are allowed, we will refer to them as *Allowed Gaps* areas.

In the options for the gap checks under *Allowed Gaps*, an *Allowed Gaps* layer can be configured.

Whenever the gap check is executed, gaps which are covered by one or more polygons in the *Allowed Gaps* Layer are not reported as topology errors.

It is also possible to configure an additional *Buffer*. This buffer is applied to each polygon on the *Allowed Gaps* Layer. This makes it possible to make the tests less susceptible to small changes in the outlines at the borders of gaps.

When *Allowed Gaps* are enabled, an additional button (*Add Allowed Gap*) for detected gap errors is available in the geometry validation dock, where gaps are reported during digitizing. If the *Add Allowed Gap* button is pushed, a new polygon with the geometry of the detected gap is inserted into the *Allowed Gaps* Layer. This makes it possible to quickly flag gaps as allowed.





14.2 Expressions

Based on layer data and prebuilt or user defined functions, **Expressions** offer a powerful way to manipulate attribute value, geometry and variables in order to dynamically change the geometry style, the content or position of the label, the value for diagram, the height of a layout item, select some features, create virtual field, ...

注解: A list of the default functions and variables for writing expressions can be found at [功能列表](#), with detailed information and examples.

14.2.1 The Expression string builder

Main dialog to build expressions, the *Expression string builder* is available from many parts in QGIS and, can particularly be accessed when:

- clicking the  button;
- *selecting features* with the  Select By Expression... tool;
- *editing attributes* with e.g. the  Field calculator tool;
- manipulating symbology, label or layout item parameters with the  Data defined override tool (see *Data defined override setup*);
- building a *geometry generator* symbol layer;
- doing some *geoprocessing*.

The Expression builder dialog offers access to the:

- *Expression tab* which, thanks to a list of predefined functions, helps to write and check the expression to use;
- *Function Editor tab* which helps to extend the list of functions by creating custom ones.

The Interface

The *Expression* tab provides the main interface to write expressions using functions, layer fields and values. It contains the following widgets:

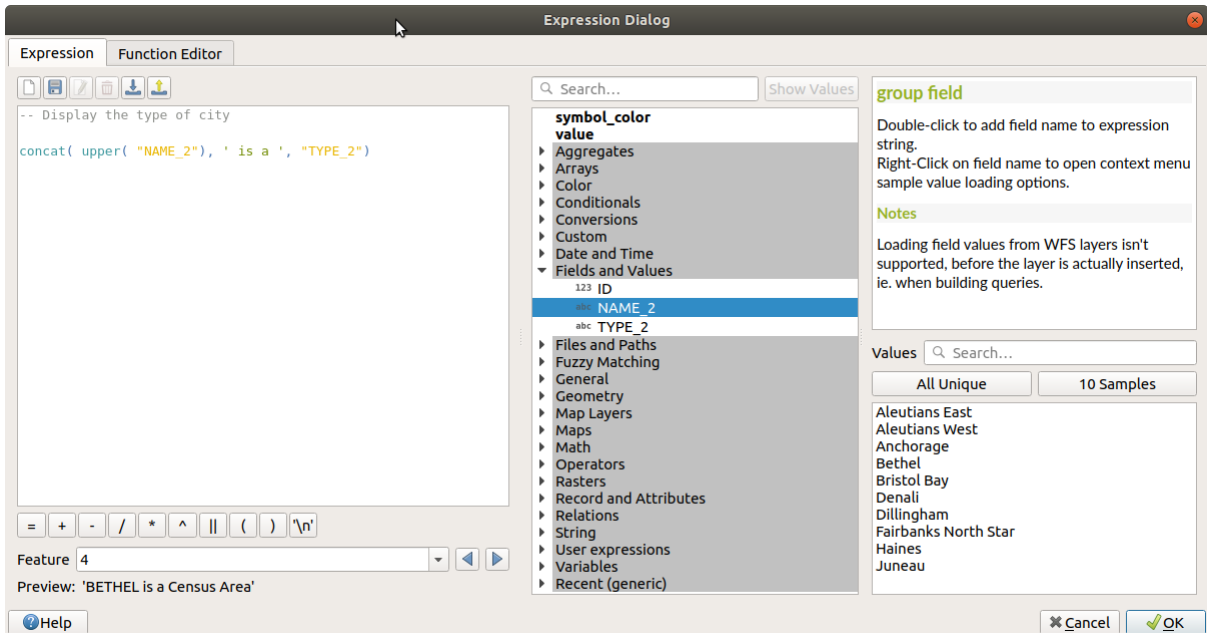


图 14.65: The Expression tab

- An expression editor area for typing or pasting expressions. Autocompletion is available to speed expression writing:

- Corresponding variables, function names and field names to the input text are shown below: use the Up and Down arrows to browse the items and press Tab to insert in the expression or simply click on the wished item.
- Function parameters are shown while filling them.

QGIS also checks the expression rightness and highlights all the errors using:

- *Underline*: for unknown functions, wrong or invalid arguments;
- *Marker*: for every other error (eg, missing parenthesis, unexpected character) at a single location.

小技巧: Document your expression with comments

When using complex expression, it is good practice to add text either as a multiline comment or inline comments to help you remember.

```

/*
Labels each region with its highest (in altitude) airport(s)
and altitude, eg 'AMBLER : 264m' for the 'Northwest Artic' region
*/
with_variable(
  'airport_alti', -- stores the highest altitude of the region
  aggregate(
    'airports',
    'max',
    "ELEV", -- the field containing the altitude
    -- and limit the airports to the region they are within
    filter := within( $geometry, geometry( @parent ) )
  ),

```


(下页继续)

```

aggregate( -- finds airports at the same altitude in the region
  'airports',
  'concatenate',
  "NAME",
  filter := within( $geometry, geometry( @parent ) )
    and "ELEV" = @airport_alti
  )
|| ' : ' || @airport_alti || 'm'
-- using || allows regions without airports to be skipped
)

```

- Above the expression editor, a set of tools helps you:

-  Clear the expression editor
- create and manage *user expressions*

- Under the expression editor, you find:

- a set of basic operators to help you build the expression
- an indication of the expected format of output when you are data-defining feature properties
- a live *Output preview* of the expression, evaluated on the first feature of the Layer by default. You can browse and evaluate other features of the layer using the *Feature* combobox (the values are taken from the *display name* property of the layer).

In case of error, it indicates it and you can access the details with the provided hyperlink.

- A function selector displays the list of functions, variables, fields... organized in groups. A search box is available to filter the list and quickly find a particular function or field. Double-clicking an item adds it to the expression editor.
- A help panel displays help for each selected item in the function selector.

小技巧: Press **Ctrl+Click** when hovering a function name in an expression to automatically display its help in the dialog.

A field's values widget shown when a field is selected in the function selector helps to fetch features attributes:

- Look for a particular field value
- Display the list of *All Unique* or *10 Samples* values. Also available from right-click.



When the field is mapped with another layer or a set of values, i.e. if the *field widget* is of *RelationReference*, *ValueRelation* or *ValueMap* type, it's possible to list all the values of the mapped field (from the referenced layer, table or list). Moreover, you can filter this list to *Only show values in use* in the current field.

Double-clicking a field value in the widget adds it to the expression editor.

小技巧: The right panel, showing functions help or field values, can be collapsed (invisible) in the dialog. Press the *Show Values* or *Show Help* button to get it back.

Writing an expression

QGIS expressions are used to select features or set values. Writing an expression in QGIS follows some rules:

1. **The dialog defines the context:** if you are used to SQL, you probably know queries of the type *select features from layer where condition* or *update layer set field = new_value where condition*. A QGIS expression also needs all these information but the tool you use to open the expression builder dialog provides parts of them. For example, giving a layer (building) with a field (height):
 - pressing the  Select by expression tool means that you want to "select features from buildings". The **condition** is the only information you need to provide in the expression text widget, e.g. type "height" > 20 to select buildings that are higher than 20.
 - with this selection made, pressing the  Field calculator button and choosing "height" as *Update existing field*, you already provide the command "update buildings set height = ??? where height > 20". The only remaining bits you have to provide in this case is the **new value**, e.g. just enter 50 to set the height of the previously selected buildings.
2. **Pay attention to quotes:** single quotes return a literal, so a text placed between single quotes ('145') is interpreted as a string. Double quotes will give you the value of that text so use them for fields ("myfield"). Fields can also be used without quotes (myfield). No quotes for numbers (3.16).

注解: Functions normally take as argument a string for field name. Do:

```
attribute( @atlas_feature, 'height' ) -- returns the value stored in the
↳ "height" attribute of the current atlas feature
```

And not:

```
attribute( @atlas_feature, "height" ) -- fetches the value of the attribute
↳ named "height" (e.g. 100), and use that value as a field
-- from which to return the atlas
↳ feature value. Probably wrong as a field named "100" may not exist.
```

小技巧: Use named parameters to ease expression reading

Some functions require many parameters to be set. The expression engine supports the use of named parameters. This means that instead of writing the cryptic expression `clamp(1, 2, 9)`, you can use `clamp(min:=1, value:=2, max:=9)`. This also allows arguments to be switched, e.g. `clamp(value:=2, max:=9, min:=1)`. Using named parameters helps clarify what the arguments for an expression function refer to, which is helpful when you are trying to interpret an expression later!

Some use cases of expressions

- From the Field Calculator, calculate a "pop_density" field using the existing "total_pop" and "area_km2" fields:

```
"total_pop" / "area_km2"
```

- Label or categorize features based on their area:

```
CASE WHEN $area > 10 000 THEN 'Larger' ELSE 'Smaller' END
```

- Update the field "density_level" with categories according to the "pop_density" values:

```
CASE WHEN "pop_density" < 50 THEN 'Low population density'
      WHEN "pop_density" >= 50 and "pop_density" < 150 THEN 'Medium population
↳ density'
```

(下页继续)

(续上页)

```
WHEN "pop_density" >= 150 THEN 'High population density'
END
```

- Apply a categorized style to all the features according to whether their average house price is smaller or higher than 10000€ per square metre:

```
"price_m2" > 10000
```

- Using the "Select By Expression..." tool, select all the features representing areas of "High population density" and whose average house price is higher than 10000€ per square metre:

```
"density_level" = 'High population density' and "price_m2" > 10000
```

The previous expression could also be used to define which features to label or show on the map.

- Create a different symbol (type) for the layer, using the geometry generator:

```
point_on_surface( $geometry )
```


- Given a point feature, generate a closed line (using `make_line`) around its geometry:

```
make_line(
  -- using an array of points placed around the original
  array_foreach(
    -- list of angles for placing the projected points (every 90°)
    array:=generate_series( 0, 360, 90 ),
    -- translate the point 20 units in the given direction (angle)
    expression:=project( $geometry, distance:=20, azimuth:=radians( @element ) )
  )
)
```





- In a print layout label, display the name of the "airports" features that are within the layout "Map 1" item:


```
with_variable( 'extent',
  map_get( item_variables( 'Map 1' ), 'map_extent' ),
  aggregate( 'airports', 'concatenate', "NAME",
    intersects( $geometry, @extent ), ' , '
  )
)
```

Saving Expressions

Using the  Add current expression to user expressions button above the expression editor frame, you can save important expressions you want to have quick access to. These are available from the **User expressions** group in the middle panel. They are saved under the user profile (<userprofile>/QGIS/QGIS3.ini file) and available in all expression dialogs inside all projects of the current user profile.

A set of tools available above the expression editor frame helps you manage the user expressions:

-  Add the current expression to user expressions: store the expression in the user profile. A label and a help text can be added for easy identification.
-  Edit selected expression from user expressions, as well as their help and label
-  Remove selected expression from user expressions
-  Import user expressions from a .json file into the active user profile folder

-  Export user expressions as a .json file; all the user expressions in the user profile QGIS3.ini file are shared

14.2.2 Function Editor

With the *Function Editor* tab, you are able to write your own functions in Python language. This provides a handy and comfortable way to address particular needs that would not be covered by the predefined functions.

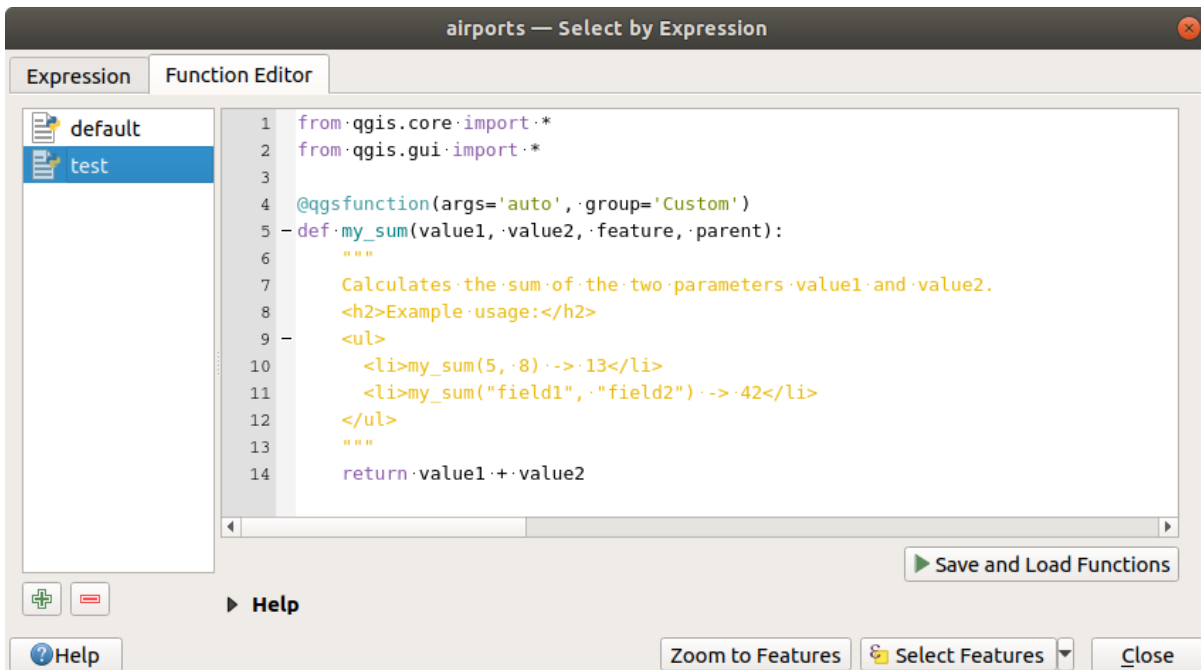





图 14.66: The Function Editor tab


To create a new function:

1. Press the  **New File** button.
2. Enter a name to use in the form that pops up and press *OK*.
A new item of the name you provide is added in the left panel of the *Function Editor* tab; this is a Python .py file based on QGIS template file and stored in the /python/expressions folder under the active *user profile* directory.
3. The right panel displays the content of the file: a python script template. Update the code and its help according to your needs.
4. Press the  *Save and Load Functions* button. The function you wrote is added to the functions tree in the *Expression* tab, by default under the *Custom* group.
5. Enjoy your new function.
6. If the function requires improvements, enable the *Function Editor* tab, do the changes and press again the  *Save and Load Functions* button to make them available in the file, hence in any expression tab.

Custom Python functions are stored under the user profile directory, meaning that at each QGIS startup, it will auto load all the functions defined with the current user profile. Be aware that new functions are only saved in the /python/expressions folder and not in the project file. If you share a project that uses one of your custom functions you will need to also share the .py file in the /python/expressions folder.

To delete a custom function:

1. Enable the *Function Editor* tab

2. Select the function in the list
3. Press the  Remove selected function. The function is removed from the list and the corresponding .py file deleted from the user profile folder.

Example

Here's a short example on how to create your own `my_sum` function that will operate with two values.

```

from qgis.core import *
from qgis.gui import *

@qgsfunction(args='auto', group='Custom')
def my_sum(value1, value2, feature, parent):
    """
    Calculates the sum of the two parameters value1 and value2.
    <h2>Example usage:</h2>
    <ul>
    <li>my_sum(5, 8) -> 13</li>
    <li>my_sum("field1", "field2") -> 42</li>
    </ul>
    """
    return value1 + value2
    
```

When using the `args='auto'` function argument the number of function arguments required will be calculated by the number of arguments the function has been defined with in Python (minus 2 - `feature`, and `parent`). The `group='Custom'` argument indicates the group in which the function should be listed in the Expression dialog.

It is also possible to add keywords arguments like:

- `usesgeometry=True` if the expression requires access to the features geometry. By default `False`.
- `handlesnull=True` if the expression has custom handling for NULL values. If `False` (default), the result will always be NULL as soon as any parameter is NULL.
- `referenced_columns=[list]`: An array of attribute names that are required to the function. Defaults to `[QgsFeatureRequest.ALL_ATTRIBUTES]`.

The previous example function can then be used in expressions:

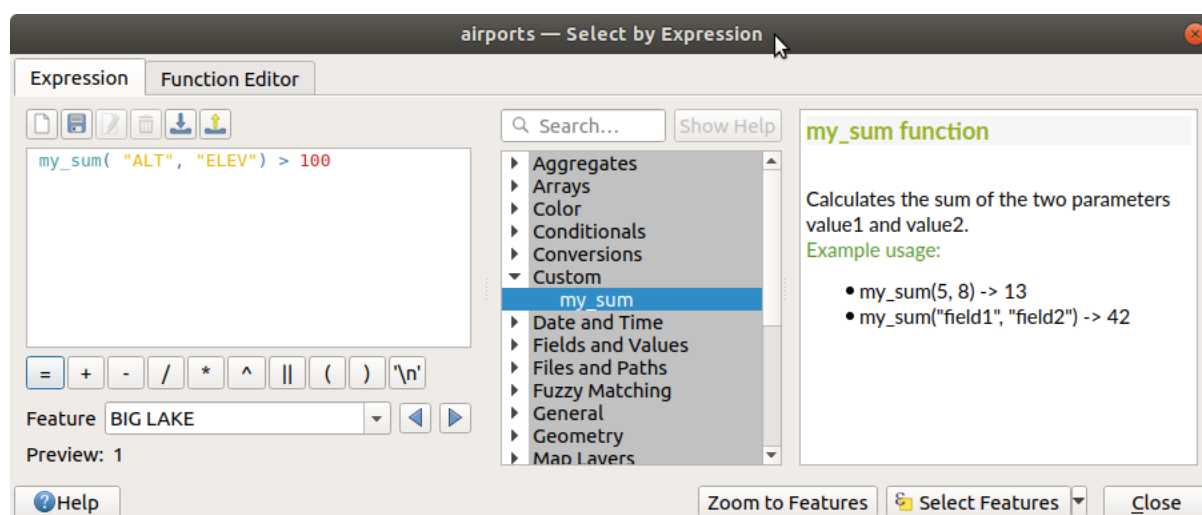


图 14.67: Custom Function added to the Expression tab

Further information about creating Python code can be found in the [PyQGIS-Developer-Cookbook](#).

14.3 功能列表

下面列出了 QGIS 中可用的函数、操作符和变量，按照功能分类

14.3.1 Aggregates Functions

This group contains functions which aggregate values over layers and fields.

- *aggregate*
- *array_agg*
- *collect*
- *concatenate*
- *concatenate_unique*
- *count*
- *count_distinct*
- *count_missing*
- *iqr*
- *majority*
- *max_length*
- *maximum*
- *mean*
- *median*
- *min_length*
- *minimum*
- *minority*
- *q1*
- *q3*
- *range*
- *relation_aggregate*
- *stdev*
- *sum*

aggregate

Returns an aggregate value calculated using features from another layer.

语法	<code>aggregate(layer, aggregate, expression, [filter], [concatenator=""], [order_by])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - a string, representing either a layer name or layer ID • aggregate - a string corresponding to the aggregate to calculate. Valid options are: <ul style="list-style-type: none"> - count - count_distinct - count_missing - 最小值 - max - sum - mean - median - stdev - stdevsample - range - minority - majority - q1: 四分之一分位数 - q3: 四分之三分位数 - iqr: inter quartile range - min_length: minimum string length - max_length: maximum string length - concatenate: join strings with a concatenator - concatenate_unique: join unique strings with a concatenator - collect: create an aggregated multipart geometry - array_agg: create an array of aggregated values • expression - sub expression or field name to aggregate • filter - optional filter expression to limit the features used for calculating the aggregate. Fields and geometry are from the features on the joined layer. The source feature can be accessed with the variable @parent. • concatenator - optional string to use to join values for 'concatenate' aggregate • order_by - optional filter expression to order the features used for calculating the aggregate. Fields and geometry are from the features on the joined layer. By default, the features will be returned in an unspecified order.
Examples	<ul style="list-style-type: none"> • <code>aggregate(layer:='rail_stations', aggregate:='sum', expression:="passengers")</code> → sum of all values from the passengers field in the rail_stations layer • <code>aggregate('rail_stations', 'sum', "passengers"/7)</code> → calculates a daily average of "passengers" by dividing the "passengers" field by 7 before summing the values • <code>aggregate(layer:='rail_stations', aggregate:='sum', expression:="passengers", filter:="class">3)</code> → sums up all values from the "passengers" field from features where the "class" attribute is greater than 3 only • <code>aggregate(layer:='rail_stations', aggregate:='concatenate', expression:="name", concatenator:=',')</code> → comma separated list of the name field for all features in the rail_stations layer • <code>aggregate(layer:='countries', aggregate:='max', expression:="code", filter:=intersects(\$geometry, geometry(@parent)))</code> → The country code of an intersecting country on the layer 'countries' • <code>aggregate(layer:='rail_stations', aggregate:='sum', expression:="passengers", filter:=contains(@atlas_geometry, \$geometry))</code> → sum of all values from the passengers field in the rail_stations within the current atlas feature • <code>aggregate(layer:='rail_stations', aggregate:='collect', expression:=centroid(\$geometry), filter:="region_name" = attribute(@parent, 'name'))</code> → aggregates rail_stations of the same region as current feature
334	Chapter 14: 使用矢量数据

array_agg

Returns an array of aggregated values from a field or expression.

语法	array_agg(expression, [group_by], [filter], [order_by]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate • order_by - optional expression to use to order features used to calculate aggregate. By default, the features will be returned in an unspecified order.
Examples	<ul style="list-style-type: none"> • array_agg("name", group_by:="state") → list of name values, grouped by state field

collect

Returns the multipart geometry of aggregated geometries from an expression

语法	collect(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - geometry expression to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • collect(\$geometry) → multipart geometry of aggregated geometries • collect(centroid(\$geometry), group_by:="region", filter:= "use" = 'civilian') → aggregated centroids of the civilian features based on their region value

concatenate

Returns all aggregated strings from a field or expression joined by a delimiter.

语法	concatenate(expression, [group_by], [filter], [concatenator], [order_by]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate • concatenator - optional string to use to join values. Empty by default. • order_by - optional expression to use to order features used to calculate aggregate. By default, the features will be returned in an unspecified order.
Examples	<ul style="list-style-type: none"> • concatenate("town_name", group_by:="state", concatenator:=',') → comma separated list of town_names, grouped by state field

concatenate_unique

Returns all unique strings from a field or expression joined by a delimiter.

语法	concatenate_unique(expression, [group_by], [filter], [concatenator], [order_by]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate • concatenator - optional string to use to join values. Empty by default. • order_by - optional expression to use to order features used to calculate aggregate. By default, the features will be returned in an unspecified order.
Examples	<ul style="list-style-type: none"> • concatenate_unique("town_name", group_by:="state", concatenator:=', ') → comma separated list of unique town_names, grouped by state field

count

Returns the count of matching features.

语法	count(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • count("stations", group_by:="state") → count of stations, grouped by state field

count_distinct

Returns the count of distinct values.

语法	count_distinct(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • count_distinct("stations", group_by:="state") → count of distinct stations values, grouped by state field

count_missing

Returns the count of missing (NULL) values.

语法	count_missing(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • count_missing("stations", group_by:="state") → count of missing (NULL) station values, grouped by state field

iqr

Returns the calculated inter quartile range from a field or expression.

语法	iqr(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • iqr("population", group_by:="state") → inter quartile range of population value, grouped by state field

majority

Returns the aggregate majority of values (most commonly occurring value) from a field or expression.

语法	majority(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • majority("class", group_by:="state") → most commonly occurring class value, grouped by state field

max_length

Returns the maximum length of strings from a field or expression.

语法	max_length(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • max_length("town_name", group_by:="state") → maximum length of town_name, grouped by state field

maximum

Returns the aggregate maximum value from a field or expression.

语法	maximum(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • maximum("population", group_by:="state") → maximum population value, grouped by state field

mean

Returns the aggregate mean value from a field or expression.

语法	mean(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • mean("population", group_by:="state") → mean population value, grouped by state field

median

Returns the aggregate median value from a field or expression.

语法	median(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • median("population", group_by:="state") → median population value, grouped by state field

min_length

Returns the minimum length of strings from a field or expression.

语法	min_length(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • min_length("town_name", group_by:="state") → minimum length of town_name, grouped by state field

minimum

Returns the aggregate minimum value from a field or expression.

语法	minimum(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • minimum("population", group_by:="state") → minimum population value, grouped by state field

minority

Returns the aggregate minority of values (least occurring value) from a field or expression.

语法	minority(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • <code>minority("class", group_by:="state")</code> → least occurring class value, grouped by state field

q1

Returns the calculated first quartile from a field or expression.

语法	q1(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • <code>q1("population", group_by:="state")</code> → first quartile of population value, grouped by state field

q3

Returns the calculated third quartile from a field or expression.

语法	q3(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • <code>q3("population", group_by:="state")</code> → third quartile of population value, grouped by state field

range

Returns the aggregate range of values (maximum - minimum) from a field or expression.

语法	range(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • range("population", group_by:="state") → range of population values, grouped by state field

relation_aggregate

Returns an aggregate value calculated using all matching child features from a layer relation.

语法	<code>relation_aggregate(relation, aggregate, expression, [concatenator=""], [order_by])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • relation - a string, representing a relation ID • aggregate - a string corresponding to the aggregate to calculate. Valid options are: <ul style="list-style-type: none"> - count - count_distinct - count_missing - 最小值 - max - sum - mean - median - stdev - stdevsample - range - minority - majority - q1: 四分之一分位数 - q3: 四分之三分位数 - iqr: inter quartile range - min_length: minimum string length - max_length: maximum string length - concatenate: join strings with a concatenator - concatenate_unique: join unique strings with a concatenator - collect: create an aggregated multipart geometry - array_agg: create an array of aggregated values • expression - sub expression or field name to aggregate • concatenator - optional string to use to join values for 'concatenate' aggregate • order_by - optional expression to order the features used for calculating the aggregate. Fields and geometry are from the features on the joined layer. By default, the features will be returned in an unspecified order.
Examples	<ul style="list-style-type: none"> • <code>relation_aggregate(relation:='my_relation', aggregate:='mean', expression:="passengers")</code> → mean value of all matching child features using the 'my_relation' relation • <code>relation_aggregate('my_relation', 'sum', "passengers"/7)</code> → sum of the passengers field divided by 7 for all matching child features using the 'my_relation' relation • <code>relation_aggregate('my_relation', 'concatenate', "towns", concatenator:=',')</code> → comma separated list of the towns field for all matching child features using the 'my_relation' relation • <code>relation_aggregate('my_relation', 'array_agg', "id")</code> → array of the id field from all matching child features using the 'my_relation' relation

Further reading: [Creating one or many to many relations](#)

stdev

Returns the aggregate standard deviation value from a field or expression.

语法	stdev(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • stdev("population", group_by:="state") → standard deviation of population value, grouped by state field

sum

Returns the aggregate summed value from a field or expression.

语法	sum(expression, [group_by], [filter]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - sub expression of field to aggregate • group_by - optional expression to use to group aggregate calculations • filter - optional expression to use to filter features used to calculate aggregate
Examples	<ul style="list-style-type: none"> • sum("population", group_by:="state") → summed population value, grouped by state field

14.3.2 Array Functions

This group contains functions to create and manipulate arrays (also known as list data structures). The order of values within the array matters, unlike the *'map' data structure*, where the order of key-value pairs is irrelevant and values are identified by their keys.

- *array*
- *array_all*
- *array_append*
- *array_cat*
- *array_contains*
- *array_distinct*
- *array_filter*
- *array_find*
- *array_first*
- *array_foreach*
- *array_get*

- *array_insert*
- *array_intersect*
- *array_last*
- *array_length*
- *array_prepend*
- *array_remove_all*
- *array_remove_at*
- *array_reverse*
- *array_slice*
- *array_sort*
- *array_to_string*
- *generate_series*
- *regexp_matches*
- *string_to_array*

array

Returns an array containing all the values passed as parameter.

语法	array(value1, value2, ...)
参数	<ul style="list-style-type: none"> • value - a value
Examples	<ul style="list-style-type: none"> • array(2, 10) → [2, 10]

array_all

Returns true if an array contains all the values of a given array.

语法	array_all(array_a, array_b)
参数	<ul style="list-style-type: none"> • array_a - an array • array_b - the array of values to search
Examples	<ul style="list-style-type: none"> • array_all(array(1, 2, 3), array(2, 3)) → true • array_all(array(1, 2, 3), array(1, 2, 4)) → false

array_append

Returns an array with the given value added at the end.

语法	array_append(array, value)
参数	<ul style="list-style-type: none"> • array - an array • value - the value to add
Examples	<ul style="list-style-type: none"> • array_append(array(1, 2, 3), 4) → [1, 2, 3, 4]

array_cat

Returns an array containing all the given arrays concatenated.

语法	array_cat(array1, array2, ...)
参数	<ul style="list-style-type: none"> • array - an array
Examples	<ul style="list-style-type: none"> • array_cat(array(1, 2), array(2, 3)) → [1, 2, 2, 3]

array_contains

Returns true if an array contains the given value.

语法	array_contains(array, value)
参数	<ul style="list-style-type: none"> • array - an array • value - the value to search
Examples	<ul style="list-style-type: none"> • array_contains(array(1, 2, 3), 2) → true

array_distinct

Returns an array containing distinct values of the given array.

语法	array_distinct(array)
参数	<ul style="list-style-type: none"> • array - an array
Examples	<ul style="list-style-type: none"> • array_distinct(array(1, 2, 3, 2, 1)) → [1, 2, 3]

array_filter

Returns an array with only the items for which the expression evaluates to true.

语法	array_filter(array, expression)
参数	<ul style="list-style-type: none"> • array - an array • expression - an expression to evaluate on each item. The variable <i>@element</i> will be replaced by the current value.
Examples	<ul style="list-style-type: none"> • array_filter(array(1, 2, 3), @element < 3) → [1, 2]

array_find

Returns the index (0 for the first one) of a value within an array. Returns -1 if the value is not found.

语法	array_find(array, value)
参数	<ul style="list-style-type: none"> • array - an array • value - the value to search
Examples	<ul style="list-style-type: none"> • array_find(array(1, 2, 3), 2) → 1

array_first

Returns the first value of an array.

语法	array_first(array)
参数	<ul style="list-style-type: none"> • array - an array
Examples	<ul style="list-style-type: none"> • array_first(array('a', 'b', 'c')) → 'a'

array_foreach

Returns an array with the given expression evaluated on each item.

语法	array_foreach(array, expression)
参数	<ul style="list-style-type: none"> • array - an array • expression - an expression to evaluate on each item. The variable <i>@element</i> will be replaced by the current value.
Examples	<ul style="list-style-type: none"> • array_foreach(array('a', 'b', 'c'), upper(@element)) → ['A', 'B', 'C'] • array_foreach(array(1, 2, 3), @element + 10) → [11, 12, 13]

array_get

Returns the Nth value (0 for the first one) of an array.

语法	array_get(array, index)
参数	<ul style="list-style-type: none"> • array - an array • index - the index to get (0 based)
Examples	<ul style="list-style-type: none"> • array_get(array('a', 'b', 'c'), 1) → 'b'

array_insert

Returns an array with the given value added at the given position.

语法	array_insert(array, pos, value)
参数	<ul style="list-style-type: none"> • array - an array • pos - the position where to add (0 based) • value - the value to add
Examples	<ul style="list-style-type: none"> • array_insert(array(1, 2, 3), 1, 100) → [1, 100, 2, 3]

array_intersect

Returns true if at least one element of array1 exists in array2.

语法	array_intersect(array1, array2)
参数	<ul style="list-style-type: none"> • array1 - an array • array2 - another array
Examples	<ul style="list-style-type: none"> • array_intersect(array(1, 2, 3, 4), array(4, 0, 2, 5)) → true

array_last

Returns the last value of an array.

语法	array_last(array)
参数	<ul style="list-style-type: none"> • array - an array
Examples	<ul style="list-style-type: none"> • array_last(array('a', 'b', 'c')) → 'c'

array_length

Returns the number of elements of an array.

语法	array_length(array)
参数	<ul style="list-style-type: none"> • array - an array
Examples	<ul style="list-style-type: none"> • array_length(array(1, 2, 3)) → 3

array_prepend

Returns an array with the given value added at the beginning.

语法	array_prepend(array, value)
参数	<ul style="list-style-type: none"> • array - an array • value - the value to add
Examples	<ul style="list-style-type: none"> • array_prepend(array(1, 2, 3), 0) → [0, 1, 2, 3]

array_remove_all

Returns an array with all the entries of the given value removed.

语法	array_remove_all(array, value)
参数	<ul style="list-style-type: none"> • array - an array • value - the values to remove
Examples	<ul style="list-style-type: none"> • array_remove_all(array('a', 'b', 'c', 'b'), 'b') → ['a', 'c']

array_remove_at

Returns an array with the given index removed.

语法	array_remove_at(array, pos)
参数	<ul style="list-style-type: none"> • array - an array • pos - the position to remove (0 based)
Examples	<ul style="list-style-type: none"> • array_remove_at(array(1, 2, 3), 1) → [1, 3]

array_reverse

Returns the given array with array values in reversed order.

语法	array_reverse(array)
参数	<ul style="list-style-type: none"> • array - an array
Examples	<ul style="list-style-type: none"> • array_reverse(array(2, 4, 0, 10)) → [10, 0, 4, 2]

array_slice

Returns a portion of the array. The slice is defined by the start_pos and end_pos arguments.

语法	array_slice(array, start_pos, end_pos)
参数	<ul style="list-style-type: none"> • array - an array • start_pos - the index of the start position of the slice (0 based). The start_pos index is included in the slice. If you use a negative start_pos, the index is counted from the end of the list (-1 based). • end_pos - the index of the end position of the slice (0 based). The end_pos index is included in the slice. If you use a negative end_pos, the index is counted from the end of the list (-1 based).
Examples	<ul style="list-style-type: none"> • array_slice(array(1, 2, 3, 4, 5), 0, 3) → [1, 2, 3, 4] • array_slice(array(1, 2, 3, 4, 5), 0, -1) → [1, 2, 3, 4, 5] • array_slice(array(1, 2, 3, 4, 5), -5, -1) → [1, 2, 3, 4, 5] • array_slice(array(1, 2, 3, 4, 5), 0, 0) → [1] • array_slice(array(1, 2, 3, 4, 5), -2, -1) → [4, 5] • array_slice(array(1, 2, 3, 4, 5), -1, -1) → [5] • array_slice(array('Dufour', 'Valmiera', 'Chugiak', 'Brighton'), 1, 2) → ['Valmiera', 'Chugiak'] • array_slice(array('Dufour', 'Valmiera', 'Chugiak', 'Brighton'), -2, -1) → ['Chugiak', 'Brighton']

array_sort

Returns the provided array with its elements sorted.

语法	array_sort(array, [ascending=true]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • array - an array • ascending - set this parameter to false to sort the array in descending order
Examples	<ul style="list-style-type: none"> • array_sort(array(3, 2, 1)) → [1, 2, 3]

array_to_string

Concatenates array elements into a string separated by a delimiter and using optional string for empty values.

语法	array_to_string(array, [delimiter=','], [empty_value='']) [] marks optional arguments
参数	<ul style="list-style-type: none"> • array - the input array • delimiter - the string delimiter used to separate concatenated array elements • empty_value - the optional string to use as replacement for empty (zero length) matches
Examples	<ul style="list-style-type: none"> • array_to_string(array('1', '2', '3')) → '1,2,3' • array_to_string(array(1, 2, 3), '-') → '1-2-3' • array_to_string(array('1', '', '3'), ',', '0') → '1,0,3'

generate_series

Creates an array containing a sequence of numbers.

语法	generate_series(start, stop, [step=1]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • start - first value of the sequence • stop - value that ends the sequence once reached • step - value used as the increment between values
Examples	<ul style="list-style-type: none"> • generate_series(1, 5) → [1, 2, 3, 4, 5] • generate_series(5, 1, -1) → [5, 4, 3, 2, 1]

regexp_matches

Returns an array of all strings captured by capturing groups, in the order the groups themselves appear in the supplied regular expression against a string.

语法	regexp_matches(string, regex, [empty_value='']) [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - the string to capture groups from against the regular expression • regex - the regular expression used to capture groups • empty_value - the optional string to use as replacement for empty (zero length) matches
Examples	<ul style="list-style-type: none"> • regexp_matches('QGIS=>rocks', '(.*)=>(.*') → ['QGIS', 'rocks'] • regexp_matches('key=>', '(.*)=>(.*', 'empty value') → ['key', 'empty value']

string_to_array

Splits string into an array using supplied delimiter and optional string for empty values.

语法	string_to_array(string, [delimiter=','], [empty_value=""]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - the input string • delimiter - the string delimiter used to split the input string • empty_value - the optional string to use as replacement for empty (zero length) matches
Examples	<ul style="list-style-type: none"> • string_to_array('1,2,3','') → ['1', '2', '3'] • string_to_array('1,,3','','0') → ['1', '0', '3']

14.3.3 Color Functions

This group contains functions for manipulating colors.

- *color_cmyk*
- *color_cmyka*
- *color_grayscale_average*
- *color_hsl*
- *color_hsla*
- *color_hsv*
- *color_hsva*
- *color_mix_rgb*
- *color_part*
- *color_rgb*
- *color_rgba*
- *create_ramp*
- *darker*
- *lighter*
- *project_color*
- *ramp_color*
- *set_color_part*

color_cmyk

Returns a string representation of a color based on its cyan, magenta, yellow and black components

语法	color_cmyk(cyan, magenta, yellow, black)
参数	<ul style="list-style-type: none"> • cyan - cyan component of the color, as a percentage integer value from 0 to 100 • magenta - magenta component of the color, as a percentage integer value from 0 to 100 • yellow - yellow component of the color, as a percentage integer value from 0 to 100 • black - black component of the color, as a percentage integer value from 0 to 100
Examples	<ul style="list-style-type: none"> • color_cmyk(100, 50, 0, 10) → '0,115,230'

color_cmyka

Returns a string representation of a color based on its cyan, magenta, yellow, black and alpha (transparency) components

语法	color_cmyka(cyan, magenta, yellow, black, alpha)
参数	<ul style="list-style-type: none"> • cyan - cyan component of the color, as a percentage integer value from 0 to 100 • magenta - magenta component of the color, as a percentage integer value from 0 to 100 • yellow - yellow component of the color, as a percentage integer value from 0 to 100 • black - black component of the color, as a percentage integer value from 0 to 100 • alpha - alpha component as an integer value from 0 (completely transparent) to 255 (opaque).
Examples	<ul style="list-style-type: none"> • color_cmyk(100, 50, 0, 10, 200) → '0,115,230,200'

color_grayscale_average

Applies a grayscale filter and returns a string representation from a provided color.

语法	color_grayscale_average(color)
参数	<ul style="list-style-type: none"> • color - a color string
Examples	<ul style="list-style-type: none"> • color_grayscale_average('255, 100, 50') → '135,135,135,255'

color_hsl

Returns a string representation of a color based on its hue, saturation, and lightness attributes.

语法	color_hsl(hue, saturation, lightness)
参数	<ul style="list-style-type: none"> • hue - hue of the color, as an integer value from 0 to 360 • saturation - saturation percentage of the color as an integer value from 0 to 100 • lightness - lightness percentage of the color as an integer value from 0 to 100
Examples	<ul style="list-style-type: none"> • <code>color_hsl(100, 50, 70) → '166,217,140'</code>

color_hsla

Returns a string representation of a color based on its hue, saturation, lightness and alpha (transparency) attributes

语法	color_hsla(hue, saturation, lightness, alpha)
参数	<ul style="list-style-type: none"> • hue - hue of the color, as an integer value from 0 to 360 • saturation - saturation percentage of the color as an integer value from 0 to 100 • lightness - lightness percentage of the color as an integer value from 0 to 100 • alpha - alpha component as an integer value from 0 (completely transparent) to 255 (opaque).
Examples	<ul style="list-style-type: none"> • <code>color_hsla(100, 50, 70, 200) → '166,217,140,200'</code>

color_hsv

Returns a string representation of a color based on its hue, saturation, and value attributes.

语法	color_hsv(hue, saturation, value)
参数	<ul style="list-style-type: none"> • hue - hue of the color, as an integer value from 0 to 360 • saturation - saturation percentage of the color as an integer value from 0 to 100 • value - value percentage of the color as an integer from 0 to 100
Examples	<ul style="list-style-type: none"> • <code>color_hsv(40, 100, 100) → '255,170,0'</code>

color_hsva

Returns a string representation of a color based on its hue, saturation, value and alpha (transparency) attributes.

语法	color_hsva(hue, saturation, value, alpha)
参数	<ul style="list-style-type: none"> • hue - hue of the color, as an integer value from 0 to 360 • saturation - saturation percentage of the color as an integer value from 0 to 100 • value - value percentage of the color as an integer from 0 to 100 • alpha - alpha component as an integer value from 0 (completely transparent) to 255 (opaque)
Examples	<ul style="list-style-type: none"> • <code>color_hsva(40, 100, 100, 200) → '255,170,0,200'</code>

color_mix_rgb

Returns a string representing a color mixing the red, green, blue, and alpha values of two provided colors based on a given ratio.

语法	color_mix_rgb(color1, color2, ratio)
参数	<ul style="list-style-type: none"> • color1 - a color string • color2 - a color string • ratio - a ratio
Examples	<ul style="list-style-type: none"> • color_mix_rgb('0,0,0', '255,255,255', 0.5) → '127,127,127,255'

color_part

Returns a specific component from a color string, e.g., the red component or alpha component.

语法	color_part(color, component)
参数	<ul style="list-style-type: none"> • color - a color string • component - a string corresponding to the color component to return. Valid options are: <ul style="list-style-type: none"> - red: RGB red component (0-255) - green: RGB green component (0-255) - blue: RGB blue component (0-255) - alpha: alpha (transparency) value (0-255) - hue: HSV hue (0-360) - saturation: HSV saturation (0-100) - value: HSV value (0-100) - hsl_hue: HSL hue (0-360) - hsl_saturation: HSL saturation (0-100) - lightness: HSL lightness (0-100) - cyan: CMYK cyan component (0-100) - magenta: CMYK magenta component (0-100) - yellow: CMYK yellow component (0-100) - black: CMYK black component (0-100)
Examples	<ul style="list-style-type: none"> • color_part('200,10,30', 'green') → 10

color_rgb

Returns a string representation of a color based on its red, green, and blue components.

语法	color_rgb(red, green, blue)
参数	<ul style="list-style-type: none"> • red - red component as an integer value from 0 to 255 • green - green component as an integer value from 0 to 255 • blue - blue component as an integer value from 0 to 255
Examples	<ul style="list-style-type: none"> • color_rgb(255, 127, 0) → '255,127,0'

color_rgba

Returns a string representation of a color based on its red, green, blue, and alpha (transparency) components.

语法	<code>color_rgba(red, green, blue, alpha)</code>
参数	<ul style="list-style-type: none"> • red - red component as an integer value from 0 to 255 • green - green component as an integer value from 0 to 255 • blue - blue component as an integer value from 0 to 255 • alpha - alpha component as an integer value from 0 (completely transparent) to 255 (opaque).
Examples	<ul style="list-style-type: none"> • <code>color_rgba(255, 127, 0, 200)</code> → <code>'255,127,0,200'</code>

create_ramp

Returns a gradient ramp from a map of color strings and steps.

语法	<code>create_ramp(map, [discrete=false])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • map - a map of color strings and steps • discrete - set this parameter to true to create a discrete color ramp
Examples	<ul style="list-style-type: none"> • <code>ramp_color(create_ramp(map(0, '0,0,0', 1, '255,0,0')), 1)</code> → <code>'255,0,0,255'</code>

darker

Returns a darker (or lighter) color string

语法	<code>darker(color, factor)</code>
参数	<ul style="list-style-type: none"> • color - a color string • factor - an integer corresponding to the darkening factor: <ul style="list-style-type: none"> – if the factor is greater than 100, this function returns a darker color (e.g., setting factor to 200 returns a color that is half the brightness); – if the factor is less than 100, the return color is lighter, but using the <code>lighter()</code> function for this purpose is recommended; – if the factor is 0 or negative, the return value is unspecified.
Examples	<ul style="list-style-type: none"> • <code>darker('200,10,30', 200)</code> → <code>'100,5,15,255'</code>

Further reading: [lighter](#)

lighter

Returns a lighter (or darker) color string

语法	lighter(color, factor)
参数	<ul style="list-style-type: none"> • color - a color string • factor - an integer corresponding to the lightening factor: <ul style="list-style-type: none"> – if the factor is greater than 100, this function returns a lighter color (e.g., setting factor to 150 returns a color that is 50% brighter); – if the factor is less than 100, the return color is darker, but using the darker() function for this purpose is recommended; – if the factor is 0 or negative, the return value is unspecified.
Examples	<ul style="list-style-type: none"> • lighter('200,10,30', 200) → '255,158,168,255'

Further reading: [darker](#)

project_color

Returns a color from the project's color scheme.

语法	project_color(name)
参数	<ul style="list-style-type: none"> • name - a color name
Examples	<ul style="list-style-type: none"> • project_color('Logo color') → '20,140,50'

Further reading: [setting project colors](#)

ramp_color

Returns a string representing a color from a color ramp.

Saved ramp variant

Returns a string representing a color from a saved ramp

语法	ramp_color(ramp_name, value)
参数	<ul style="list-style-type: none"> • ramp_name - the name of the color ramp as a string, for example 'Spectral' • value - the position on the ramp to select the color from as a real number between 0 and 1
Examples	<ul style="list-style-type: none"> • ramp_color('Spectral', 0.3) → '253,190,115,255'

注解: The color ramps available vary between QGIS installations. This function may not give the expected results if you move your QGIS project between installations.

Expression-created ramp variant

Returns a string representing a color from an expression-created ramp

语法	ramp_color(ramp, value)
参数	<ul style="list-style-type: none"> • ramp - the color ramp • value - the position on the ramp to select the color from as a real number between 0 and 1
Examples	<ul style="list-style-type: none"> • ramp_color(create_ramp(map(0, '0,0,0', 1, '255,0,0')), 1) → '255,0,0,255'

Further reading: *Setting a Color Ramp*, *The color ramp drop-down shortcut*

set_color_part

Sets a specific color component for a color string, e.g., the red component or alpha component.

语法	set_color_part(color, component, value)
参数	<ul style="list-style-type: none"> • color - a color string • component - a string corresponding to the color component to set. Valid options are: <ul style="list-style-type: none"> – red: RGB red component (0-255) – green: RGB green component (0-255) – blue: RGB blue component (0-255) – alpha: alpha (transparency) value (0-255) – hue: HSV hue (0-360) – saturation: HSV saturation (0-100) – value: HSV value (0-100) – hsl_hue: HSL hue (0-360) – hsl_saturation: HSL saturation (0-100) – lightness: HSL lightness (0-100) – cyan: CMYK cyan component (0-100) – magenta: CMYK magenta component (0-100) – yellow: CMYK yellow component (0-100) – black: CMYK black component (0-100) • value - new value for color component, respecting the ranges listed above
Examples	<ul style="list-style-type: none"> • set_color_part('200,10,30', 'green', 50) → '200,50,30,255'

14.3.4 Conditional Functions

This group contains functions to handle conditional checks in expressions.

- *CASE*
- *coalesce*
- *if*
- *nullif*
- *regex_match*
- *try*

CASE

CASE is used to evaluate a series of conditions and return a result for the first condition met. The conditions are evaluated sequentially, and if a condition is true, the evaluation stops, and the corresponding result is returned. If none of the conditions are true, the value in the ELSE clause is returned. Furthermore, if no ELSE clause is set and none of the conditions are met, NULL is returned.

CASE

WHEN *condition* THEN *result*

[...n]

[ELSE *result*]

END

[] marks optional components

参数	<ul style="list-style-type: none"> • WHEN condition - A condition expression to evaluate • THEN result - If <i>condition</i> evaluates to True then <i>result</i> is evaluated and returned. • ELSE result - If none of the above conditions evaluated to True then <i>result</i> is evaluated and returned.
Examples	<ul style="list-style-type: none"> • CASE WHEN "name" IS NULL THEN 'None' END → Returns the string 'None' if the "name" field is NULL • CASE WHEN \$area > 10000 THEN 'Big property' WHEN \$area > 5000 THEN 'Medium property' ELSE 'Small property' END → Returns the string 'Big property' if the area is bigger than 10000, 'Medium property' if the area is between 5000 and 10000, and 'Small property' for others

coalesce

Returns the first non-NULL value from the expression list.

This function can take any number of arguments.

语法	coalesce(expression1, expression2, ...)
参数	<ul style="list-style-type: none"> • expression - any valid expression or value, regardless of type.
Examples	<ul style="list-style-type: none"> • coalesce(NULL, 2) → 2 • coalesce(NULL, 2, 3) → 2 • coalesce(7, NULL, 3*2) → 7 • coalesce("fieldA", "fallbackField", 'ERROR') → value of fieldA if it is non-NULL else the value of "fallbackField" or the string 'ERROR' if both are NULL

if

Tests a condition and returns a different result depending on the conditional check.

语法	if(condition, result_when_true, result_when_false)
参数	<ul style="list-style-type: none"> • condition - the condition which should be checked • result_when_true - the result which will be returned when the condition is true or another value that does not convert to false. • result_when_false - the result which will be returned when the condition is false or another value that converts to false like 0 or ". NULL will also be converted to false.
Examples	<ul style="list-style-type: none"> • if(1+1=2, 'Yes', 'No') → 'Yes' • if(1+1=3, 'Yes', 'No') → 'No' • if(5 > 3, 1, 0) → 1 • if('', 'It is true (not empty)', 'It is false (empty)') → 'It is false (empty)' • if(' ', 'It is true (not empty)', 'It is false (empty)') → 'It is true (not empty)' • if(0, 'One', 'Zero') → 'Zero' • if(10, 'One', 'Zero') → 'One'

nullif

Returns a NULL value if value1 equals value2; otherwise it returns value1. This can be used to conditionally substitute values with NULL.

语法	nullif(value1, value2)
参数	<ul style="list-style-type: none"> • value1 - The value that should either be used or substituted with NULL. • value2 - The control value that will trigger the NULL substitution.
Examples	<ul style="list-style-type: none"> • nullif(' (none) ', ' (none) ') → NULL • nullif('text', ' (none) ') → 'text' • nullif("name", ' ') → NULL, if name is an empty string (or already NULL), the name in any other case.

regexp_match

Return the first matching position matching a regular expression within an unicode string, or 0 if the substring is not found.

语法	regexp_match(input_string, regex)
参数	<ul style="list-style-type: none"> • input_string - the string to test against the regular expression • regex - The regular expression to test against. Backslash characters must be double escaped (e.g., "\s" to match a white space character or "\b" to match a word boundary).
Examples	<ul style="list-style-type: none"> • regexp_match('QGIS ROCKS', '\\sROCKS') → 5 • regexp_match('Budač', 'udač\\b') → 2

try

Tries an expression and returns its value if error-free. If the expression returns an error, an alternative value will be returned when provided otherwise the function will return NULL.

语法	try(expression, [alternative]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • expression - the expression which should be run • alternative - the result which will be returned if the expression returns an error.
Examples	<ul style="list-style-type: none"> • try(to_int('1'), 0) → 1 • try(to_int('a'), 0) → 0 • try(to_date('invalid_date')) → NULL

14.3.5 Conversions Functions

This group contains functions to convert one data type to another (e.g., string from/to integer, binary from/to string, string to date, ...).

<ul style="list-style-type: none"> • <i>from_base64</i> • <i>hash</i> • <i>md5</i> • <i>sha256</i> • <i>to_base64</i> • <i>to_date</i> • <i>to_datetime</i> • <i>to_decimal</i> • <i>to_dm</i> • <i>to_dms</i> • <i>to_int</i> • <i>to_interval</i> • <i>to_real</i> • <i>to_string</i> • <i>to_time</i>

from_base64

Decodes a string in the Base64 encoding into a binary value.

语法	from_base64(string)
参数	<ul style="list-style-type: none"> string - the string to decode
Examples	<ul style="list-style-type: none"> from_base64('U UdJUw==') → 'QGIS'

hash

Creates a hash from a string with a given method. One byte (8 bits) is represented with two hex "digits", so 'md4' (16 bytes) produces a 16 * 2 = 32 character long hex string and 'keccak_512' (64 bytes) produces a 64 * 2 = 128 character long hex string.

语法	hash(string, method)
参数	<ul style="list-style-type: none"> string - the string to hash method - The hash method among 'md4', 'md5', 'sha1', 'sha224', 'sha384', 'sha512', 'sha3_224', 'sha3_256', 'sha3_384', 'sha3_512', 'keccak_224', 'keccak_256', 'keccak_384', 'keccak_512'
Examples	<ul style="list-style-type: none"> hash('QGIS', 'md4') → 'c0fc71c241cdebb6e888cbac0e2b68eb' hash('QGIS', 'md5') → '57470aaa9e22adaefac7f5f342f1c6da' hash('QGIS', 'sha1') → 'f87cfb2b74cdd5867db913237024e7001e62b114' hash('QGIS', 'sha224') → '4093a619ada631c770f44bc643ead18fb393b93d6a6af1861fcfece0' hash('QGIS', 'sha256') → 'eb045cba7a797aaa06ac58830846e40c8e8c780bc0676d3393605fae50' hash('QGIS', 'sha384') → '91c1de03cc3d09fdd512e99f9dd9922efadc39ed21d3922e69a4305cc2' hash('QGIS', 'sha512') → 'c2c092f2ab743bf8edbebd028a745f30fc720408465ed369421f0a4e20' hash('QGIS', 'sha3_224') → '467f49a5039e7280d5d42fd433e80d203439e338eaabd701f0d6c170' hash('QGIS', 'sha3_256') → '540f7354b6b8a6e735f2845250f15f4f3ba4f666c55574d9e9354575' hash('QGIS', 'sha3_384') → '96052da1e77679e9a65f60d7ead961b287977823144786386eb4364' hash('QGIS', 'sha3_512') → '900d079dc69761da113980253aa8ac0414a8bd6d09879a916228f87' hash('QGIS', 'keccak_224') → '5b0ce6acef8b0a121d4ac4f3eaa8503c799ad4e26a3392d1fb2014' hash('QGIS', 'keccak_256') → '991c520aa6815392de24087f61b2ae0fd56abbfeee4a8ca019c101' hash('QGIS', 'keccak_384') → 'c57a3aed9d856fa04e5e99b62b6e027cca81ba574116d3cc1f0d'

md5

Creates a md5 hash from a string.

语法	md5(string)
参数	<ul style="list-style-type: none"> string - the string to hash
Examples	<ul style="list-style-type: none"> md5('QGIS') → '57470aaa9e22adaefac7f5f342f1c6da'

sha256

Creates a sha256 hash from a string.

语法	sha256(string)
参数	<ul style="list-style-type: none"> • string - the string to hash
Examples	<ul style="list-style-type: none"> • sha256('QGIS') → 'eb045cba7a797aaa06ac58830846e40c8e8c780bc0676d3393605fae50c05309'

to_base64

Encodes a binary value into a string, using the Base64 encoding.

语法	to_base64(value)
参数	<ul style="list-style-type: none"> • value - the binary value to encode
Examples	<ul style="list-style-type: none"> • to_base64('QGIS') → 'U UdJUw=='

to_date

Converts a string into a date object. An optional format string can be provided to parse the string; see [QDate::fromString](#) for additional documentation on the format.

语法	to_date(string, [format], [language]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - string representing a date value • format - format used to convert the string into a date • language - language (lowercase, two- or three-letter, ISO 639 language code) used to convert the string into a date
Examples	<ul style="list-style-type: none"> • to_date('2012-05-04') → 2012-05-04 • to_date('June 29, 2019', 'MMMM d, yyyy') → 2019-06-29 • to_date('29 juin, 2019', 'd MMMM, yyyy', 'fr') → 2019-06-29

to_datetime

Converts a string into a datetime object. An optional format string can be provided to parse the string; see [QDate::fromString](#) and [QTime::fromString](#) for additional documentation on the format.

语法	to_datetime(string, [format], [language]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - string representing a datetime value • format - format used to convert the string into a datetime • language - language (lowercase, two- or three-letter, ISO 639 language code) used to convert the string into a datetime
Examples	<ul style="list-style-type: none"> • to_datetime('2012-05-04 12:50:00') → 2012-05-04T12:50:00 • to_datetime('June 29, 2019 @ 12:34', 'MMMM d, yyyy @ HH:mm') → 2019-06-29T12:34 • to_datetime('29 juin, 2019 @ 12:34', 'd MMMM, yyyy @ HH:mm', 'fr') → 2019-06-29T12:34

to_decimal

Converts a degree, minute, second coordinate to its decimal equivalent.

语法	to_decimal(value)
参数	<ul style="list-style-type: none"> • value - A degree, minute, second string.
Examples	<ul style="list-style-type: none"> • to_decimal('6°21\'16.445') → 6.3545680555

to_dm

Converts a coordinate to degree, minute.

语法	to_dm(coordinate, axis, precision, [formatting=]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • coordinate - A latitude or longitude value. • axis - The axis of the coordinate. Either 'x' or 'y'. • precision - Number of decimals. • formatting - Designates the formatting type. Acceptable values are NULL (default), 'aligned' or 'suffix'.
Examples	<ul style="list-style-type: none"> • to_dm(6.1545681, 'x', 3) → 6°9.274' • to_dm(6.1545681, 'y', 4, 'aligned') → 6°09.2741'N • to_dm(6.1545681, 'y', 4, 'suffix') → 6°9.2741'N

to_dms

Converts a coordinate to degree, minute, second.

语法	to_dms(coordinate, axis, precision, [formatting=]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • coordinate - A latitude or longitude value. • axis - The axis of the coordinate. Either 'x' or 'y'. • precision - Number of decimals. • formatting - Designates the formatting type. Acceptable values are NULL (default), 'aligned' or 'suffix'.
Examples	<ul style="list-style-type: none"> • to_dms(6.1545681, 'x', 3) → 6°9'16.445" • to_dms(6.1545681, 'y', 4, 'aligned') → 6°09'16.4452"N • to_dms(6.1545681, 'y', 4, 'suffix') → 6°9'16.4452"N

to_int

Converts a string to integer number. Nothing is returned if a value cannot be converted to integer (e.g '123asd' is invalid).

语法	to_int(string)
参数	<ul style="list-style-type: none"> • string - string to convert to integer number
Examples	<ul style="list-style-type: none"> • to_int('123') → 123

to_interval

Converts a string to an interval type. Can be used to take days, hours, month, etc of a date.

语法	to_interval(string)
参数	<ul style="list-style-type: none"> • string - a string representing an interval. Allowable formats include {n} days {n} hours {n} months.
Examples	<ul style="list-style-type: none"> • to_interval('1 day 2 hours') → interval: 1.08333 days • to_interval('0.5 hours') → interval: 30 minutes • to_datetime('2012-05-05 12:00:00') - to_interval('1 day 2 hours') → 2012-05-04T10:00:00

to_real

Converts a string to a real number. Nothing is returned if a value cannot be converted to real (e.g '123.56asd' is invalid). Numbers are rounded after saving changes if the precision is smaller than the result of the conversion.

语法	to_real(string)
参数	<ul style="list-style-type: none"> • string - string to convert to real number
Examples	<ul style="list-style-type: none"> • to_real('123.45') → 123.45

to_string

Converts a number to string.

语法	to_string(number)
参数	<ul style="list-style-type: none"> • number - Integer or real value. The number to convert to string.
Examples	<ul style="list-style-type: none"> • to_string(123) → '123'

to_time

Converts a string into a time object. An optional format string can be provided to parse the string; see [QTime::fromString](#) for additional documentation on the format.

语法	to_time(string, [format], [language]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - string representing a time value • format - format used to convert the string into a time • language - language (lowercase, two- or three-letter, ISO 639 language code) used to convert the string into a time
Examples	<ul style="list-style-type: none"> • to_time('12:30:01') → 12:30:01 • to_time('12:34', 'HH:mm') → 12:34:00 • to_time('12:34', 'HH:mm', 'fr') → 12:34:00

14.3.6 Custom Functions

This group contains functions created by the user. See [Function Editor](#) for more details.

14.3.7 Date and Time Functions

This group contains functions for handling date and time data. This group shares several functions with the *Conversions Functions* (to_date, to_time, to_datetime, to_interval) and *String Functions* (format_date) groups.

注解: Storing date, datetime and intervals on fields

The ability to store *date*, *time* and *datetime* values directly on fields depends on the data source's provider (e.g., Shapefile accepts *date* format, but not *datetime* or *time* format). The following are some suggestions to overcome this limitation:

- *date*, *datetime* and *time* can be converted and stored in text type fields using the *format_date()* function.
 - *Intervals* can be stored in integer or decimal type fields after using one of the date extraction functions (e.g., *day()* to get the interval expressed in days)
-

- *age*
- *datetime_from_epoch*
- *day*
- *day_of_week*
- *epoch*
- *format_date*
- *hour*
- *make_date*
- *make_datetime*
- *make_interval*
- *make_time*
- *minute*
- *month*
- *now*
- *second*
- *to_date*
- *to_datetime*
- *to_interval*
- *to_time*
- *week*
- *year*

age

Returns the difference between two dates or datetimes.

The difference is returned as an `Interval` and needs to be used with one of the following functions in order to extract useful information:

- `year`
- `month`
- `week`
- `day`
- `hour`
- `minute`
- `second`

语法	<code>age(datetime1, datetime2)</code>
参数	<ul style="list-style-type: none"> • datetime1 - a string, date or datetime representing the later date • datetime2 - a string, date or datetime representing the earlier date
Examples	<ul style="list-style-type: none"> • <code>day(age('2012-05-12', '2012-05-02')) → 10</code> • <code>hour(age('2012-05-12', '2012-05-02')) → 240</code>

datetime_from_epoch

Returns a datetime whose date and time are the number of milliseconds, msec, that have passed since 1970-01-01T00:00:00.000, Coordinated Universal Time (Qt.UTC), and converted to Qt.LocalTime.

语法	<code>datetime_from_epoch(int)</code>
参数	<ul style="list-style-type: none"> • int - number (milliseconds)
Examples	<ul style="list-style-type: none"> • <code>datetime_from_epoch(1483225200000) → 2017-01-01T00:00:00</code>

day

Extracts the day from a date, or the number of days from an interval.

Date variant

Extracts the day from a date or datetime.

语法	<code>day(date)</code>
参数	<ul style="list-style-type: none"> • date - a date or datetime value
Examples	<ul style="list-style-type: none"> • <code>day('2012-05-12') → 12</code>

Interval variant

Calculates the length in days of an interval.

语法	day(interval)
参数	<ul style="list-style-type: none"> • interval - interval value to return number of days from
Examples	<ul style="list-style-type: none"> • <code>day(to_interval('3 days')) → 3</code> • <code>day(to_interval('3 weeks 2 days')) → 23</code> • <code>day(age('2012-01-01', '2010-01-01')) → 730</code>

day_of_week

Returns the day of the week for a specified date or datetime. The returned value ranges from 0 to 6, where 0 corresponds to a Sunday and 6 to a Saturday.

语法	day_of_week(date)
参数	<ul style="list-style-type: none"> • date - date or datetime value
Examples	<ul style="list-style-type: none"> • <code>day_of_week(to_date('2015-09-21')) → 1</code>

epoch

Returns the interval in milliseconds between the unix epoch and a given date value.

语法	epoch(date)
参数	<ul style="list-style-type: none"> • date - a date or datetime value
Examples	<ul style="list-style-type: none"> • <code>epoch(to_date('2017-01-01')) → 1483203600000</code>

format_date

Formats a date type or string into a custom string format. Uses Qt date/time format strings. See [QDateTime::toString](#).

语法	<pre>format_date(datetime, format, [language])</pre> <p>[] marks optional arguments</p>																																																
参数	<ul style="list-style-type: none"> • datetime - date, time or datetime value • format - String template used to format the string. <table border="1" data-bbox="459 387 1385 748"> <thead> <tr> <th>Expression</th> <th>Output</th> </tr> </thead> <tbody> <tr> <td>d</td> <td>the day as number without a leading zero (1 to 31)</td> </tr> <tr> <td>dd</td> <td>the day as number with a leading zero (01 to 31)</td> </tr> <tr> <td>ddd</td> <td>the abbreviated localized day name (e.g. 'Mon' to 'Sun')</td> </tr> <tr> <td>dddd</td> <td>the long localized day name (e.g. 'Monday' to 'Sunday')</td> </tr> <tr> <td>M</td> <td>the month as number without a leading zero (1-12)</td> </tr> <tr> <td>MM</td> <td>the month as number with a leading zero (01-12)</td> </tr> <tr> <td>MMM</td> <td>the abbreviated localized month name (e.g. 'Jan' to 'Dec')</td> </tr> <tr> <td>MMMM</td> <td>the long localized month name (e.g. 'January' to 'December')</td> </tr> <tr> <td>yy</td> <td>the year as two digit number (00-99)</td> </tr> <tr> <td>yyyy</td> <td>the year as four digit number</td> </tr> </tbody> </table> <p>These expressions may be used for the time part of the format string:</p> <table border="1" data-bbox="459 826 1385 1254"> <thead> <tr> <th>Expression</th> <th>Output</th> </tr> </thead> <tbody> <tr> <td>h</td> <td>the hour without a leading zero (0 to 23 or 1 to 12 if AM/PM display)</td> </tr> <tr> <td>hh</td> <td>the hour with a leading zero (00 to 23 or 01 to 12 if AM/PM display)</td> </tr> <tr> <td>H</td> <td>the hour without a leading zero (0 to 23, even with AM/PM display)</td> </tr> <tr> <td>HH</td> <td>the hour with a leading zero (00 to 23, even with AM/PM display)</td> </tr> <tr> <td>m</td> <td>the minute without a leading zero (0 to 59)</td> </tr> <tr> <td>mm</td> <td>the minute with a leading zero (00 to 59)</td> </tr> <tr> <td>s</td> <td>the second without a leading zero (0 to 59)</td> </tr> <tr> <td>ss</td> <td>the second with a leading zero (00 to 59)</td> </tr> <tr> <td>z</td> <td>the milliseconds without trailing zeroes (0 to 999)</td> </tr> <tr> <td>zzz</td> <td>the milliseconds with trailing zeroes (000 to 999)</td> </tr> <tr> <td>AP or A</td> <td>interpret as an AM/PM time. <i>AP</i> must be either 'AM' or 'PM'.</td> </tr> <tr> <td>ap or a</td> <td>Interpret as an AM/PM time. <i>ap</i> must be either 'am' or 'pm'.</td> </tr> </tbody> </table> <ul style="list-style-type: none"> • language - language (lowercase, two- or three-letter, ISO 639 language code) used to format the date into a custom string 	Expression	Output	d	the day as number without a leading zero (1 to 31)	dd	the day as number with a leading zero (01 to 31)	ddd	the abbreviated localized day name (e.g. 'Mon' to 'Sun')	dddd	the long localized day name (e.g. 'Monday' to 'Sunday')	M	the month as number without a leading zero (1-12)	MM	the month as number with a leading zero (01-12)	MMM	the abbreviated localized month name (e.g. 'Jan' to 'Dec')	MMMM	the long localized month name (e.g. 'January' to 'December')	yy	the year as two digit number (00-99)	yyyy	the year as four digit number	Expression	Output	h	the hour without a leading zero (0 to 23 or 1 to 12 if AM/PM display)	hh	the hour with a leading zero (00 to 23 or 01 to 12 if AM/PM display)	H	the hour without a leading zero (0 to 23, even with AM/PM display)	HH	the hour with a leading zero (00 to 23, even with AM/PM display)	m	the minute without a leading zero (0 to 59)	mm	the minute with a leading zero (00 to 59)	s	the second without a leading zero (0 to 59)	ss	the second with a leading zero (00 to 59)	z	the milliseconds without trailing zeroes (0 to 999)	zzz	the milliseconds with trailing zeroes (000 to 999)	AP or A	interpret as an AM/PM time. <i>AP</i> must be either 'AM' or 'PM'.	ap or a	Interpret as an AM/PM time. <i>ap</i> must be either 'am' or 'pm'.
Expression	Output																																																
d	the day as number without a leading zero (1 to 31)																																																
dd	the day as number with a leading zero (01 to 31)																																																
ddd	the abbreviated localized day name (e.g. 'Mon' to 'Sun')																																																
dddd	the long localized day name (e.g. 'Monday' to 'Sunday')																																																
M	the month as number without a leading zero (1-12)																																																
MM	the month as number with a leading zero (01-12)																																																
MMM	the abbreviated localized month name (e.g. 'Jan' to 'Dec')																																																
MMMM	the long localized month name (e.g. 'January' to 'December')																																																
yy	the year as two digit number (00-99)																																																
yyyy	the year as four digit number																																																
Expression	Output																																																
h	the hour without a leading zero (0 to 23 or 1 to 12 if AM/PM display)																																																
hh	the hour with a leading zero (00 to 23 or 01 to 12 if AM/PM display)																																																
H	the hour without a leading zero (0 to 23, even with AM/PM display)																																																
HH	the hour with a leading zero (00 to 23, even with AM/PM display)																																																
m	the minute without a leading zero (0 to 59)																																																
mm	the minute with a leading zero (00 to 59)																																																
s	the second without a leading zero (0 to 59)																																																
ss	the second with a leading zero (00 to 59)																																																
z	the milliseconds without trailing zeroes (0 to 999)																																																
zzz	the milliseconds with trailing zeroes (000 to 999)																																																
AP or A	interpret as an AM/PM time. <i>AP</i> must be either 'AM' or 'PM'.																																																
ap or a	Interpret as an AM/PM time. <i>ap</i> must be either 'am' or 'pm'.																																																
Examples	<ul style="list-style-type: none"> • <code>format_date('2012-05-15', 'dd.MM.yyyy')</code> → '15.05.2012' • <code>format_date('2012-05-15', 'd MMMM yyyy', 'fr')</code> → '15 mai 2012' • <code>format_date('2012-05-15', 'dddd')</code> → 'Tuesday' • <code>format_date('2012-05-15 13:54:20', 'dd.MM.yy')</code> → '15.05.12' • <code>format_date('13:54:20', 'hh:mm AP')</code> → '01:54 PM' 																																																

hour

Extracts the hour part from a datetime or time, or the number of hours from an interval.

Time variant

Extracts the hour part from a time or datetime.

语法	hour(datetime)
参数	<ul style="list-style-type: none"> • datetime - a time or datetime value
Examples	<ul style="list-style-type: none"> • hour(to_datetime('2012-07-22 13:24:57')) → 13

Interval variant

Calculates the length in hours of an interval.

语法	hour(interval)
参数	<ul style="list-style-type: none"> • interval - interval value to return number of hours from
Examples	<ul style="list-style-type: none"> • hour(to_interval('3 hours')) → 3 • hour(age('2012-07-22T13:00:00', '2012-07-22T10:00:00')) → 3 • hour(age('2012-01-01', '2010-01-01')) → 17520

make_date

Creates a date value from year, month and day numbers.

语法	make_date(year, month, day)
参数	<ul style="list-style-type: none"> • year - Year number. Years 1 to 99 are interpreted as is. Year 0 is invalid. • month - Month number, where 1=January • day - Day number, beginning with 1 for the first day in the month
Examples	<ul style="list-style-type: none"> • make_date(2020, 5, 4) → date value 2020-05-04

make_datetime

Creates a datetime value from year, month, day, hour, minute and second numbers.

语法	<code>make_datetime(year, month, day, hour, minute, second)</code>
参数	<ul style="list-style-type: none"> • year - Year number. Years 1 to 99 are interpreted as is. Year 0 is invalid. • month - Month number, where 1=January • day - Day number, beginning with 1 for the first day in the month • hour - Hour number • minute - Minutes • second - Seconds (fractional values include milliseconds)
Examples	<ul style="list-style-type: none"> • <code>make_datetime(2020, 5, 4, 13, 45, 30.5)</code> → datetime value 2020-05-04 13:45:30.500

make_interval

Creates an interval value from year, month, weeks, days, hours, minute and seconds values.

语法	<code>make_interval([years=0], [months=0], [weeks=0], [days=0], [hours=0], [minutes=0], [seconds=0])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • years - Number of years (assumes a 365.25 day year length). • months - Number of months (assumes a 30 day month length) • weeks - Number of weeks • days - Number of days • hours - Number of hours • minutes - Number of minutes • seconds - Number of seconds
Examples	<ul style="list-style-type: none"> • <code>make_interval(hours:=3)</code> → interval: 3 hours • <code>make_interval(days:=2, hours:=3)</code> → interval: 2.125 days • <code>make_interval(minutes:=0.5, seconds:=5)</code> → interval: 35 seconds

make_time

Creates a time value from hour, minute and second numbers.

语法	<code>make_time(hour, minute, second)</code>
参数	<ul style="list-style-type: none"> • hour - Hour number • minute - Minutes • second - Seconds (fractional values include milliseconds)
Examples	<ul style="list-style-type: none"> • <code>make_time(13, 45, 30.5)</code> → time value 13:45:30.500

minute

Extracts the minutes part from a datetime or time, or the number of minutes from an interval.

Time variant

Extracts the minutes part from a time or datetime.

语法	minute(datetime)
参数	<ul style="list-style-type: none"> • datetime - a time or datetime value
Examples	<ul style="list-style-type: none"> • <code>minute(to_datetime('2012-07-22 13:24:57')) → 24</code>

Interval variant

Calculates the length in minutes of an interval.

语法	minute(interval)
参数	<ul style="list-style-type: none"> • interval - interval value to return number of minutes from
Examples	<ul style="list-style-type: none"> • <code>minute(to_interval('3 minutes')) → 3</code> • <code>minute(age('2012-07-22T00:20:00', '2012-07-22T00:00:00')) → 20</code> • <code>minute(age('2012-01-01', '2010-01-01')) → 1051200</code>

month

Extracts the month part from a date, or the number of months from an interval.

Date variant

Extracts the month part from a date or datetime.

语法	month(date)
参数	<ul style="list-style-type: none"> • date - a date or datetime value
Examples	<ul style="list-style-type: none"> • <code>month('2012-05-12') → 05</code>

Interval variant

Calculates the length in months of an interval.

语法	month(interval)
参数	<ul style="list-style-type: none"> • interval - interval value to return number of months from
Examples	<ul style="list-style-type: none"> • <code>month(to_interval('3 months')) → 3</code> • <code>month(age('2012-01-01', '2010-01-01')) → 4.03333</code>

now

Returns the current date and time. The function is static and will return consistent results while evaluating. The time returned is the time when the expression is prepared.

语法	now()
Examples	<ul style="list-style-type: none"> • <code>now()</code> → 2012-07-22T13:24:57

second

Extracts the seconds part from a datetime or time, or the number of seconds from an interval.

Time variant

Extracts the seconds part from a time or datetime.

语法	second(datetime)
参数	<ul style="list-style-type: none"> • datetime - a time or datetime value
Examples	<ul style="list-style-type: none"> • <code>second(to_datetime('2012-07-22 13:24:57'))</code> → 57

Interval variant

Calculates the length in seconds of an interval.

语法	second(interval)
参数	<ul style="list-style-type: none"> • interval - interval value to return number of seconds from
Examples	<ul style="list-style-type: none"> • <code>second(to_interval('3 minutes'))</code> → 180 • <code>second(age('2012-07-22T00:20:00', '2012-07-22T00:00:00'))</code> → 1200 • <code>second(age('2012-01-01', '2010-01-01'))</code> → 63072000

to_date

Converts a string into a date object. An optional format string can be provided to parse the string; see [QDate::fromString](#) for additional documentation on the format.

语法	to_date(string, [format], [language]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - string representing a date value • format - format used to convert the string into a date • language - language (lowercase, two- or three-letter, ISO 639 language code) used to convert the string into a date
Examples	<ul style="list-style-type: none"> • <code>to_date('2012-05-04')</code> → 2012-05-04 • <code>to_date('June 29, 2019', 'MMMM d, yyyy')</code> → 2019-06-29 • <code>to_date('29 juin, 2019', 'd MMMM, yyyy', 'fr')</code> → 2019-06-29

to_datetime

Converts a string into a datetime object. An optional format string can be provided to parse the string; see `QDate::fromString` and `QTime::fromString` for additional documentation on the format.

语法	<code>to_datetime(string, [format], [language])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - string representing a datetime value • format - format used to convert the string into a datetime • language - language (lowercase, two- or three-letter, ISO 639 language code) used to convert the string into a datetime
Examples	<ul style="list-style-type: none"> • <code>to_datetime('2012-05-04 12:50:00')</code> → 2012-05-04T12:50:00 • <code>to_datetime('June 29, 2019 @ 12:34', 'MMMM d, yyyy @ HH:mm')</code> → 2019-06-29T12:34 • <code>to_datetime('29 juin, 2019 @ 12:34', 'd MMMM, yyyy @ HH:mm', 'fr')</code> → 2019-06-29T12:34

to_interval

Converts a string to an interval type. Can be used to take days, hours, month, etc of a date.

语法	<code>to_interval(string)</code>
参数	<ul style="list-style-type: none"> • string - a string representing an interval. Allowable formats include {n} days {n} hours {n} months.
Examples	<ul style="list-style-type: none"> • <code>to_interval('1 day 2 hours')</code> → interval: 1.08333 days • <code>to_interval('0.5 hours')</code> → interval: 30 minutes • <code>to_datetime('2012-05-05 12:00:00') - to_datetime('2012-05-04 12:00:00')</code> → 2012-05-04T10:00:00

to_time

Converts a string into a time object. An optional format string can be provided to parse the string; see `QTime::fromString` for additional documentation on the format.

语法	<code>to_time(string, [format], [language])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - string representing a time value • format - format used to convert the string into a time • language - language (lowercase, two- or three-letter, ISO 639 language code) used to convert the string into a time
Examples	<ul style="list-style-type: none"> • <code>to_time('12:30:01')</code> → 12:30:01 • <code>to_time('12:34', 'HH:mm')</code> → 12:34:00 • <code>to_time('12:34', 'HH:mm', 'fr')</code> → 12:34:00

week

Extracts the week number from a date, or the number of weeks from an interval.

Date variant

Extracts the week number from a date or datetime.

语法	week(date)
参数	<ul style="list-style-type: none"> • date - a date or datetime value
Examples	<ul style="list-style-type: none"> • week('2012-05-12') → 19

Interval variant

Calculates the length in weeks of an interval.

语法	week(interval)
参数	<ul style="list-style-type: none"> • interval - interval value to return number of months from
Examples	<ul style="list-style-type: none"> • week(to_interval('3 weeks')) → 3 • week(age('2012-01-01', '2010-01-01')) → 104.285

year

Extracts the year part from a date, or the number of years from an interval.

Date variant

Extracts the year part from a date or datetime.

语法	year(date)
参数	<ul style="list-style-type: none"> • date - a date or datetime value
Examples	<ul style="list-style-type: none"> • year('2012-05-12') → 2012

Interval variant

Calculates the length in years of an interval.

语法	year(interval)
参数	<ul style="list-style-type: none"> • interval - interval value to return number of years from
Examples	<ul style="list-style-type: none"> • year(to_interval('3 years')) → 3 • year(age('2012-01-01', '2010-01-01')) → 1.9986

Some examples:

Besides these functions, subtracting dates, datetimes or times using the – (minus) operator will return an interval.

Adding or subtracting an interval to dates, datetimes or times, using the + (plus) and - (minus) operators, will return a datetime.

- Get the number of days until QGIS 3.0 release:

```
to_date('2017-09-29') - to_date(now())  
-- Returns <interval: 203 days>
```

- The same with time:

```
to_datetime('2017-09-29 12:00:00') - now()  
-- Returns <interval: 202.49 days>
```

- Get the datetime of 100 days from now:

```
now() + to_interval('100 days')  
-- Returns <datetime: 2017-06-18 01:00:00>
```

14.3.8 Fields and Values

Contains a list of fields from the layer.

Double-click a field name to have it added to your expression. You can also type the field name (preferably inside double quotes) or its *alias*.

To retrieve fields values to use in an expression, select the appropriate field and, in the shown widget, choose between *10 Samples* and *All Unique*. Requested values are then displayed and you can use the *Search* box at the top of the list to filter the result. Sample values can also be accessed via right-clicking on a field.

To add a value to the expression you are writing, double-click on it in the list. If the value is of a string type, it should be simple quoted, otherwise no quote is needed.

14.3.9 Files and Paths Functions

This group contains functions which manipulate file and path names.

- *base_file_name*
- *file_exists*
- *file_name*
- *file_path*
- *file_size*
- *file_suffix*
- *is_directory*
- *is_file*

base_file_name

Returns the base name of the file without the directory or file suffix.

语法	base_file_name(path)
参数	<ul style="list-style-type: none"> • path - a file path
Examples	<ul style="list-style-type: none"> • <code>base_file_name('/home/qgis/data/country_boundaries.shp')</code> → 'country_boundaries'

file_exists

Returns true if a file path exists.

语法	file_exists(path)
参数	<ul style="list-style-type: none"> • path - a file path
Examples	<ul style="list-style-type: none"> • <code>file_exists('/home/qgis/data/country_boundaries.shp')</code> → true

file_name

Returns the name of a file (including the file extension), excluding the directory.

语法	file_name(path)
参数	<ul style="list-style-type: none"> • path - a file path
Examples	<ul style="list-style-type: none"> • <code>file_name('/home/qgis/data/country_boundaries.shp')</code> → 'country_boundaries.shp'

file_path

Returns the directory component of a file path. This does not include the file name.

语法	file_path(path)
参数	<ul style="list-style-type: none"> • path - a file path
Examples	<ul style="list-style-type: none"> • <code>file_path('/home/qgis/data/country_boundaries.shp')</code> → '/home/qgis/data'

file_size

Returns the size (in bytes) of a file.

语法	file_size(path)
参数	<ul style="list-style-type: none"> • path - a file path
Examples	<ul style="list-style-type: none"> • <code>file_size('/home/qgis/data/country_boundaries.geojson')</code> → 5674

file_suffix

Returns the file suffix (extension) from a file path.

语法	file_suffix(path)
参数	<ul style="list-style-type: none"> • path - a file path
Examples	<ul style="list-style-type: none"> • <code>file_suffix('/home/qgis/data/country_boundaries.shp')</code> → 'shp'

is_directory

Returns true if a path corresponds to a directory.

语法	is_directory(path)
参数	<ul style="list-style-type: none"> • path - a file path
Examples	<ul style="list-style-type: none"> • <code>is_directory('/home/qgis/data/country_boundaries.shp')</code> → false • <code>is_directory('/home/qgis/data/')</code> → true

is_file

Returns true if a path corresponds to a file.

语法	is_file(path)
参数	<ul style="list-style-type: none"> • path - a file path
Examples	<ul style="list-style-type: none"> • <code>is_file('/home/qgis/data/country_boundaries.shp')</code> → true • <code>is_file('/home/qgis/data/')</code> → false

14.3.10 Form Functions

This group contains functions that operate exclusively under the attribute form context. For example, in *field's widgets* settings.

- *current_parent_value*
- *current_value*

current_parent_value

Only usable in an embedded form context, this function returns the current, unsaved value of a field in the parent form currently being edited. This will differ from the parent feature's actual attribute values for features which are currently being edited or have not yet been added to a parent layer. When used in a value-relation widget filter expression, this function should be wrapped into a 'coalesce()' that can retrieve the actual parent feature from the layer when the form is not used in an embedded context.

语法	current_parent_value(field_name)
参数	<ul style="list-style-type: none"> • field_name - a field name in the current parent form
Examples	<ul style="list-style-type: none"> • current_parent_value('FIELD_NAME') → The current value of a field 'FIELD_NAME' in the parent form.

current_value

Returns the current, unsaved value of a field in the form or table row currently being edited. This will differ from the feature's actual attribute values for features which are currently being edited or have not yet been added to a layer.

语法	current_value(field_name)
参数	<ul style="list-style-type: none"> • field_name - a field name in the current form or table row
Examples	<ul style="list-style-type: none"> • current_value('FIELD_NAME') → The current value of field 'FIELD_NAME'.

14.3.11 Fuzzy Matching Functions

This group contains functions for fuzzy comparisons between values.

- *hamming_distance*
- *levenshtein*
- *longest_common_substring*
- *soundex*

hamming_distance

Returns the Hamming distance between two strings. This equates to the number of characters at corresponding positions within the input strings where the characters are different. The input strings must be the same length, and the comparison is case-sensitive.

语法	hamming_distance(string1, string2)
参数	<ul style="list-style-type: none"> • string1 - a string • string2 - a string
Examples	<ul style="list-style-type: none"> • hamming_distance('abc', 'xec') → 2 • hamming_distance('abc', 'ABc') → 2 • hamming_distance(upper('abc'), upper('ABC')) → 0

levenshtein

Returns the Levenshtein edit distance between two strings. This equates to the minimum number of character edits (insertions, deletions or substitutions) required to change one string to another.

The Levenshtein distance is a measure of the similarity between two strings. Smaller distances mean the strings are more similar, and larger distances indicate more different strings. The distance is case sensitive.

语法	levenshtein(string1, string2)
参数	<ul style="list-style-type: none"> • string1 - a string • string2 - a string
Examples	<ul style="list-style-type: none"> • levenshtein('kittens', 'mitten') → 2 • levenshtein('Kitten', 'kitten') → 1 • levenshtein(upper('Kitten'), upper('kitten')) → 0

longest_common_substring

Returns the longest common substring between two strings. This substring is the longest string that is a substring of the two input strings. For example, the longest common substring of "ABABC" and "BABCA" is "BABC". The substring is case sensitive.

语法	longest_common_substring(string1, string2)
参数	<ul style="list-style-type: none"> • string1 - a string • string2 - a string
Examples	<ul style="list-style-type: none"> • longest_common_substring('ABABC', 'BABCA') → 'BABC' • longest_common_substring('abcDeF', 'abcdef') → 'abc' • longest_common_substring(upper('abcDeF'), upper('abcdex')) → 'ABCDE'

soundex

Returns the Soundex representation of a string. Soundex is a phonetic matching algorithm, so strings with similar sounds should be represented by the same Soundex code.

语法	soundex(string)
参数	<ul style="list-style-type: none"> • string - a string
Examples	<ul style="list-style-type: none"> • soundex('robert') → 'R163' • soundex('rupert') → 'R163' • soundex('rubin') → 'R150'

14.3.12 General Functions

This group contains general assorted functions.

- *env*
- *eval*
- *eval_template*
- *is_layer_visible*
- *layer_property*
- *var*
- *with_variable*

env

Gets an environment variable and returns its content as a string. If the variable is not found, NULL will be returned. This is handy to inject system specific configuration like drive letters or path prefixes. Definition of environment variables depends on the operating system, please check with your system administrator or the operating system documentation how this can be set.

语法	env(name)
参数	<ul style="list-style-type: none"> • name - The name of the environment variable which should be retrieved.
Examples	<ul style="list-style-type: none"> • env('LANG') → 'en_US.UTF-8' • env('MY_OWN_PREFIX_VAR') → 'Z:' • env('I_DO_NOT_EXIST') → NULL

eval

Evaluates an expression which is passed in a string. Useful to expand dynamic parameters passed as context variables or fields.

语法	eval(expression)
参数	<ul style="list-style-type: none"> • expression - an expression string
Examples	<ul style="list-style-type: none"> • <code>eval('\nice\')</code> → 'nice' • <code>eval(@expression_var)</code> → [whatever the result of evaluating @expression_var might be...]

eval_template

Evaluates a template which is passed in a string. Useful to expand dynamic parameters passed as context variables or fields.

语法	eval_template(template)
参数	<ul style="list-style-type: none"> • template - a template string
Examples	<ul style="list-style-type: none"> • <code>eval_template('QGIS [% upper(\rocks\)]')</code> → QGIS ROCKS

is_layer_visible

Returns true if a specified layer is visible.

语法	is_layer_visible(layer)
参数	<ul style="list-style-type: none"> • layer - a string, representing either a layer name or layer ID
Examples	<ul style="list-style-type: none"> • <code>is_layer_visible('baseraster')</code> → True

layer_property

Returns a matching layer property or metadata value.

语法	layer_property(layer, property)
参数	<ul style="list-style-type: none"> • layer - a string, representing either a layer name or layer ID • property - a string corresponding to the property to return. Valid options are: <ul style="list-style-type: none"> - name: layer name - id: layer ID - title: metadata title string - abstract: metadata abstract string - keywords: metadata keywords - data_url: metadata URL - attribution: metadata attribution string - attribution_url: metadata attribution URL - source: layer source - min_scale: minimum display scale for layer - max_scale: maximum display scale for layer - is_editable: if layer is in edit mode - crs: layer CRS - crs_definition: layer CRS full definition - crs_description: layer CRS description - extent: layer extent (as a geometry object) - distance_units: layer distance units - type: layer type, e.g., Vector or Raster - storage_type: storage format (vector layers only) - geometry_type: geometry type, e.g., Point (vector layers only) - feature_count: approximate feature count for layer (vector layers only) - path: File path to the layer data source. Only available for file based layers.
Examples	<ul style="list-style-type: none"> • layer_property('streets', 'title') → 'Basemap Streets' • layer_property('airports', 'feature_count') → 120 • layer_property('landsat', 'crs') → 'EPSG:4326'

Further reading: *vector*, *raster* and *mesh* layer properties

var

Returns the value stored within a specified variable.

语法	var(name)
参数	<ul style="list-style-type: none"> • name - a variable name
Examples	<ul style="list-style-type: none"> • var('qgis_version') → '2.12'

Further reading: List of default *variables*

with_variable

This function sets a variable for any expression code that will be provided as 3rd argument. This is only useful for complicated expressions, where the same calculated value needs to be used in different places.

语法	with_variable(name, value, expression)
参数	<ul style="list-style-type: none"> • name - the name of the variable to set • value - the value to set • expression - the expression for which the variable will be available
Examples	<ul style="list-style-type: none"> • with_variable('my_sum', 1 + 2 + 3, @my_sum * 2 + @my_sum * 5) → 42

14.3.13 Geometry Functions

This group contains functions that operate on geometry objects (e.g. buffer, transform, \$area).

- *angle_at_vertex*
- *\$area*
- *area*
- *azimuth*
- *boundary*
- *bounds*
- *bounds_height*
- *bounds_width*
- *buffer*
- *buffer_by_m*
- *centroid*
- *close_line*
- *closest_point*
- *collect_geometries*
- *combine*
- *contains*
- *convex_hull*
- *crosses*
- *difference*
- *disjoint*
- *distance*
- *distance_to_vertex*
- *end_point*
- *extend*

- *exterior_ring*
- *extrude*
- *flip_coordinates*
- *force_rhr*
- *geom_from_gml*
- *geom_from_wkb*
- *geom_from_wkt*
- *geom_to_wkb*
- *geom_to_wkt*
- *\$geometry*
- *geometry*
- *geometry_n*
- *hausdorff_distance*
- *inclination*
- *interior_ring_n*
- *intersection*
- *intersects*
- *intersects_bbox*
- *is_closed*
- *is_empty*
- *is_empty_or_null*
- *is_multipart*
- *is_valid*
- *\$length*
- *length*
- *line_interpolate_angle*
- *line_interpolate_point*
- *line_locate_point*
- *line_merge*
- *line_substring*
- *m*
- *m_max*
- *m_min*
- *main_angle*
- *make_circle*
- *make_ellipse*
- *make_line*
- *make_point*

- *make_point_m*
- *make_polygon*
- *make_rectangle_3points*
- *make_regular_polygon*
- *make_square*
- *make_triangle*
- *minimal_circle*
- *nodes_to_points*
- *num_geometries*
- *num_interior_rings*
- *num_points*
- *num_rings*
- *offset_curve*
- *order_parts*
- *oriented_bbox*
- *overlaps*
- *overlay_contains*
- *overlay_crosses*
- *overlay_disjoint*
- *overlay_equals*
- *overlay_intersects*
- *overlay_nearest*
- *overlay_touches*
- *overlay_within*
- *\$perimeter*
- *perimeter*
- *point_n*
- *point_on_surface*
- *pole_of_inaccessibility*
- *project*
- *relate*
- *reverse*
- *rotate*
- *segments_to_lines*
- *shortest_line*
- *simplify*
- *simplify_vw*
- *single_sided_buffer*

- *smooth*
- *start_point*
- *sym_difference*
- *tapered_buffer*
- *touches*
- *transform*
- *translate*
- *union*
- *wedge_buffer*
- *within*
- *\$x*
- *x*
- *\$x_at*
- *x_max*
- *x_min*
- *\$y*
- *y*
- *\$y_at*
- *y_max*
- *y_min*
- *z*
- *z_max*
- *z_min*

angle_at_vertex

Returns the bisector angle (average angle) to the geometry for a specified vertex on a linestring geometry. Angles are in degrees clockwise from north.

语法	<code>angle_at_vertex(geometry, vertex)</code>
参数	<ul style="list-style-type: none"> • geometry - a linestring geometry • vertex - vertex index, starting from 0; if the value is negative, the selected vertex index will be its total count minus the absolute value
Examples	<ul style="list-style-type: none"> • <code>angle_at_vertex(geometry:=geom_from_wkt('LineString(0 0, 10 0, 10 10)'),vertex:=1) → 45.0</code>

\$area

Returns the area of the current feature. The area calculated by this function respects both the current project's ellipsoid setting and area unit settings. For example, if an ellipsoid has been set for the project then the calculated area will be ellipsoidal, and if no ellipsoid is set then the calculated area will be planimetric.

语法	\$area
Examples	<ul style="list-style-type: none"> • <code>\$area</code> → 42

area

Returns the area of a geometry polygon object. Calculations are always planimetric in the Spatial Reference System (SRS) of this geometry, and the units of the returned area will match the units for the SRS. This differs from the calculations performed by the \$area function, which will perform ellipsoidal calculations based on the project's ellipsoid and area unit settings.

语法	area(geometry)
参数	<ul style="list-style-type: none"> • geometry - polygon geometry object
Examples	<ul style="list-style-type: none"> • <code>area(geom_from_wkt('POLYGON((0 0, 4 0, 4 2, 0 2, 0 0))')</code>) → 8.0

azimuth

Returns the north-based azimuth as the angle in radians measured clockwise from the vertical on point_a to point_b.

语法	azimuth(point_a, point_b)
参数	<ul style="list-style-type: none"> • point_a - point geometry • point_b - point geometry
Examples	<ul style="list-style-type: none"> • <code>degrees(azimuth(make_point(25, 45), make_point(75, 100)))</code> → 42.273689 • <code>degrees(azimuth(make_point(75, 100), make_point(25, 45)))</code> → 222.273689

boundary

Returns the closure of the combinatorial boundary of the geometry (ie the topological boundary of the geometry). For instance, a polygon geometry will have a boundary consisting of the linestrings for each ring in the polygon. Some geometry types do not have a defined boundary, e.g., points or geometry collections, and will return NULL.

语法	boundary(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • geom_to_wkt(boundary(geom_from_wkt('Polygon((1 1, 0 0, -1 1, 1 1))')))) → 'LineString(1 1,0 0,-1 1,1 1)' • geom_to_wkt(boundary(geom_from_wkt('LineString(1 1,0 0,-1 1)')))) → 'MultiPoint ((1 1),(-1 1))'

Further reading: *Boundary* algorithm

bounds

Returns a geometry which represents the bounding box of an input geometry. Calculations are in the Spatial Reference System of this geometry.

语法	bounds(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • bounds(\$geometry) → bounding box of the current feature's geometry • geom_to_wkt(bounds(geom_from_wkt('Polygon((1 1, 0 0, -1 1, 1 1))')))) → 'Polygon((-1 0, 1 0, 1 1, -1 1, -1 0))'

Further reading: *Bounding boxes* algorithm

bounds_height

Returns the height of the bounding box of a geometry. Calculations are in the Spatial Reference System of this geometry.

语法	bounds_height(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • bounds_height(\$geometry) → height of bounding box of the current feature's geometry • bounds_height(geom_from_wkt('Polygon((1 1, 0 0, -1 1, 1 1))')) → 1

bounds_width

Returns the width of the bounding box of a geometry. Calculations are in the Spatial Reference System of this geometry.

语法	<code>bounds_width(geometry)</code>
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>bounds_width(\$geometry)</code> → width of bounding box of the current feature's geometry • <code>bounds_width(geom_from_wkt('Polygon((1 1, 0 0, -1 1, 1 1))'))</code> → 2

buffer

Returns a geometry that represents all points whose distance from this geometry is less than or equal to distance. Calculations are in the Spatial Reference System of this geometry.

语法	<code>buffer(geometry, distance, [segments=8])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - a geometry • distance - buffer distance in layer units • segments - number of segments to use to represent a quarter circle when a round join style is used. A larger number results in a smoother buffer with more nodes.
Examples	<ul style="list-style-type: none"> • <code>buffer(\$geometry, 10.5)</code> → polygon of the current feature's geometry buffered by 10.5 units

Further reading: [Buffer algorithm](#)

buffer_by_m

Creates a buffer along a line geometry where the buffer diameter varies according to the m-values at the line vertices.

语法	<code>buffer_by_m(geometry, [segments=8])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - input geometry. Must be a (multi)line geometry with m values. • segments - number of segments to approximate quarter-circle curves in the buffer.
Examples	<ul style="list-style-type: none"> • <code>buffer_by_m(geometry:=geom_from_wkt('LINESTRINGM(1 2 0.5, 4 2 0.2)'), segments:=8)</code> → A variable width buffer starting with a diameter of 0.5 and ending with a diameter of 0.2 along the linestring geometry.

Further reading: [Variable width buffer \(by M value\) algorithm](#)

centroid

Returns the geometric center of a geometry.

语法	centroid(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • centroid(\$geometry) → a point geometry

Further reading: *Centroids* algorithm

close_line

Returns a closed line string of the input line string by appending the first point to the end of the line, if it is not already closed. If the geometry is not a line string or multi line string then the result will be NULL.

语法	close_line(geometry)
参数	<ul style="list-style-type: none"> • geometry - a line string geometry
Examples	<ul style="list-style-type: none"> • geom_to_wkt(close_line(geom_from_wkt('LINESTRING(0 0, 1 0, 1 1)')) → 'LineString(0 0, 1 0, 1 1, 0 0)' • geom_to_wkt(close_line(geom_from_wkt('LINESTRING(0 0, 1 0, 1 1, 0 0)')) → 'LineString(0 0, 1 0, 1 1, 0 0)'

closest_point

Returns the point on geometry1 that is closest to geometry2.

语法	closest_point(geometry1, geometry2)
参数	<ul style="list-style-type: none"> • geometry1 - geometry to find closest point on • geometry2 - geometry to find closest point to
Examples	<ul style="list-style-type: none"> • geom_to_wkt(closest_point(geom_from_wkt('LINESTRING (20 80, 98 190, 110 180, 50 75)'), geom_from_wkt('POINT(100 100)')) → 'Point(73.0769 115.384)'

collect_geometries

Collects a set of geometries into a multi-part geometry object.

List of arguments variant

Geometry parts are specified as separate arguments to the function.

语法	<code>collect_geometries(geometry1, geometry2, ...)</code>
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(collect_geometries(make_point(1,2), make_point(3,4), make_point(5,6)))</code> → <code>'MultiPoint ((1 2),(3 4),(5 6))'</code>

Array variant

Geometry parts are specified as an array of geometry parts.

语法	<code>collect_geometries(array)</code>
参数	<ul style="list-style-type: none"> • array - array of geometry objects
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(collect_geometries(array(make_point(1,2), make_point(3,4), make_point(5,6))))</code> → <code>'MultiPoint ((1 2),(3 4),(5 6))'</code>

Further reading: [Collect geometries algorithm](#)

combine

Returns the combination of two geometries.

语法	<code>combine(geometry1, geometry2)</code>
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(combine(geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)'), geom_from_wkt('LINESTRING(3 3, 4 4, 2 1)')))</code> → <code>'MULTILINESTRING((4 4, 2 1), (3 3, 4 4), (4 4, 5 5))'</code> • <code>geom_to_wkt(combine(geom_from_wkt('LINESTRING(3 3, 4 4)'), geom_from_wkt('LINESTRING(3 3, 6 6, 2 1)')))</code> → <code>'LINESTRING(3 3, 4 4, 6 6, 2 1)'</code>

contains

Tests whether a geometry contains another. Returns true if and only if no points of geometry2 lie in the exterior of geometry1, and at least one point of the interior of geometry2 lies in the interior of geometry1.

语法	<code>contains(geometry1, geometry2)</code>
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>contains(geom_from_wkt('POLYGON((0 0, 0 1, 1 1, 1 0, 0 0))'), geom_from_wkt('POINT(0.5 0.5)')) → true</code> • <code>contains(geom_from_wkt('POLYGON((0 0, 0 1, 1 1, 1 0, 0 0))'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → false</code>

Further reading: [overlay_contains](#)

convex_hull

Returns the convex hull of a geometry. It represents the minimum convex geometry that encloses all geometries within the set.

语法	<code>convex_hull(geometry)</code>
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(convex_hull(geom_from_wkt('LINESTRING(3 3, 4 4, 4 10)'))) → 'POLYGON((3 3, 4 10, 4 4, 3 3))'</code>

Further reading: [Convex hull](#) algorithm

crosses

Tests whether a geometry crosses another. Returns true if the supplied geometries have some, but not all, interior points in common.

语法	<code>crosses(geometry1, geometry2)</code>
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>crosses(geom_from_wkt('LINESTRING(3 5, 4 4, 5 3)'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → true</code> • <code>crosses(geom_from_wkt('POINT(4 5)'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → false</code>

Further reading: [overlay_crosses](#)

difference

Returns a geometry that represents that part of geometry1 that does not intersect with geometry2.

语法	difference(geometry1, geometry2)
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • geom_to_wkt(difference(geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)'), geom_from_wkt('LINESTRING(3 3, 4 4)'))) → 'LINESTRING(4 4, 5 5)'

Further reading: [Difference algorithm](#)

disjoint

Tests whether geometries do not spatially intersect. Returns true if the geometries do not share any space together.

语法	disjoint(geometry1, geometry2)
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • disjoint(geom_from_wkt('POLYGON((0 0, 0 1, 1 1, 1 0, 0 0))'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → true • disjoint(geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)'), geom_from_wkt('POINT(4 4)')) → false

Further reading: [overlay_disjoint](#)

distance

Returns the minimum distance (based on spatial ref) between two geometries in projected units.

语法	distance(geometry1, geometry2)
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • distance(geom_from_wkt('POINT(4 4)'), geom_from_wkt('POINT(4 8)')) → 4

distance_to_vertex

Returns the distance along the geometry to a specified vertex.

语法	distance_to_vertex(geometry, vertex)
参数	<ul style="list-style-type: none"> • geometry - a linestring geometry • vertex - vertex index, starting from 0; if the value is negative, the selected vertex index will be its total count minus the absolute value
Examples	<ul style="list-style-type: none"> • distance_to_vertex(geometry:=geom_from_wkt('LineString(0 0, 10 0, 10 10)'),vertex:=1) → 10.0

end_point

Returns the last node from a geometry.

语法	end_point(geometry)
参数	<ul style="list-style-type: none"> • geometry - geometry object
Examples	<ul style="list-style-type: none"> • geom_to_wkt(end_point(geom_from_wkt('LINESTRING(4 0, 4 2, 0 2)'))) → 'Point (0 2)'

Further reading: *Extract specific vertices* algorithm

extend

Extends the start and end of a linestring geometry by a specified amount. Lines are extended using the bearing of the first and last segment in the line. For a multilinestring, all the parts are extended. Distances are in the Spatial Reference System of this geometry.

语法	extend(geometry, start_distance, end_distance)
参数	<ul style="list-style-type: none"> • geometry - a (multi)linestring geometry • start_distance - distance to extend the start of the line • end_distance - distance to extend the end of the line.
Examples	<ul style="list-style-type: none"> • geom_to_wkt(extend(geom_from_wkt('LineString(0 0, 1 0, 1 1)'),1,2)) → 'LineString (-1 0, 1 0, 1 3)' • geom_to_wkt(extend(geom_from_wkt('MultiLineString((0 0, 1 0, 1 1),(2 2, 0 2, 0 5)'),1,2)) → 'MultiLineString ((-1 0, 1 0, 1 3),(3 2, 0 2, 0 7))'

Further reading: *Extend lines* algorithm

exterior_ring

Returns a line string representing the exterior ring of a polygon geometry. If the geometry is not a polygon then the result will be NULL.

语法	exterior_ring(geometry)
参数	<ul style="list-style-type: none"> • geometry - a polygon geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(exterior_ring(geom_from_wkt('POLYGON((-1 -1, 4 0, 4 2, 0 2, -1 -1),(0.1 0.1, 0.1 0.2, 0.2 0.2, 0.2, 0.1, 0.1 0.1))')))</code> → <code>'LineString (-1 -1, 4 0, 4 2, 0 2, -1 -1)'</code>

extrude

Returns an extruded version of the input (Multi-)Curve or (Multi-)Linestring geometry with an extension specified by x and y.

语法	extrude(geometry, x, y)
参数	<ul style="list-style-type: none"> • geometry - a polygon geometry • x - x extension, numeric value • y - y extension, numeric value
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(extrude(geom_from_wkt('LineString(1 2, 3 2, 4 3)'), 1, 2))</code> → <code>'Polygon ((1 2, 3 2, 4 3, 5 5, 4 4, 2 4, 1 2))'</code> • <code>geom_to_wkt(extrude(geom_from_wkt('MultiLineString((1 2, 3 2), (4 3, 8 3))'), 1, 2))</code> → <code>'MultiPolygon (((1 2, 3 2, 4 4, 2 4, 1 2)),((4 3, 8 3, 9 5, 5 5, 4 3)))'</code>

flip_coordinates

Returns a copy of the geometry with the x and y coordinates swapped. Useful for repairing geometries which have had their latitude and longitude values reversed.

语法	flip_coordinates(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(flip_coordinates(make_point(1, 2)))</code> → <code>'Point (2 1)'</code>

Further reading: [Swap X and Y coordinates](#) algorithm

force_rhr

Forces a geometry to respect the Right-Hand-Rule, in which the area that is bounded by a polygon is to the right of the boundary. In particular, the exterior ring is oriented in a clockwise direction and the interior rings in a counter-clockwise direction.

语法	force_rhr(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry. Any non-polygon geometries are returned unchanged.
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(force_rhr(geometry:=geom_from_wkt('POLYGON((-1 -1, 4 0, 4 2, 0 2, -1 -1))))</code> → <code>'Polygon((-1 -1, 0 2, 4 2, 4 0, -1 -1))'</code>

Further reading: *Force right-hand-rule algorithm*

geom_from_gml

Returns a geometry from a GML representation of geometry.

语法	geom_from_gml(gml)
参数	<ul style="list-style-type: none"> • gml - GML representation of a geometry as a string
Examples	<ul style="list-style-type: none"> • <code>geom_from_gml('<gml:LineString srsName="EPSG:4326"><gml:coordinates>4, 4 5,5 6,6</gml:coordinates></gml:LineString>')</code> → a line geometry object

geom_from_wkb

Returns a geometry created from a Well-Known Binary (WKB) representation.

语法	geom_from_wkb(binary)
参数	<ul style="list-style-type: none"> • binary - Well-Known Binary (WKB) representation of a geometry (as a binary blob)
Examples	<ul style="list-style-type: none"> • <code>geom_from_wkb(geom_to_wkb(make_point(4,5)))</code> → a point geometry object

geom_from_wkt

Returns a geometry created from a Well-Known Text (WKT) representation.

语法	geom_from_wkt(text)
参数	<ul style="list-style-type: none"> • text - Well-Known Text (WKT) representation of a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_from_wkt('POINT(4 5)')</code> → a geometry object

geom_to_wkb

Returns the Well-Known Binary (WKB) representation of a geometry

语法	geom_to_wkb(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • geom_to_wkb(\$geometry) → binary blob containing a geometry object

geom_to_wkt

Returns the Well-Known Text (WKT) representation of the geometry without SRID metadata.

语法	geom_to_wkt(geometry, [precision=8]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - a geometry • precision - numeric precision
Examples	<ul style="list-style-type: none"> • geom_to_wkt(make_point(6, 50)) → 'POINT(6 50)' • geom_to_wkt(centroid(geom_from_wkt('Polygon((1 1, 0 0, -1 1, 1 1))')))) → 'POINT(0 0.66666667)' • geom_to_wkt(centroid(geom_from_wkt('Polygon((1 1, 0 0, -1 1, 1 1))')), 2) → 'POINT(0 0.67)'

\$geometry

Returns the geometry of the current feature. Can be used for processing with other functions.

语法	\$geometry
Examples	<ul style="list-style-type: none"> • geom_to_wkt(\$geometry) → 'POINT(6 50)'

geometry

Returns a feature's geometry.

语法	geometry(feature)
参数	<ul style="list-style-type: none"> • feature - a feature object
Examples	<ul style="list-style-type: none"> • geom_to_wkt(geometry(get_feature(layer, attributeField, value))) → 'POINT(6 50)' • intersects(\$geometry, geometry(get_feature(layer, attributeField, value))) → true

geometry_n

Returns a specific geometry from a geometry collection, or NULL if the input geometry is not a collection.

语法	geometry_n(geometry, index)
参数	<ul style="list-style-type: none"> • geometry - geometry collection • index - index of geometry to return, where 1 is the first geometry in the collection
Examples	<ul style="list-style-type: none"> • geom_to_wkt(geometry_n(geom_from_wkt('GEOMETRYCOLLECTION (POINT (0 1), POINT(0 0), POINT(1 0), POINT(1 1))'), 3)) → 'Point (1 0)'

hausdorff_distance

Returns the Hausdorff distance between two geometries. This is basically a measure of how similar or dissimilar 2 geometries are, with a lower distance indicating more similar geometries.

The function can be executed with an optional densify fraction argument. If not specified, an approximation to the standard Hausdorff distance is used. This approximation is exact or close enough for a large subset of useful cases. Examples of these are:

- computing distance between Linestrings that are roughly parallel to each other, and roughly equal in length. This occurs in matching linear networks.
- Testing similarity of geometries.

If the default approximate provided by this method is insufficient, specify the optional densify fraction argument. Specifying this argument performs a segment densification before computing the discrete Hausdorff distance. The parameter sets the fraction by which to densify each segment. Each segment will be split into a number of equal-length subsegments, whose fraction of the total length is closest to the given fraction. Decreasing the densify fraction parameter will make the distance returned approach the true Hausdorff distance for the geometries.

语法	hausdorff_distance(geometry1, geometry2, [densify_fraction]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry • densify_fraction - densify fraction amount
Examples	<ul style="list-style-type: none"> • hausdorff_distance(geom_from_wkt('LINESTRING (0 0, 2 1)'), geom_from_wkt('LINESTRING (0 0, 2 0)')) → 2 • hausdorff_distance(geom_from_wkt('LINESTRING (130 0, 0 0, 0 150)'), geom_from_wkt('LINESTRING (10 10, 10 150, 130 10)')) → 14.142135623 • hausdorff_distance(geom_from_wkt('LINESTRING (130 0, 0 0, 0 150)'), geom_from_wkt('LINESTRING (10 10, 10 150, 130 10)'), 0.5) → 70.0

inclination

Returns the inclination measured from the zenith (0) to the nadir (180) on `point_a` to `point_b`.

语法	<code>inclination(point_a, point_b)</code>
参数	<ul style="list-style-type: none"> • point_a - point geometry • point_b - point geometry
Examples	<ul style="list-style-type: none"> • <code>inclination(make_point(5, 10, 0), make_point(5, 10, 5)) → 0.0</code> • <code>inclination(make_point(5, 10, 0), make_point(5, 10, 0)) → 90.0</code> • <code>inclination(make_point(5, 10, 0), make_point(50, 100, 0)) → 90.0</code> • <code>inclination(make_point(5, 10, 0), make_point(5, 10, -5)) → 180.0</code>

interior_ring_n

Returns a specific interior ring from a polygon geometry, or NULL if the geometry is not a polygon.

语法	<code>interior_ring_n(geometry, index)</code>
参数	<ul style="list-style-type: none"> • geometry - polygon geometry • index - index of interior to return, where 1 is the first interior ring
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(interior_ring_n(geom_from_wkt('POLYGON((-1 -1, 4 0, 4 2, 0 2, -1 -1), (-0.1 -0.1, 0.4 0, 0.4 0.2, 0 0.2, -0.1 -0.1), (-1 -1, 4 0, 4 2, 0 2, -1 -1))'), 1)) → 'LineString (-0.1 -0.1, 0.4 0, 0.4 0.2, 0 0.2, -0.1 -0.1)'</code>

intersection

Returns a geometry that represents the shared portion of two geometries.

语法	<code>intersection(geometry1, geometry2)</code>
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(intersection(geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)'), geom_from_wkt('LINESTRING(3 3, 4 4)'))) → 'LINESTRING(3 3, 4 4)'</code> • <code>geom_to_wkt(intersection(geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)'), geom_from_wkt('MULTIPOINT(3.5 3.5, 4 5)'))) → 'POINT(3.5 3.5)'</code>

Further reading: [Intersection](#) algorithm

intersects

Tests whether a geometry intersects another. Returns true if the geometries spatially intersect (share any portion of space) and false if they do not.

语法	<code>intersects(geometry1, geometry2)</code>
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>intersects(geom_from_wkt('POINT(4 4)'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → true</code> • <code>intersects(geom_from_wkt('POINT(4 5)'), geom_from_wkt('POINT(5 5)')) → false</code>

Further reading: [overlay_intersects](#)

intersects_bbox

Tests whether a geometry's bounding box overlaps another geometry's bounding box. Returns true if the geometries spatially intersect the bounding box defined and false if they do not.

语法	<code>intersects_bbox(geometry1, geometry2)</code>
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>intersects_bbox(geom_from_wkt('POINT(4 5)'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → true</code> • <code>intersects_bbox(geom_from_wkt('POINT(6 5)'), geom_from_wkt('POLYGON((3 3, 4 4, 5 5, 3 3))')) → false</code>

is_closed

Returns true if a line string is closed (start and end points are coincident), or false if a line string is not closed. If the geometry is not a line string then the result will be NULL.

语法	<code>is_closed(geometry)</code>
参数	<ul style="list-style-type: none"> • geometry - a line string geometry
Examples	<ul style="list-style-type: none"> • <code>is_closed(geom_from_wkt('LINESTRING(0 0, 1 1, 2 2)')) → false</code> • <code>is_closed(geom_from_wkt('LINESTRING(0 0, 1 1, 2 2, 0 0)')) → true</code>

is_empty

Returns true if a geometry is empty (without coordinates), false if the geometry is not empty and NULL if there is no geometry. See also `is_empty_or_null`.

语法	<code>is_empty(geometry)</code>
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>is_empty(geom_from_wkt('LINESTRING(0 0, 1 1, 2 2)')) → false</code> • <code>is_empty(geom_from_wkt('LINESTRING EMPTY')) → true</code> • <code>is_empty(geom_from_wkt('POINT(7 4)')) → false</code> • <code>is_empty(geom_from_wkt('POINT EMPTY')) → true</code>

is_empty_or_null

Returns true if a geometry is NULL or empty (without coordinates) or false otherwise. This function is like the expression '`$geometry IS NULL or is_empty($geometry)`'

语法	<code>is_empty_or_null(geometry)</code>
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>is_empty_or_null(NULL) → true</code> • <code>is_empty_or_null(geom_from_wkt('LINESTRING(0 0, 1 1, 2 2)')) → false</code> • <code>is_empty_or_null(geom_from_wkt('LINESTRING EMPTY')) → true</code> • <code>is_empty_or_null(geom_from_wkt('POINT(7 4)')) → false</code> • <code>is_empty_or_null(geom_from_wkt('POINT EMPTY')) → true</code>

is_multipart

Returns true if the geometry is of Multi type.

语法	<code>is_multipart(geometry)</code>
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>is_multipart(geom_from_wkt('MULTIPOINT ((0 0), (1 1), (2 2)')) → true</code> • <code>is_multipart(geom_from_wkt('POINT (0 0)')) → false</code>

is_valid

Returns true if a geometry is valid; if it is well-formed in 2D according to the OGC rules.

语法	is_valid(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • is_valid(geom_from_wkt('LINESTRING(0 0, 1 1, 2 2, 0 0)')) → true • is_valid(geom_from_wkt('LINESTRING(0 0)')) → false

\$length

Returns the length of a linestring. If you need the length of a border of a polygon, use \$perimeter instead. The length calculated by this function respects both the current project's ellipsoid setting and distance unit settings. For example, if an ellipsoid has been set for the project then the calculated length will be ellipsoidal, and if no ellipsoid is set then the calculated length will be planimetric.

语法	\$length
Examples	<ul style="list-style-type: none"> • \$length → 42.4711

length

Returns the number of characters in a string or the length of a geometry linestring.

String variant

Returns the number of characters in a string.

语法	length(string)
参数	<ul style="list-style-type: none"> • string - string to count length of
Examples	<ul style="list-style-type: none"> • length('hello') → 5

Geometry variant

Calculate the length of a geometry line object. Calculations are always planimetric in the Spatial Reference System (SRS) of this geometry, and the units of the returned length will match the units for the SRS. This differs from the calculations performed by the \$length function, which will perform ellipsoidal calculations based on the project's ellipsoid and distance unit settings.

语法	length(geometry)
参数	<ul style="list-style-type: none"> • geometry - line geometry object
Examples	<ul style="list-style-type: none"> • length(geom_from_wkt('LINESTRING(0 0, 4 0)')) → 4.0

line_interpolate_angle

Returns the angle parallel to the geometry at a specified distance along a linestring geometry. Angles are in degrees clockwise from north.

语法	line_interpolate_angle(geometry, distance)
参数	<ul style="list-style-type: none"> • geometry - a linestring geometry • distance - distance along line to interpolate angle at
Examples	<ul style="list-style-type: none"> • line_interpolate_angle(geometry:=geom_from_wkt('LineString(0 0, 10 0)'),distance:=5) → 90.0

line_interpolate_point

Returns the point interpolated by a specified distance along a linestring geometry.

语法	line_interpolate_point(geometry, distance)
参数	<ul style="list-style-type: none"> • geometry - a linestring geometry • distance - distance along line to interpolate
Examples	<ul style="list-style-type: none"> • geom_to_wkt(line_interpolate_point(geometry:=geom_from_wkt('LineString(0 0, 10 0)'),distance:=5)) → 'Point(5 0)'

Further reading: *Interpolate point on line* algorithm

line_locate_point

Returns the distance along a linestring corresponding to the closest position the linestring comes to a specified point geometry.

语法	line_locate_point(geometry, point)
参数	<ul style="list-style-type: none"> • geometry - a linestring geometry • point - point geometry to locate closest position on linestring to
Examples	<ul style="list-style-type: none"> • line_locate_point(geometry:=geom_from_wkt('LineString(0 0, 10 0)'),point:=geom_from_wkt('Point(5 0)')) → 5.0

line_merge

Returns a LineString or MultiLineString geometry, where any connected LineStrings from the input geometry have been merged into a single linestring. This function will return NULL if passed a geometry which is not a LineString/MultiLineString.

语法	line_merge(geometry)
参数	<ul style="list-style-type: none"> • geometry - a LineString/MultiLineString geometry
Examples	<ul style="list-style-type: none"> • geom_to_wkt(line_merge(geom_from_wkt('MULTILINESTRING((0 0, 1 1),(1 1, 2 2))')) → 'LineString(0 0,1 1,2 2)' • geom_to_wkt(line_merge(geom_from_wkt('MULTILINESTRING((0 0, 1 1),(11 1, 21 2))')) → 'MultiLineString((0 0, 1 1),(11 1, 21 2)')

line_substring

Returns the portion of a line (or curve) geometry which falls between the specified start and end distances (measured from the beginning of the line). Z and M values are linearly interpolated from existing values.

语法	line_substring(geometry, start_distance, end_distance)
参数	<ul style="list-style-type: none"> • geometry - a linestring or curve geometry • start_distance - distance to start of substring • end_distance - distance to end of substring
Examples	<ul style="list-style-type: none"> • geom_to_wkt(line_substring(geometry:=geom_from_wkt('LineString(0 0, 10 0)'), start_distance:=2, end_distance:=6)) → 'LineString(2 0,6 0)'

Further reading: [Line substring algorithm](#)

m

Returns the m value of a point geometry.

语法	m(geometry)
参数	<ul style="list-style-type: none"> • geometry - a point geometry
Examples	<ul style="list-style-type: none"> • m(geom_from_wkt('POINTM(2 5 4)')) → 4

m_max

Returns the maximum m (measure) value of a geometry.

语法	m_max(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry containing m values
Examples	<ul style="list-style-type: none"> • m_max(make_point_m(0,0,1)) → 1 • m_max(make_line(make_point_m(0,0,1), make_point_m(-1, -1,2), make_point_m(-2,-2,0))) → 2

m_min

Returns the minimum m (measure) value of a geometry.

语法	m_min(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry containing m values
Examples	<ul style="list-style-type: none"> • m_min(make_point_m(0,0,1)) → 1 • m_min(make_line(make_point_m(0,0,1), make_point_m(-1, -1,2), make_point_m(-2,-2,0))) → 0

main_angle

Returns the main angle of a geometry (clockwise, in degrees from North), which represents the angle of the oriented minimal bounding rectangle which completely covers the geometry.

语法	main_angle(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • main_angle(geom_from_wkt('Polygon ((321577 129614, 321581 129618, 321585 129615, 321581 129610, 321577 129614))')) → 38.66

make_circle

Creates a circular polygon.

语法	make_circle(center, radius, [segments=36]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • center - center point of the circle • radius - radius of the circle • segments - optional argument for polygon segmentation. By default this value is 36
Examples	<ul style="list-style-type: none"> • geom_to_wkt(make_circle(make_point(10,10), 5, 4)) → 'Polygon ((10 15, 15 10, 10 5, 5 10, 10 15))' • geom_to_wkt(make_circle(make_point(10,10,5), 5, 4)) → 'PolygonZ ((10 15 5, 15 10 5, 10 5 5, 5 10 5, 10 15 5))' • geom_to_wkt(make_circle(make_point(10,10,5,30), 5, 4)) → 'PolygonZM ((10 15 5 30, 15 10 5 30, 10 5 5 30, 5 10 5 30, 10 15 5 30))'

make_ellipse

Creates an elliptical polygon.

语法	make_ellipse(center, semi_major_axis, semi_minor_axis, azimuth, [segments=36]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • center - center point of the ellipse • semi_major_axis - semi-major axis of the ellipse • semi_minor_axis - semi-minor axis of the ellipse • azimuth - orientation of the ellipse • segments - optional argument for polygon segmentation. By default this value is 36
Examples	<ul style="list-style-type: none"> • geom_to_wkt (make_ellipse (make_point (10,10), 5, 2, 90, 4)) → 'Polygon ((15 10, 10 8, 5 10, 10 12, 15 10))' • geom_to_wkt (make_ellipse (make_point (10,10,5), 5, 2, 90, 4)) → 'PolygonZ ((15 10 5, 10 8 5, 5 10 5, 10 12 5, 15 10 5))' • geom_to_wkt (make_ellipse (make_point (10,10,5,30), 5, 2, 90, 4)) → 'PolygonZM ((15 10 5 30, 10 8 5 30, 5 10 5 30, 10 12 5 30, 15 10 5 30))'

make_line

Creates a line geometry from a series of point geometries.

List of arguments variant

Line vertices are specified as separate arguments to the function.

语法	make_line(point1, point2, ...)
参数	<ul style="list-style-type: none"> • point - a point geometry (or array of points)
Examples	<ul style="list-style-type: none"> • geom_to_wkt (make_line (make_point (2, 4), make_point (3, 5))) → 'LineString (2 4, 3 5)' • geom_to_wkt (make_line (make_point (2, 4), make_point (3, 5), make_point (9, 7))) → 'LineString (2 4, 3 5, 9 7)'

Array variant

Line vertices are specified as an array of points.

语法	make_line(array)
参数	<ul style="list-style-type: none"> • array - array of points
Examples	<ul style="list-style-type: none"> • geom_to_wkt (make_line (array (make_point (2, 4), make_point (3, 5), make_point (9, 7)))) → 'LineString (2 4, 3 5, 9 7)'

make_point

Creates a point geometry from an x and y (and optional z and m) value.

语法	make_point(x, y, [z], [m]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • x - x coordinate of point • y - y coordinate of point • z - optional z coordinate of point • m - optional m value of point
Examples	<ul style="list-style-type: none"> • geom_to_wkt(make_point(2, 4)) → 'Point (2 4)' • geom_to_wkt(make_point(2, 4, 6)) → 'PointZ (2 4 6)' • geom_to_wkt(make_point(2, 4, 6, 8)) → 'PointZM (2 4 6 8)'

make_point_m

Creates a point geometry from an x, y coordinate and m value.

语法	make_point_m(x, y, m)
参数	<ul style="list-style-type: none"> • x - x coordinate of point • y - y coordinate of point • m - m value of point
Examples	<ul style="list-style-type: none"> • geom_to_wkt(make_point_m(2, 4, 6)) → 'PointM (2 4 6)'

make_polygon

Creates a polygon geometry from an outer ring and optional series of inner ring geometries.

语法	make_polygon(outerRing, [innerRing1], [innerRing2], ...) [] marks optional arguments
参数	<ul style="list-style-type: none"> • outerRing - closed line geometry for polygon's outer ring • innerRing - optional closed line geometry for inner ring
Examples	<ul style="list-style-type: none"> • geom_to_wkt(make_polygon(geom_from_wkt('LINESTRING(0 0, 0 1, 1 1, 1 0, 0 0)'))) → 'Polygon ((0 0, 0 1, 1 1, 1 0, 0 0))' • geom_to_wkt(make_polygon(geom_from_wkt('LINESTRING(0 0, 0 1, 1 1, 1 0, 0 0)'), geom_from_wkt('LINESTRING(0.1 0.1, 0.1 0.2, 0.2 0.2, 0.2 0.1, 0.1 0.1)'), geom_from_wkt('LINESTRING(0.8 0.8, 0.8 0.9, 0.9 0.9, 0.9 0.8, 0.8 0.8)'))) → 'Polygon ((0 0, 0 1, 1 1, 1 0, 0 0),(0.1 0.1, 0.1 0.2, 0.2 0.2, 0.2 0.1, 0.1 0.1),(0.8 0.8, 0.8 0.9, 0.9 0.9, 0.9 0.8, 0.8 0.8))'

make_rectangle_3points

Creates a rectangle from 3 points.

语法	make_rectangle_3points(point1, point2, point3, [option=0]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • point1 - First point. • point2 - Second point. • point3 - Third point. • option - An optional argument to construct the rectangle. By default this value is 0. Value can be 0 (distance) or 1 (projected). Option distance: Second distance is equal to the distance between 2nd and 3rd point. Option projected: Second distance is equal to the distance of the perpendicular projection of the 3rd point on the segment or its extension.
Examples	<ul style="list-style-type: none"> • geom_to_wkt (make_rectangle (make_point (0, 0), make_point (0, 5), make_point (5, 5), 0)) → 'Polygon ((0 0, 0 5, 5 5, 5 0, 0 0))' • geom_to_wkt (make_rectangle (make_point (0, 0), make_point (0, 5), make_point (5, 3), 1)) → 'Polygon ((0 0, 0 5, 5 5, 5 0, 0 0))'

make_regular_polygon

Creates a regular polygon.

语法	make_regular_polygon(center, radius, number_sides, [circle=0]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • center - center of the regular polygon • radius - second point. The first if the regular polygon is inscribed. The midpoint of the first side if the regular polygon is circumscribed. • number_sides - Number of sides/edges of the regular polygon • circle - Optional argument to construct the regular polygon. By default this value is 0. Value can be 0 (inscribed) or 1 (circumscribed)
Examples	<ul style="list-style-type: none"> • geom_to_wkt (make_regular_polygon (make_point (0, 0), make_point (0, 5), 5)) → 'Polygon ((0 5, 4.76 1.55, 2.94 -4.05, -2.94 -4.05, -4.76 1.55, 0 5))' • geom_to_wkt (make_regular_polygon (make_point (0, 0), project (make_point (0, 0), 4.0451, radians (36)), 5)) → 'Polygon ((0 5, 4.76 1.55, 2.94 -4.05, -2.94 -4.05, -4.76 1.55, 0 5))'

make_square

Creates a square from a diagonal.

语法	make_square(point1, point2)
参数	<ul style="list-style-type: none"> • point1 - First point of the diagonal • point2 - Last point of the diagonal
Examples	<ul style="list-style-type: none"> • geom_to_wkt(make_square(make_point(0,0), make_point(5,5))) → 'Polygon ((0 0, -0 5, 5 5, 5 0, 0 0))' • geom_to_wkt(make_square(make_point(5,0), make_point(5,5))) → 'Polygon ((5 0, 2.5 2.5, 5 5, 7.5 2.5, 5 0))'

make_triangle

Creates a triangle polygon.

语法	make_triangle(point1, point2, point3)
参数	<ul style="list-style-type: none"> • point1 - first point of the triangle • point2 - second point of the triangle • point3 - third point of the triangle
Examples	<ul style="list-style-type: none"> • geom_to_wkt(make_triangle(make_point(0,0), make_point(5,5), make_point(0,10))) → 'Triangle ((0 0, 5 5, 0 10, 0 0))' • geom_to_wkt(boundary(make_triangle(make_point(0,0), make_point(5,5), make_point(0,10)))) → 'LineString (0 0, 5 5, 0 10, 0 0)'

minimal_circle

Returns the minimal enclosing circle of a geometry. It represents the minimum circle that encloses all geometries within the set.

语法	minimal_circle(geometry, [segments=36]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - a geometry • segments - optional argument for polygon segmentation. By default this value is 36
Examples	<ul style="list-style-type: none"> • geom_to_wkt(minimal_circle(geom_from_wkt('LINESTRING(0 5, 0 -5, 2 1)'), 4)) → 'Polygon ((0 5, 5 -0, -0 -5, -5 0, 0 5))' • geom_to_wkt(minimal_circle(geom_from_wkt('MULTIPOINT(1 2, 3 4, 3 2)'), 4)) → 'Polygon ((3 4, 3 2, 1 2, 1 4, 3 4))'

Further reading: [Minimum enclosing circles algorithm](#)

nodes_to_points

Returns a multipoint geometry consisting of every node in the input geometry.

语法	<code>nodes_to_points(geometry, [ignore_closing_nodes=false])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - geometry object • ignore_closing_nodes - optional argument specifying whether to include duplicate nodes which close lines or polygons rings. Defaults to false, set to true to avoid including these duplicate nodes in the output collection.
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(nodes_to_points(geom_from_wkt('LINESTRING(0 0, 1 1, 2 2)')))</code> → <code>'MultiPoint((0 0),(1 1),(2 2))'</code> • <code>geom_to_wkt(nodes_to_points(geom_from_wkt('POLYGON((-1 -1, 4 0, 4 2, 0 2, -1 -1)'),true))</code> → <code>'MultiPoint((-1 -1),(4 0),(4 2),(0 2))'</code>

Further reading: [Extract vertices algorithm](#)

num_geometries

Returns the number of geometries in a geometry collection, or NULL if the input geometry is not a collection.

语法	<code>num_geometries(geometry)</code>
参数	<ul style="list-style-type: none"> • geometry - geometry collection
Examples	<ul style="list-style-type: none"> • <code>num_geometries(geom_from_wkt('GEOMETRYCOLLECTION(POINT(0 1), POINT(0 0), POINT(1 0), POINT(1 1))'))</code> → 4

num_interior_rings

Returns the number of interior rings in a polygon or geometry collection, or NULL if the input geometry is not a polygon or collection.

语法	<code>num_interior_rings(geometry)</code>
参数	<ul style="list-style-type: none"> • geometry - input geometry
Examples	<ul style="list-style-type: none"> • <code>num_interior_rings(geom_from_wkt('POLYGON((-1 -1, 4 0, 4 2, 0 2, -1 -1), (-0.1 -0.1, 0.4 0, 0.4 0.2, 0 0.2, -0.1 -0.1))'))</code> → 1

num_points

Returns the number of vertices in a geometry.

语法	num_points(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • num_points(\$geometry) → number of vertices in the current feature's geometry

num_rings

Returns the number of rings (including exterior rings) in a polygon or geometry collection, or NULL if the input geometry is not a polygon or collection.

语法	num_rings(geometry)
参数	<ul style="list-style-type: none"> • geometry - input geometry
Examples	<ul style="list-style-type: none"> • num_rings(geom_from_wkt('POLYGON((-1 -1, 4 0, 4 2, 0 2, -1 -1), (-0.1 -0.1, 0.4 0, 0.4 0.2, 0 0.2, -0.1 -0.1))') → 2

offset_curve

Returns a geometry formed by offsetting a linestring geometry to the side. Distances are in the Spatial Reference System of this geometry.

语法	offset_curve(geometry, distance, [segments=8], [join=1], [miter_limit=2.0]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - a (multi)linestring geometry • distance - offset distance. Positive values will be buffered to the left of lines, negative values to the right • segments - number of segments to use to represent a quarter circle when a round join style is used. A larger number results in a smoother line with more nodes. • join - join style for corners, where 1 = round, 2 = miter and 3 = bevel • miter_limit - limit on the miter ratio used for very sharp corners (when using miter joins only)
Examples	<ul style="list-style-type: none"> • offset_curve(\$geometry, 10.5) → line offset to the left by 10.5 units • offset_curve(\$geometry, -10.5) → line offset to the right by 10.5 units • offset_curve(\$geometry, 10.5, segments=16, join=1) → line offset to the left by 10.5 units, using more segments to result in a smoother curve • offset_curve(\$geometry, 10.5, join=3) → line offset to the left by 10.5 units, using a beveled join

Further reading: [Offset lines](#) algorithm

order_parts

Orders the parts of a MultiGeometry by a given criteria

语法	order_parts(geometry, orderby, ascending)
参数	<ul style="list-style-type: none"> • geometry - a multi-type geometry • orderby - an expression string defining the order criteria • ascending - boolean, True for ascending, False for descending
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(order_parts(geom_from_wkt('MultiPolygon (((1 1, 5 1, 5 5, 1 5, 1 1)), ((1 1, 9 1, 9 9, 1 9, 1 1)))'), 'area(\$geometry)', False)) → 'MultiPolygon (((1 1, 9 1, 9 9, 1 9, 1 1)), ((1 1, 5 1, 5 5, 1 5, 1 1)))'</code> • <code>geom_to_wkt(order_parts(geom_from_wkt('LineString(1 2, 3 2, 4 3)'), '1', True)) → 'LineString(1 2, 3 2, 4 3)'</code>

oriented_bbox

Returns a geometry which represents the minimal oriented bounding box of an input geometry.

语法	oriented_bbox(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(oriented_bbox(geom_from_wkt('MULTIPOINT(1 2, 3 4, 3 2)'))) → 'Polygon ((3 2, 3 4, 1 4, 1 2, 3 2))'</code>

Further reading: *Oriented minimum bounding box* algorithm

overlaps

Tests whether a geometry overlaps another. Returns true if the geometries share space, are of the same dimension, but are not completely contained by each other.

语法	overlaps(geometry1, geometry2)
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>overlaps(geom_from_wkt('LINESTRING(3 5, 4 4, 5 5, 5 3)'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → true</code> • <code>overlaps(geom_from_wkt('LINESTRING(0 0, 1 1)'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → false</code>

overlay_contains

Returns whether the current feature spatially contains at least one feature from a target layer, or an array of expression-based results for the features in the target layer contained in the current feature.

Read more on the underlying GEOS "Contains" predicate, as described in PostGIS [ST_Contains](#) function.

语法	<pre>overlay_contains(layer, [expression], [filter], [limit], [cache=false])</pre> <p>[] marks optional arguments</p>
参数	<ul style="list-style-type: none"> • layer - the layer whose overlay is checked • expression - an optional expression to evaluate on the features from the target layer. If not set, the function will just return a boolean indicating whether there is at least one match. • filter - an optional expression to filter the target features to check. If not set, all the features will be checked. • limit - an optional integer to limit the number of matching features. If not set, all the matching features will be returned. • cache - set this to true to build a local spatial index (most of the time, this is unwanted, unless you are working with a particularly slow data provider)
Examples	<ul style="list-style-type: none"> • <code>overlay_contains('regions')</code> → true if the current feature spatially contains a region • <code>overlay_contains('regions', filter:= population > 10000)</code> → true if the current feature spatially contains a region with a population greater than 10000 • <code>overlay_contains('regions', name)</code> → an array of names, for the regions contained in the current feature • <code>array_to_string(overlay_contains('regions', name))</code> → a string as a comma separated list of names, for the regions contained in the current feature • <code>array_length(overlay_contains('regions', name))</code> → the number of regions contained in the current feature • <code>array_sort(overlay_contains(layer:='regions', expression:="name", filter:= population > 10000))</code> → an ordered array of names, for the regions contained in the current feature and with a population greater than 10000 • <code>overlay_contains(layer:='regions', expression:= geom_to_wkt(\$geometry), limit:=2)</code> → an array of geometries (in WKT), for up to two regions contained in the current feature

Further reading: *contains*, *array manipulation*, *Select by location* algorithm

overlay_crosses

Returns whether the current feature spatially crosses at least one feature from a target layer, or an array of expression-based results for the features in the target layer crossed by the current feature.

Read more on the underlying GEOS "Crosses" predicate, as described in PostGIS [ST_Crosses](#) function.

语法	<pre>overlay_crosses(layer, [expression], [filter], [limit], [cache=false])</pre> <p>[] marks optional arguments</p>
参数	<ul style="list-style-type: none"> • layer - the layer whose overlay is checked • expression - an optional expression to evaluate on the features from the target layer. If not set, the function will just return a boolean indicating whether there is at least one match. • filter - an optional expression to filter the target features to check. If not set, all the features will be checked. • limit - an optional integer to limit the number of matching features. If not set, all the matching features will be returned. • cache - set this to true to build a local spatial index (most of the time, this is unwanted, unless you are working with a particularly slow data provider)
Examples	<ul style="list-style-type: none"> • <code>overlay_crosses('regions')</code> → true if the current feature spatially crosses a region • <code>overlay_crosses('regions', filter:= population > 10000)</code> → true if the current feature spatially crosses a region with a population greater than 10000 • <code>overlay_crosses('regions', name)</code> → an array of names, for the regions crossed by the current feature • <code>array_to_string(overlay_crosses('regions', name))</code> → a string as a comma separated list of names, for the regions crossed by the current feature • <code>array_sort(overlay_crosses(layer:='regions', expression:="name", filter:= population > 10000))</code> → an ordered array of names, for the regions crossed by the current feature and with a population greater than 10000 • <code>overlay_crosses(layer:='regions', expression:= geom_to_wkt(\$geometry), limit:=2)</code> → an array of geometries (in WKT), for up to two regions crossed by the current feature

Further reading: [crosses](#), [array manipulation](#), [Select by location](#) algorithm

overlay_disjoint

Returns whether the current feature is spatially disjoint from all the features of a target layer, or an array of expression-based results for the features in the target layer that are disjoint from the current feature.

Read more on the underlying GEOS "Disjoint" predicate, as described in PostGIS [ST_Disjoint](#) function.

语法	<code>overlay_disjoint(layer, [expression], [filter], [limit], [cache=false])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - the layer whose overlay is checked • expression - an optional expression to evaluate on the features from the target layer. If not set, the function will just return a boolean indicating whether there is at least one match. • filter - an optional expression to filter the target features to check. If not set, all the features will be checked. • limit - an optional integer to limit the number of matching features. If not set, all the matching features will be returned. • cache - set this to true to build a local spatial index (most of the time, this is unwanted, unless you are working with a particularly slow data provider)
Examples	<ul style="list-style-type: none"> • <code>overlay_disjoint('regions')</code> → true if the current feature is spatially disjoint from all the regions • <code>overlay_disjoint('regions', filter:= population > 10000)</code> → true if the current feature is spatially disjoint from all the regions with a population greater than 10000 • <code>overlay_disjoint('regions', name)</code> → an array of names, for the regions spatially disjoint from the current feature • <code>array_to_string(overlay_disjoint('regions', name))</code> → a string as a comma separated list of names, for the regions spatially disjoint from the current feature • <code>array_sort(overlay_disjoint(layer:='regions', expression:="name", filter:= population > 10000))</code> → an ordered array of names, for the regions spatially disjoint from the current feature and with a population greater than 10000 • <code>overlay_disjoint(layer:='regions', expression:= geom_to_wkt(\$geometry), limit:=2)</code> → an array of geometries (in WKT), for up to two regions spatially disjoint from the current feature

Further reading: [disjoint](#), [array manipulation](#), [Select by location](#) algorithm

overlay_equals

Returns whether the current feature spatially equals to at least one feature from a target layer, or an array of expression-based results for the features in the target layer that are spatially equal to the current feature.

Read more on the underlying GEOS "Equals" predicate, as described in PostGIS [ST_Equals](#) function.

语法	<code>overlay_equals(layer, [expression], [filter], [limit], [cache=false])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - the layer whose overlay is checked • expression - an optional expression to evaluate on the features from the target layer. If not set, the function will just return a boolean indicating whether there is at least one match. • filter - an optional expression to filter the target features to check. If not set, all the features will be checked. • limit - an optional integer to limit the number of matching features. If not set, all the matching features will be returned. • cache - set this to true to build a local spatial index (most of the time, this is unwanted, unless you are working with a particularly slow data provider)
Examples	<ul style="list-style-type: none"> • <code>overlay_equals('regions')</code> → true if the current feature is spatially equal to a region • <code>overlay_equals('regions', filter:= population > 10000)</code> → true if the current feature is spatially equal to a region with a population greater than 10000 • <code>overlay_equals('regions', name)</code> → an array of names, for the regions spatially equal to the current feature • <code>array_to_string(overlay_equals('regions', name))</code> → a string as a comma separated list of names, for the regions spatially equal to the current feature • <code>array_sort(overlay_equals(layer:='regions', expression:="name", filter:= population > 10000))</code> → an ordered array of names, for the regions spatially equal to the current feature and with a population greater than 10000 • <code>overlay_equals(layer:='regions', expression:= geom_to_wkt(\$geometry), limit:=2)</code> → an array of geometries (in WKT), for up to two regions spatially equal to the current feature

Further reading: *array manipulation*, *Select by location* algorithm

overlay_intersects

Returns whether the current feature spatially intersects at least one feature from a target layer, or an array of expression-based results for the features in the target layer intersected by the current feature.

Read more on the underlying GEOS "Intersects" predicate, as described in PostGIS *ST_Intersects* function.

语法	<code>overlay_intersects(layer, [expression], [filter], [limit], [cache=false])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - the layer whose overlay is checked • expression - an optional expression to evaluate on the features from the target layer. If not set, the function will just return a boolean indicating whether there is at least one match. • filter - an optional expression to filter the target features to check. If not set, all the features will be checked. • limit - an optional integer to limit the number of matching features. If not set, all the matching features will be returned. • cache - set this to true to build a local spatial index (most of the time, this is unwanted, unless you are working with a particularly slow data provider)
Examples	<ul style="list-style-type: none"> • <code>overlay_intersects('regions')</code> → true if the current feature spatially intersects a region • <code>overlay_intersects('regions', filter:= population > 10000)</code> → true if the current feature spatially intersects a region with a population greater than 10000 • <code>overlay_intersects('regions', name)</code> → an array of names, for the regions intersected by the current feature • <code>array_to_string(overlay_intersects('regions', name))</code> → a string as a comma separated list of names, for the regions intersected by the current feature • <code>array_sort(overlay_intersects(layer:='regions', expression:="name", filter:= population > 10000))</code> → an ordered array of names, for the regions intersected by the current feature and with a population greater than 10000 • <code>overlay_intersects(layer:='regions', expression:= geom_to_wkt(\$geometry), limit:=2)</code> → an array of geometries (in WKT), for up to two regions intersected by the current feature

Further reading: [intersects](#), [array manipulation](#), [Select by location algorithm](#)

overlay_nearest

Returns whether the current feature has feature(s) from a target layer within a given distance, or an array of expression-based results for the features in the target layer within a distance from the current feature.

Note: This function can be slow and consume a lot of memory for large layers.

语法	<code>overlay_nearest(layer, [expression], [filter], [limit=1], [max_distance], [cache=false])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - the target layer • expression - an optional expression to evaluate on the features from the target layer. If not set, the function will just return a boolean indicating whether there is at least one match. • filter - an optional expression to filter the target features to check. If not set, all the features in the target layer will be used. • limit - an optional integer to limit the number of matching features. If not set, only the nearest feature will be returned. If set to -1, returns all the matching features. • max_distance - an optional distance to limit the search of matching features. If not set, all the features in the target layer will be used. • cache - set this to true to build a local spatial index (most of the time, this is unwanted, unless you are working with a particularly slow data provider)
Examples	<ul style="list-style-type: none"> • <code>overlay_nearest('airports')</code> → true if the "airports" layer has at least one feature • <code>overlay_nearest('airports', max_distance:= 5000)</code> → true if there is an airport within a distance of 5000 map units from the current feature • <code>overlay_nearest('airports', name)</code> → the name of the closest airport to the current feature, as an array • <code>array_to_string(overlay_nearest('airports', name))</code> → the name of the closest airport to the current feature, as a string • <code>overlay_nearest(layer:='airports', expression:= name, max_distance:= 5000)</code> → the name of the closest airport within a distance of 5000 map units from the current feature, as an array • <code>overlay_nearest(layer:='airports', expression:="name", filter:= "Use"='Civilian', limit:=3)</code> → an array of names, for up to the three closest civilian airports ordered by distance • <code>overlay_nearest(layer:='airports', expression:="name", limit:= -1, max_distance:= 5000)</code> → an array of names, for all the airports within a distance of 5000 map units from the current feature, ordered by distance

Further reading: *array manipulation*, *Join attributes by nearest* algorithm

overlay_touches

Returns whether the current feature spatially touches at least one feature from a target layer, or an array of expression-based results for the features in the target layer touched by the current feature.

Read more on the underlying GEOS "Touches" predicate, as described in PostGIS [ST_Touches](#) function.

语法	<pre>overlay_touches(layer, [expression], [filter], [limit], [cache=false])</pre> <p>[] marks optional arguments</p>
参数	<ul style="list-style-type: none"> • layer - the layer whose overlay is checked • expression - an optional expression to evaluate on the features from the target layer. If not set, the function will just return a boolean indicating whether there is at least one match. • filter - an optional expression to filter the target features to check. If not set, all the features will be checked. • limit - an optional integer to limit the number of matching features. If not set, all the matching features will be returned. • cache - set this to true to build a local spatial index (most of the time, this is unwanted, unless you are working with a particularly slow data provider)
Examples	<ul style="list-style-type: none"> • <code>overlay_touches('regions')</code> → true if the current feature spatially touches a region • <code>overlay_touches('regions', filter:= population > 10000)</code> → true if the current feature spatially touches a region with a population greater than 10000 • <code>overlay_touches('regions', name)</code> → an array of names, for the regions touched by the current feature • <code>string_to_array(overlay_touches('regions', name))</code> → a string as a comma separated list of names, for the regions touched by the current feature • <code>array_sort(overlay_touches(layer:='regions', expression:="name", filter:= population > 10000))</code> → an ordered array of names, for the regions touched by the current feature and with a population greater than 10000 • <code>overlay_touches(layer:='regions', expression:= geom_to_wkt(\$geometry), limit:=2)</code> → an array of geometries (in WKT), for up to two regions touched by the current feature

Further reading: [touches](#), [array manipulation](#), [Select by location](#) algorithm

overlay_within

Returns whether the current feature is spatially within at least one feature from a target layer, or an array of expression-based results for the features in the target layer that contain the current feature.

Read more on the underlying GEOS "Within" predicate, as described in PostGIS [ST_Within](#) function.

语法	<code>overlay_within(layer, [expression], [filter], [limit], [cache=false])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - the layer whose overlay is checked • expression - an optional expression to evaluate on the features from the target layer. If not set, the function will just return a boolean indicating whether there is at least one match. • filter - an optional expression to filter the target features to check. If not set, all the features will be checked. • limit - an optional integer to limit the number of matching features. If not set, all the matching features will be returned. • cache - set this to true to build a local spatial index (most of the time, this is unwanted, unless you are working with a particularly slow data provider)
Examples	<ul style="list-style-type: none"> • <code>overlay_within('regions')</code> → true if the current feature is spatially within a region • <code>overlay_within('regions', filter:= population > 10000)</code> → true if the current feature is spatially within a region with a population greater than 10000 • <code>overlay_within('regions', name)</code> → an array of names, for the regions containing the current feature • <code>array_to_string(overlay_within('regions', name))</code> → a string as a comma separated list of names, for the regions containing the current feature • <code>array_sort(overlay_within(layer:='regions', expression:="name", filter:= population > 10000))</code> → an ordered array of names, for the regions containing the current feature and with a population greater than 10000 • <code>overlay_within(layer:='regions', expression:= geom_to_wkt(\$geometry), limit:=2)</code> → an array of geometries (in WKT), for up to two regions containing the current feature

Further reading: *within*, *array manipulation*, *Select by location* algorithm

\$perimeter

Returns the perimeter length of the current feature. The perimeter calculated by this function respects both the current project's ellipsoid setting and distance unit settings. For example, if an ellipsoid has been set for the project then the calculated perimeter will be ellipsoidal, and if no ellipsoid is set then the calculated perimeter will be planimetric.

语法	<code>\$perimeter</code>
Examples	<ul style="list-style-type: none"> • <code>\$perimeter</code> → 42

perimeter

Returns the perimeter of a geometry polygon object. Calculations are always planimetric in the Spatial Reference System (SRS) of this geometry, and the units of the returned perimeter will match the units for the SRS. This differs from the calculations performed by the \$perimeter function, which will perform ellipsoidal calculations based on the project's ellipsoid and distance unit settings.

语法	perimeter(geometry)
参数	<ul style="list-style-type: none"> • geometry - polygon geometry object
Examples	<ul style="list-style-type: none"> • <code>perimeter(geom_from_wkt('POLYGON((0 0, 4 0, 4 2, 0 2, 0 0)))') → 12.0</code>

point_n

Returns a specific node from a geometry.

语法	point_n(geometry, index)
参数	<ul style="list-style-type: none"> • geometry - geometry object • index - index of node to return, where 1 is the first node; if the value is negative, the selected vertex index will be its total count minus the absolute value
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(point_n(geom_from_wkt('POLYGON((0 0, 4 0, 4 2, 0 2, 0 0)))', 2)) → 'Point(4 0)'</code>

Further reading: [Extract specific vertices](#) algorithm

point_on_surface

Returns a point guaranteed to lie on the surface of a geometry.

语法	point_on_surface(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>point_on_surface(\$geometry) → a point geometry</code>

Further reading: [Point on Surface](#) algorithm

pole_of_inaccessibility

Calculates the approximate pole of inaccessibility for a surface, which is the most distant internal point from the boundary of the surface. This function uses the 'polylabel' algorithm (Vladimir Agafonkin, 2016), which is an iterative approach guaranteed to find the true pole of inaccessibility within a specified tolerance. More precise tolerances require more iterations and will take longer to calculate.

语法	pole_of_inaccessibility(geometry, tolerance)
参数	<ul style="list-style-type: none"> • geometry - a geometry • tolerance - maximum distance between the returned point and the true pole location
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(pole_of_inaccessibility(geom_from_wkt('POLYGON((0 1, 0 9, 3 10, 3 3, 10 3, 10 1, 0 1))'), 0.1))' → 'Point(1.546875 2.546875)'</code>

Further reading: *Pole of inaccessibility* algorithm

project

Returns a point projected from a start point using a distance, a bearing (azimuth) and an elevation in radians.

语法	project(point, distance, azimuth, [elevation]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • point - start point • distance - distance to project • azimuth - azimuth in radians clockwise, where 0 corresponds to north • elevation - angle of inclination in radians
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(project(make_point(1, 2), 3, radians(270)))</code> → <code>'Point(-2, 2)'</code>

Further reading: *Project points (Cartesian)* algorithm

relate

Tests the Dimensional Extended 9 Intersection Model (DE-9IM) representation of the relationship between two geometries.

Relationship variant

Returns the Dimensional Extended 9 Intersection Model (DE-9IM) representation of the relationship between two geometries.

语法	relate(geometry, geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>relate(geom_from_wkt('LINESTRING(40 40,120 120)'), geom_from_wkt('LINESTRING(40 40,60 120)'))</code> → <code>'FF1F00102'</code>

Pattern match variant

Tests whether the DE-9IM relationship between two geometries matches a specified pattern.

语法	relate(geometry, geometry, pattern)
参数	<ul style="list-style-type: none"> • geometry - a geometry • geometry - a geometry • pattern - DE-9IM pattern to match
Examples	<ul style="list-style-type: none"> • <code>relate(geom_from_wkt('LINESTRING(40 40,120 120)'), geom_from_wkt('LINESTRING(40 40,60 120)'), '**1F001**')</code> → <code>True</code>

reverse

Reverses the direction of a line string by reversing the order of its vertices.

语法	reverse(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(reverse(geom_from_wkt('LINESTRING(0 0, 1 1, 2 2)')))</code> → <code>'LINESTRING(2 2, 1 1, 0 0)'</code>

Further reading: [Reverse line direction](#) algorithm

rotate

Returns a rotated version of a geometry. Calculations are in the Spatial Reference System of this geometry.

语法	rotate(geometry, rotation, [center]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - a geometry • rotation - clockwise rotation in degrees • center - rotation center point. If not specified, the center of the geometry's bounding box is used.
Examples	<ul style="list-style-type: none"> • <code>rotate(\$geometry, 45, make_point(4, 5))</code> → geometry rotated 45 degrees clockwise around the (4, 5) point • <code>rotate(\$geometry, 45)</code> → geometry rotated 45 degrees clockwise around the center of its bounding box

segments_to_lines

Returns a multi line geometry consisting of a line for every segment in the input geometry.

语法	segments_to_lines(geometry)
参数	<ul style="list-style-type: none"> • geometry - geometry object
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(segments_to_lines(geom_from_wkt('LINESTRING(0 0, 1 1, 2 2)')))</code> → <code>'MultiLineString((0 0, 1 1),(1 1, 2 2))'</code>

Further reading: [Explode lines](#) algorithm

shortest_line

Returns the shortest line joining geometry1 to geometry2. The resultant line will start at geometry1 and end at geometry2.

语法	shortest_line(geometry1, geometry2)
参数	<ul style="list-style-type: none"> • geometry1 - geometry to find shortest line from • geometry2 - geometry to find shortest line to
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(shortest_line(geom_from_wkt('LINESTRING (20 80, 98 190, 110 180, 50 75)'), geom_from_wkt('POINT(100 100)))) → 'LineString(73.0769 115.384, 100 100)'</code>

simplify

Simplifies a geometry by removing nodes using a distance based threshold (ie, the Douglas Peucker algorithm). The algorithm preserves large deviations in geometries and reduces the number of vertices in nearly straight segments.

语法	simplify(geometry, tolerance)
参数	<ul style="list-style-type: none"> • geometry - a geometry • tolerance - maximum deviation from straight segments for points to be removed
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(simplify(geometry:=geom_from_wkt('LineString(0 0, 5 0.1, 10 0)'), tolerance:=5)) → 'LineString(0 0, 10 0)'</code>

Further reading: [Simplify](#) algorithm

simplify_vw

Simplifies a geometry by removing nodes using an area based threshold (ie, the Visvalingam-Whyatt algorithm). The algorithm removes vertices which create small areas in geometries, e.g., narrow spikes or nearly straight segments.

语法	simplify_vw(geometry, tolerance)
参数	<ul style="list-style-type: none"> • geometry - a geometry • tolerance - a measure of the maximum area created by a node for the node to be removed
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(simplify_vw(geometry:=geom_from_wkt('LineString(0 0, 5 0, 5.01 10, 5.02 0, 10 0)'), tolerance:=5)) → 'LineString(0 0, 10 0)'</code>

Further reading: [Simplify](#) algorithm

single_sided_buffer

Returns a geometry formed by buffering out just one side of a linestring geometry. Distances are in the Spatial Reference System of this geometry.

语法	single_sided_buffer(geometry, distance, [segments=8], [join=1], [miter_limit=2.0]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - a (multi)linestring geometry • distance - buffer distance. Positive values will be buffered to the left of lines, negative values to the right • segments - number of segments to use to represent a quarter circle when a round join style is used. A larger number results in a smoother buffer with more nodes. • join - join style for corners, where 1 = round, 2 = miter and 3 = bevel • miter_limit - limit on the miter ratio used for very sharp corners (when using miter joins only)
Examples	<ul style="list-style-type: none"> • single_sided_buffer(\$geometry, 10.5) → line buffered to the left by 10.5 units • single_sided_buffer(\$geometry, -10.5) → line buffered to the right by 10.5 units • single_sided_buffer(\$geometry, 10.5, segments=16, join=1) → line buffered to the left by 10.5 units, using more segments to result in a smoother buffer • single_sided_buffer(\$geometry, 10.5, join=3) → line buffered to the left by 10.5 units, using a beveled join

Further reading: *Single sided buffer* algorithm

smooth

Smooths a geometry by adding extra nodes which round off corners in the geometry. If input geometries contain Z or M values, these will also be smoothed and the output geometry will retain the same dimensionality as the input geometry.

语法	smooth(geometry, [iterations=1], [offset=0.25], [min_length=-1], [max_angle=180]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - a geometry • iterations - number of smoothing iterations to apply. Larger numbers result in smoother but more complex geometries. • offset - value between 0 and 0.5 which controls how tightly the smoothed geometry follow the original geometry. Smaller values result in a tighter smoothing, larger values result in looser smoothing. • min_length - minimum length of segments to apply smoothing to. This parameter can be used to avoid placing excessive additional nodes in shorter segments of the geometry. • max_angle - maximum angle at node for smoothing to be applied (0-180). By lowering the maximum angle intentionally sharp corners in the geometry can be preserved. For instance, a value of 80 degrees will retain right angles in the geometry.
Examples	<ul style="list-style-type: none"> • geom_to_wkt(smooth(geometry:=geom_from_wkt('LineString(0 0, 5 0, 5 5)'), iterations:=1, offset:=0.2, min_length:=-1, max_angle:=180)) → 'LineString(0 0, 4 0, 5 1, 5 5)'

Further reading: *Smooth* algorithm

start_point

Returns the first node from a geometry.

语法	start_point(geometry)
参数	<ul style="list-style-type: none"> • geometry - geometry object
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(start_point(geom_from_wkt('LINESTRING(4 0, 4 2, 0 2)')))</code> → 'Point (4 0)'

Further reading: *Extract specific vertices* algorithm

sym_difference

Returns a geometry that represents the portions of two geometries that do not intersect.

语法	sym_difference(geometry1, geometry2)
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(sym_difference(geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)'), geom_from_wkt('LINESTRING(3 3, 8 8)')))</code> → 'LINESTRING(5 5, 8 8)'

Further reading: *Symmetrical difference* algorithm

tapered_buffer

Creates a buffer along a line geometry where the buffer diameter varies evenly over the length of the line.

语法	tapered_buffer(geometry, start_width, end_width, [segments=8]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • geometry - input geometry. Must be a (multi)line geometry. • start_width - width of buffer at start of line, • end_width - width of buffer at end of line. • segments - number of segments to approximate quarter-circle curves in the buffer.
Examples	<ul style="list-style-type: none"> • <code>tapered_buffer(geometry:=geom_from_wkt('LINESTRING(1 2, 4 2)'), start_width:=1, end_width:=2, segments:=8)</code> → A tapered buffer starting with a diameter of 1 and ending with a diameter of 2 along the linestring geometry.

Further reading: *Tapered buffers* algorithm

touches

Tests whether a geometry touches another. Returns true if the geometries have at least one point in common, but their interiors do not intersect.

语法	touches(geometry1, geometry2)
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • touches(geom_from_wkt('LINESTRING(5 3, 4 4)'), geom_from_wkt('LINESTRING(3 3, 4 4, 5 5)')) → true • touches(geom_from_wkt('POINT(4 4)'), geom_from_wkt('POINT(5 5)')) → false

Further reading: [overlay_touches](#)

transform

Returns the geometry transformed from a source CRS to a destination CRS.

语法	transform(geometry, source_auth_id, dest_auth_id)
参数	<ul style="list-style-type: none"> • geometry - a geometry • source_auth_id - the source auth CRS ID • dest_auth_id - the destination auth CRS ID
Examples	<ul style="list-style-type: none"> • geom_to_wkt(transform(make_point(488995.53240249, 7104473.38600835), 'EPSG:2154', 'EPSG:4326')) → 'POINT(0 51)'

Further reading: [Reproject layer](#) algorithm

translate

Returns a translated version of a geometry. Calculations are in the Spatial Reference System of this geometry.

语法	translate(geometry, dx, dy)
参数	<ul style="list-style-type: none"> • geometry - a geometry • dx - delta x • dy - delta y
Examples	<ul style="list-style-type: none"> • translate(\$geometry, 5, 10) → a geometry of the same type like the original one

Further reading: [Translate](#) algorithm

union

Returns a geometry that represents the point set union of the geometries.

语法	<code>union(geometry1, geometry2)</code>
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>geom_to_wkt(union(make_point(4, 4), make_point(5, 5))) → 'MULTIPOINT(4 4, 5 5)'</code>

wedge_buffer

Returns a wedge shaped buffer originating from a point geometry.

语法	<code>wedge_buffer(center, azimuth, width, outer_radius, [inner_radius=0.0])</code> [] marks optional arguments
参数	<ul style="list-style-type: none"> • center - center point (origin) of buffer. Must be a point geometry. • azimuth - angle (in degrees) for the middle of the wedge to point. • width - buffer width (in degrees). Note that the wedge will extend to half of the angular width either side of the azimuth direction. • outer_radius - outer radius for buffers • inner_radius - optional inner radius for buffers
Examples	<ul style="list-style-type: none"> • <code>wedge_buffer(center:=geom_from_wkt('POINT(1 2)'), azimuth:=90, width:=180, outer_radius:=1) → A wedge shaped buffer centered on the point (1,2), facing to the East, with a width of 180 degrees and outer radius of 1.</code>

Further reading: [Create wedge buffers](#) algorithm

within

Tests whether a geometry is within another. Returns true if the geometry1 is completely within geometry2.

语法	<code>within(geometry1, geometry2)</code>
参数	<ul style="list-style-type: none"> • geometry1 - a geometry • geometry2 - a geometry
Examples	<ul style="list-style-type: none"> • <code>within(geom_from_wkt('POINT(0.5 0.5)'), geom_from_wkt('POLYGON((0 0, 0 1, 1 1, 1 0, 0 0))')) → true</code> • <code>within(geom_from_wkt('POINT(5 5)'), geom_from_wkt('POLYGON((0 0, 0 1, 1 1, 1 0, 0 0))')) → false</code>

Further reading: [overlay_within](#)

\$x

Returns the x coordinate of the current point feature. If the feature is a multipoint feature, then the x-coordinate of the first point will be returned.

语法	\$x
Examples	<ul style="list-style-type: none"> • <code>\$x</code> → 42

x

Returns the x coordinate of a point geometry, or the x coordinate of the centroid for a non-point geometry.

语法	x(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>x(geom_from_wkt('POINT(2 5)'))</code> → 2 • <code>x(\$geometry)</code> → x coordinate of the current feature's centroid

\$x_at

Retrieves a x coordinate of the current feature's geometry.

语法	\$x_at(i)
参数	<ul style="list-style-type: none"> • i - index of point of a line (indices start at 0; negative values apply from the last index, starting at -1)
Examples	<ul style="list-style-type: none"> • <code>\$x_at(1)</code> → 5

x_max

Returns the maximum x coordinate of a geometry. Calculations are in the spatial reference system of this geometry.

语法	x_max(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • <code>x_max(geom_from_wkt('LINESTRING(2 5, 3 6, 4 8)'))</code> → 4

x_min

Returns the minimum x coordinate of a geometry. Calculations are in the spatial reference system of this geometry.

语法	x_min(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • x_min(geom_from_wkt('LINESTRING(2 5, 3 6, 4 8)')) → 2

\$y

Returns the y coordinate of the current point feature. If the feature is a multipoint feature, then the y-coordinate of the first point will be returned.

语法	\$y
Examples	<ul style="list-style-type: none"> • \$y → 42

y

Returns the y coordinate of a point geometry, or the y coordinate of the centroid for a non-point geometry.

语法	y(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • y(geom_from_wkt('POINT(2 5)')) → 5 • y(\$geometry) → y coordinate of the current feature's centroid

\$y_at

Retrieves a y coordinate of the current feature's geometry.

语法	\$y_at(i)
参数	<ul style="list-style-type: none"> • i - index of point of a line (indices start at 0; negative values apply from the last index, starting at -1)
Examples	<ul style="list-style-type: none"> • \$y_at(1) → 2

y_max

Returns the maximum y coordinate of a geometry. Calculations are in the spatial reference system of this geometry.

语法	y_max(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • y_max(geom_from_wkt('LINESTRING(2 5, 3 6, 4 8)')) → 8

y_min

Returns the minimum y coordinate of a geometry. Calculations are in the spatial reference system of this geometry.

语法	y_min(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry
Examples	<ul style="list-style-type: none"> • y_min(geom_from_wkt('LINESTRING(2 5, 3 6, 4 8)')) → 5

z

Returns the z coordinate of a point geometry, or NULL if the geometry has no z value.

语法	z(geometry)
参数	<ul style="list-style-type: none"> • geometry - a point geometry
Examples	<ul style="list-style-type: none"> • z(geom_from_wkt('POINTZ(2 5 7)')) → 7

z_max

Returns the maximum z coordinate of a geometry, or NULL if the geometry has no z value.

语法	z_max(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry with z coordinate
Examples	<ul style="list-style-type: none"> • z_max(geom_from_wkt('POINT (0 0 1)')) → 1 • z_max(geom_from_wkt('MULTIPOINT (0 0 1 , 1 1 3)')) → 3 • z_max(make_line(make_point(0,0,0), make_point(-1,-1,-2))) → 0 • z_max(geom_from_wkt('LINESTRING(0 0 0, 1 0 2, 1 1 -1)')) → 2 • z_max(geom_from_wkt('POINT (0 0)')) → NULL

z_min

Returns the minimum z coordinate of a geometry, or NULL if the geometry has no z value.

语法	z_min(geometry)
参数	<ul style="list-style-type: none"> • geometry - a geometry with z coordinate
Examples	<ul style="list-style-type: none"> • z_min(geom_from_wkt('POINT (0 0 1)')) → 1 • z_min(geom_from_wkt('MULTIPOINT (0 0 1 , 1 1 3)')) → 1 • z_min(make_line(make_point(0,0,0), make_point(-1,-1,-2))) → -2 • z_min(geom_from_wkt('LINESTRING(0 0 0, 1 0 2, 1 1 -1)')) → -1 • z_min(geom_from_wkt('POINT (0 0)')) → NULL

14.3.14 Layout Functions

This group contains functions to manipulate print layout items properties.

- *item_variables*

item_variables

Returns a map of variables from a layout item inside this print layout.

语法	item_variables(id)
参数	<ul style="list-style-type: none"> • id - layout item ID
Examples	<ul style="list-style-type: none"> • map_get(item_variables('Map 0'), 'map_scale') → scale of the item 'Map 0' in the current print layout

Further reading: List of default *variables*

14.3.15 Map Layers

This group contains a list of the available layers in the current project. This offers a convenient way to write expressions referring to multiple layers, such as when performing *aggregates*, *attribute* or *spatial* queries.

It also provides some convenient functions to manipulate layers.

- *decode_uri*

decode_uri

Takes a layer and decodes the uri of the underlying data provider. It depends on the dataprovider, which data is available.

语法	decode_uri(layer, [part]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - The layer for which the uri should be decoded. • part - The part of the uri to return. If unspecified, a map with all uri parts will be returned.
Examples	<ul style="list-style-type: none"> • <code>decode_uri(@layer)</code> → {'layerId': '0', 'layerName': '', 'path': '/home/qgis/shapefile.shp'} • <code>decode_uri(@layer)</code> → {'layerId': NULL, 'layerName': 'layer', 'path': '/home/qgis/geopackage.gpkg'} • <code>decode_uri(@layer, 'path')</code> → 'C:\my_data\qgis\shape.shp'

14.3.16 Maps Functions

This group contains functions to create or manipulate keys and values of map data structures (also known as dictionary objects, key-value pairs, or associative arrays). Unlike the *list data structure* where values order matters, the order of the key-value pairs in the map object is not relevant and values are identified by their keys.

- *from_json*
- *hstore_to_map*
- *json_to_map*
- *map*
- *map_akeys*
- *map_aval*
- *map_concat*
- *map_delete*
- *map_exist*
- *map_get*
- *map_insert*
- *map_to_hstore*
- *map_to_json*
- *to_json*

from_json

Loads a JSON formatted string.

语法	from_json(string)
参数	<ul style="list-style-type: none"> • string - JSON string
Examples	<ul style="list-style-type: none"> • from_json('{ "qgis": "rocks" }') → { 'qgis': 'rocks' } • from_json('[1, 2, 3]') → [1,2,3]

hstore_to_map

Creates a map from a hstore-formatted string.

语法	hstore_to_map(string)
参数	<ul style="list-style-type: none"> • string - the input string
Examples	<ul style="list-style-type: none"> • hstore_to_map('qgis=>rocks') → { 'qgis': 'rocks' }

json_to_map

Creates a map from a json-formatted string.

语法	json_to_map(string)
参数	<ul style="list-style-type: none"> • string - the input string
Examples	<ul style="list-style-type: none"> • json_to_map('{ "qgis": "rocks" }') → { 'qgis': 'rocks' }

map

Returns a map containing all the keys and values passed as pair of parameters.

语法	map(key1, value1, key2, value2, ...)
参数	<ul style="list-style-type: none"> • key - a key (string) • value - a value
Examples	<ul style="list-style-type: none"> • map('1', 'one', '2', 'two') → { '1': 'one', '2': 'two' }

map_akeys

Returns all the keys of a map as an array.

语法	map_akeys(map)
参数	<ul style="list-style-type: none"> • map - a map
Examples	<ul style="list-style-type: none"> • map_akeys (map ('1', 'one', '2', 'two')) → ['1', '2']

map_aval

Returns all the values of a map as an array.

语法	map_aval(map)
参数	<ul style="list-style-type: none"> • map - a map
Examples	<ul style="list-style-type: none"> • map_aval (map ('1', 'one', '2', 'two')) → ['one', 'two']

map_concat

Returns a map containing all the entries of the given maps. If two maps contain the same key, the value of the second map is taken.

语法	map_concat(map1, map2, ...)
参数	<ul style="list-style-type: none"> • map - a map
Examples	<ul style="list-style-type: none"> • map_concat (map ('1', 'one', '2', 'overridden'), map ('2', 'two', '3', 'three')) → { '1': 'one', '2': 'two', '3': 'three' }

map_delete

Returns a map with the given key and its corresponding value deleted.

语法	map_delete(map, key)
参数	<ul style="list-style-type: none"> • map - a map • key - the key to delete
Examples	<ul style="list-style-type: none"> • map_delete (map ('1', 'one', '2', 'two'), '2') → { '1': 'one' }

map_exist

Returns true if the given key exists in the map.

语法	map_exist(map, key)
参数	<ul style="list-style-type: none"> • map - a map • key - the key to lookup
Examples	<ul style="list-style-type: none"> • map_exist (map('1', 'one', '2', 'two'), '3') → false

map_get

Returns the value of a map, given its key. Returns NULL if the key does not exist.

语法	map_get(map, key)
参数	<ul style="list-style-type: none"> • map - a map • key - the key to lookup
Examples	<ul style="list-style-type: none"> • map_get (map('1', 'one', '2', 'two'), '2') → 'two' • map_get (item_variables('Map 0'), 'map_scale') → scale of the item 'Map 0' (if it exists) in the current print layout

map_insert

Returns a map with an added key/value. If the key already exists, its value is overridden.

语法	map_insert(map, key, value)
参数	<ul style="list-style-type: none"> • map - a map • key - the key to add • value - the value to add
Examples	<ul style="list-style-type: none"> • map_insert (map('1', 'one'), '3', 'three') → { '1': 'one', '3': 'three' } • map_insert (map('1', 'one', '2', 'overridden'), '2', 'two') → { '1': 'one', '2': 'two' }

map_to_hstore

Merge map elements into a hstore-formatted string.

语法	map_to_hstore(map)
参数	<ul style="list-style-type: none"> • map - the input map
Examples	<ul style="list-style-type: none"> • map_to_hstore (map('qgis', 'rocks')) → "qgis=>rocks"

map_to_json

Merge map elements into a json-formatted string.

语法	map_to_json(map)
参数	<ul style="list-style-type: none"> • map - the input map
Examples	<ul style="list-style-type: none"> • <code>map_to_json(map('qgis', 'rocks'))</code> → <code>{"qgis": "rocks"}</code>

to_json

Create a JSON formatted string from a map, array or other value.

语法	to_json(value)
参数	<ul style="list-style-type: none"> • value - The input value
Examples	<ul style="list-style-type: none"> • <code>to_json(map('qgis', 'rocks'))</code> → <code>{"qgis": "rocks"}</code> • <code>to_json(array(1, 2, 3))</code> → <code>[1,2,3]</code>

14.3.17 Mathematical Functions

This group contains math functions (e.g., square root, sin and cos).

<ul style="list-style-type: none"> • <i>abs</i> • <i>acos</i> • <i>asin</i> • <i>atan</i> • <i>atan2</i> • <i>azimuth</i> • <i>ceil</i> • <i>clamp</i> • <i>cos</i> • <i>degrees</i> • <i>exp</i> • <i>floor</i> • <i>inclination</i> • <i>ln</i> • <i>log</i> • <i>log10</i> • <i>max</i> • 最小值
--

- *pi*
- *radians*
- *rand*
- *randf*
- *round*
- *scale_exp*
- *scale_linear*
- *sin*
- *sqrt*
- *tan*

abs

Returns the absolute value of a number.

语法	abs(value)
参数	<ul style="list-style-type: none"> • value - a number
Examples	<ul style="list-style-type: none"> • <code>abs(-2) → 2</code>

acos

Returns the inverse cosine of a value in radians.

语法	acos(value)
参数	<ul style="list-style-type: none"> • value - cosine of an angle in radians
Examples	<ul style="list-style-type: none"> • <code>acos(0.5) → 1.0471975511966</code>

asin

Returns the inverse sine of a value in radians.

语法	asin(value)
参数	<ul style="list-style-type: none"> • value - sine of an angle in radians
Examples	<ul style="list-style-type: none"> • <code>asin(1.0) → 1.5707963267949</code>

atan

Returns the inverse tangent of a value in radians.

语法	atan(value)
参数	<ul style="list-style-type: none"> • value - tan of an angle in radians
Examples	<ul style="list-style-type: none"> • atan(0.5) → 0.463647609000806

atan2

Returns the inverse tangent of dy/dx by using the signs of the two arguments to determine the quadrant of the result.

语法	atan2(dy, dx)
参数	<ul style="list-style-type: none"> • dy - y coordinate difference • dx - x coordinate difference
Examples	<ul style="list-style-type: none"> • atan2(1.0, 1.732) → 0.523611477769969

azimuth

Returns the north-based azimuth as the angle in radians measured clockwise from the vertical on point_a to point_b.

语法	azimuth(point_a, point_b)
参数	<ul style="list-style-type: none"> • point_a - point geometry • point_b - point geometry
Examples	<ul style="list-style-type: none"> • degrees(azimuth(make_point(25, 45), make_point(75, 100))) → 42.273689 • degrees(azimuth(make_point(75, 100), make_point(25, 45))) → 222.273689

ceil

Rounds a number upwards.

语法	ceil(value)
参数	<ul style="list-style-type: none"> • value - a number
Examples	<ul style="list-style-type: none"> • ceil(4.9) → 5 • ceil(-4.9) → -4

clamp

Restricts an input value to a specified range.

语法	clamp(minimum, input, maximum)
参数	<ul style="list-style-type: none"> • minimum - the smallest value <i>input</i> is allowed to take. • input - a value which will be restricted to the range specified by <i>minimum</i> and <i>maximum</i> • maximum - the largest value <i>input</i> is allowed to take
Examples	<ul style="list-style-type: none"> • clamp(1, 5, 10) → 5 <i>input</i> is between 1 and 10 so is returned unchanged • clamp(1, 0, 10) → 1 <i>input</i> is less than minimum value of 1, so function returns 1 • clamp(1, 11, 10) → 10 <i>input</i> is greater than maximum value of 10, so function returns 10

cos

Returns cosine of an angle.

语法	cos(angle)
参数	<ul style="list-style-type: none"> • angle - angle in radians
Examples	<ul style="list-style-type: none"> • cos(1.571) → 0.000796326710733263

degrees

Converts from radians to degrees.

语法	degrees(radians)
参数	<ul style="list-style-type: none"> • radians - numeric value
Examples	<ul style="list-style-type: none"> • degrees(3.14159) → 180 • degrees(1) → 57.2958

exp

Returns exponential of an value.

语法	exp(value)
参数	<ul style="list-style-type: none"> • value - number to return exponent of
Examples	<ul style="list-style-type: none"> • exp(1.0) → 2.71828182845905

floor

Rounds a number downwards.

语法	floor(value)
参数	<ul style="list-style-type: none"> • value - a number
Examples	<ul style="list-style-type: none"> • floor(4.9) → 4 • floor(-4.9) → -5

inclination

Returns the inclination measured from the zenith (0) to the nadir (180) on point_a to point_b.

语法	inclination(point_a, point_b)
参数	<ul style="list-style-type: none"> • point_a - point geometry • point_b - point geometry
Examples	<ul style="list-style-type: none"> • inclination(make_point(5, 10, 0), make_point(5, 10, 5)) → 0.0 • inclination(make_point(5, 10, 0), make_point(5, 10, 0)) → 90.0 • inclination(make_point(5, 10, 0), make_point(50, 100, 0)) → 90.0 • inclination(make_point(5, 10, 0), make_point(5, 10, -5)) → 180.0

ln

Returns the natural logarithm of a value.

语法	ln(value)
参数	<ul style="list-style-type: none"> • value - numeric value
Examples	<ul style="list-style-type: none"> • ln(1) → 0 • ln(2.7182818284590452354) → 1

log

Returns the value of the logarithm of the passed value and base.

语法	log(base, value)
参数	<ul style="list-style-type: none"> • base - any positive number • value - any positive number
Examples	<ul style="list-style-type: none"> • log(2, 32) → 5 • log(0.5, 32) → -5

log10

Returns the value of the base 10 logarithm of the passed expression.

语法	log10(value)
参数	<ul style="list-style-type: none"> • value - any positive number
Examples	<ul style="list-style-type: none"> • log10(1) → 0 • log10(100) → 2

max

Returns the largest value in a set of values.

语法	max(value1, value2, ...)
参数	<ul style="list-style-type: none"> • value - a number
Examples	<ul style="list-style-type: none"> • max(2, 10.2, 5.5) → 10.2 • max(20.5, NULL, 6.2) → 20.5

最小值

Returns the smallest value in a set of values.

语法	min(value1, value2, ...)
参数	<ul style="list-style-type: none"> • value - a number
Examples	<ul style="list-style-type: none"> • min(20.5, 10, 6.2) → 6.2 • min(2, -10.3, NULL) → -10.3

pi

Returns value of pi for calculations.

语法	pi()
Examples	<ul style="list-style-type: none"> • pi() → 3.14159265358979

radians

Converts from degrees to radians.

语法	radians(degrees)
参数	<ul style="list-style-type: none"> • degrees - numeric value
Examples	<ul style="list-style-type: none"> • radians(180) → 3.14159 • radians(57.2958) → 1

rand

Returns a random integer within the range specified by the minimum and maximum argument (inclusive). If a seed is provided, the returned will always be the same, depending on the seed.

语法	rand(min, max, [seed=NULL]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • min - an integer representing the smallest possible random number desired • max - an integer representing the largest possible random number desired • seed - any value to use as seed
Examples	<ul style="list-style-type: none"> • rand(1, 10) → 8

randf

Returns a random float within the range specified by the minimum and maximum argument (inclusive). If a seed is provided, the returned will always be the same, depending on the seed.

语法	randf([min=0.0], [max=1.0], [seed=NULL]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • min - an float representing the smallest possible random number desired • max - an float representing the largest possible random number desired • seed - any value to use as seed
Examples	<ul style="list-style-type: none"> • randf(1, 10) → 4.59258286403147

round

Rounds a number to number of decimal places.

语法	round(value, [places=0]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • value - decimal number to be rounded • places - Optional integer representing number of places to round decimals to. Can be negative.
Examples	<ul style="list-style-type: none"> • round(1234.567, 2) → 1234.57 • round(1234.567) → 1235

scale_exp

Transforms a given value from an input domain to an output range using an exponential curve. This function can be used to ease values in or out of the specified output range.

语法	scale_exp(value, domain_min, domain_max, range_min, range_max, exponent)
参数	<ul style="list-style-type: none"> • value - A value in the input domain. The function will return a corresponding scaled value in the output range. • domain_min - Specifies the minimum value in the input domain, the smallest value the input value should take. • domain_max - Specifies the maximum value in the input domain, the largest value the input value should take. • range_min - Specifies the minimum value in the output range, the smallest value which should be output by the function. • range_max - Specifies the maximum value in the output range, the largest value which should be output by the function. • exponent - A positive value (greater than 0), which dictates the way input values are mapped to the output range. Large exponents will cause the output values to 'ease in', starting slowly before accelerating as the input values approach the domain maximum. Smaller exponents (less than 1) will cause output values to 'ease out', where the mapping starts quickly but slows as it approaches the domain maximum.
Examples	<ul style="list-style-type: none"> • scale_exp(5, 0, 10, 0, 100, 2) → 25 easing in, using an exponent of 2 • scale_exp(3, 0, 10, 0, 100, 0.5) → 54.772 easing out, using an exponent of 0.5

scale_linear

Transforms a given value from an input domain to an output range using linear interpolation.

语法	<code>scale_linear(value, domain_min, domain_max, range_min, range_max)</code>
参数	<ul style="list-style-type: none"> • value - A value in the input domain. The function will return a corresponding scaled value in the output range. • domain_min - Specifies the minimum value in the input domain, the smallest value the input value should take. • domain_max - Specifies the maximum value in the input domain, the largest value the input value should take. • range_min - Specifies the minimum value in the output range, the smallest value which should be output by the function. • range_max - Specifies the maximum value in the output range, the largest value which should be output by the function.
Examples	<ul style="list-style-type: none"> • <code>scale_linear(5, 0, 10, 0, 100) → 50</code> • <code>scale_linear(0.2, 0, 1, 0, 360) → 72</code> scaling a value between 0 and 1 to an angle between 0 and 360 • <code>scale_linear(1500, 1000, 10000, 9, 20) → 9.6111111</code> scaling a population which varies between 1000 and 10000 to a font size between 9 and 20

sin

Returns the sine of an angle.

语法	<code>sin(angle)</code>
参数	<ul style="list-style-type: none"> • angle - angle in radians
Examples	<ul style="list-style-type: none"> • <code>sin(1.571) → 0.999999682931835</code>

sqrt

Returns square root of a value.

语法	<code>sqrt(value)</code>
参数	<ul style="list-style-type: none"> • value - a number
Examples	<ul style="list-style-type: none"> • <code>sqrt(9) → 3</code>

tan

Returns the tangent of an angle.

语法	tan(angle)
参数	<ul style="list-style-type: none"> angle - angle in radians
Examples	<ul style="list-style-type: none"> tan(1.0) → 1.5574077246549

14.3.18 Operators

This group contains operators (e.g., +, -, *). Note that for most of the mathematical functions below, if one of the inputs is NULL then the result is NULL.

Function	Description
a + b	Addition of two values (a plus b)
a - b	Subtraction of two values (a minus b).
a * b	Multiplication of two values (a multiplied by b)
a / b	Division of two values (a divided by b)
a % b	Remainder of division of a by b (eg, 7 % 2 = 1, or 2 fits into 7 three times with remainder 1)
a ^ b	Power of two values (for example, 2^2=4 or 2^3=8)
a < b	Compares two values and evaluates to 1 if the left value is less than the right value (a is smaller than b)
a <= b	Compares two values and evaluates to 1 if the left value is less than or equal to the right value
a <> b	Compares two values and evaluates to 1 if they are not equal
a = b	Compares two values and evaluates to 1 if they are equal
a != b	a and b are not equal
a > b	Compares two values and evaluates to 1 if the left value is greater than the right value (a is larger than b)
a >= b	Compares two values and evaluates to 1 if the left value is greater than or equal to the right value
a ~ b	a matches the regular expression b
	Joins two values together into a string. If one of the values is NULL the result will be NULL
`\n`	Inserts a new line in a string
LIKE	Returns 1 if the first parameter matches the supplied pattern
ILIKE	Returns 1 if the first parameter matches case-insensitive the supplied pattern (ILIKE can be used instead of LIKE to make the match case-insensitive)
a IS b	Tests whether two values are identical. Returns 1 if a is the same as b
a OR b	Returns 1 when condition a or condition b is true
a AND b	Returns 1 when conditions a and b are true
NOT	Negates a condition
"Column_name"	Value of the field <i>Column_name</i> , take care to not be confused with simple quote, see below
'string'	a string value, take care to not be confused with double quote, see above
NULL	null value
a IS NULL	a has no value
a IS NOT NULL	a has a value
a IN (value[,value])	a is below the values listed
a NOT IN (value[,value])	a is not below the values listed

Some examples:

- Joins a string and a value from a column name:

```
'My feature''s id is: ' || "gid"
```

- Test if the "description" attribute field starts with the 'Hello' string in the value (note the position of the % character):

```
"description" LIKE 'Hello%'
```

14.3.19 Processing Functions

This group contains functions that operate on processing algorithms.

- *parameter*

parameter

Returns the value of a processing algorithm input parameter.

语法	parameter(name)
参数	<ul style="list-style-type: none">• name - name of the corresponding input parameter
Examples	<ul style="list-style-type: none">• <code>parameter('BUFFER_SIZE') → 5.6</code>

14.3.20 Rasters Functions

This group contains functions to operate on raster layer.

- *raster_statistic*
- *raster_value*

raster_statistic

Returns statistics from a raster layer.

语法	raster_statistic(layer, band, property)
参数	<ul style="list-style-type: none"> • layer - a string, representing either a raster layer name or layer ID • band - integer representing the band number from the raster layer, starting at 1 • property - a string corresponding to the property to return. Valid options are: <ul style="list-style-type: none"> - min: minimum value - max: maximum value - avg: average (mean) value - stdev: standard deviation of values - range: range of values (max - min) - sum: sum of all values from raster
Examples	<ul style="list-style-type: none"> • raster_statistic('lc', 1, 'avg') → Average value from band 1 from 'lc' raster layer • raster_statistic('ac2010', 3, 'min') → Minimum value from band 3 from 'ac2010' raster layer

raster_value

Returns the raster value found at the provided point.

语法	raster_value(layer, band, point)
参数	<ul style="list-style-type: none"> • layer - the name or id of a raster layer • band - the band number to sample the value from. • point - point geometry (for multipart geometries having more than one part, a NULL value will be returned)
Examples	<ul style="list-style-type: none"> • raster_value('dem', 1, make_point(1,1)) → 25

14.3.21 Record and Attributes Functions

This group contains functions that operate on record identifiers.

<ul style="list-style-type: none"> • <i>attribute</i> • <i>attributes</i> • <i>\$currentfeature</i> • <i>display_expression</i> • <i>get_feature</i> • <i>get_feature_by_id</i> • <i>\$id</i> • <i>is_selected</i> • <i>maptip</i> • <i>num_selected</i> • <i>represent_value</i>
--

- *sqlite_fetch_and_increment*
- *uuid*

attribute

Returns an attribute from a feature.

Variant 1

Returns the value of an attribute from the current feature.

语法	attribute(attribute_name)
参数	<ul style="list-style-type: none"> • attribute_name - name of attribute to be returned
Examples	<ul style="list-style-type: none"> • <code>attribute('name')</code> → value stored in 'name' attribute for the current feature

Variant 2

Allows the target feature and attribute name to be specified.

语法	attribute(feature, attribute_name)
参数	<ul style="list-style-type: none"> • feature - a feature • attribute_name - name of attribute to be returned
Examples	<ul style="list-style-type: none"> • <code>attribute(@atlas_feature, 'name')</code> → value stored in 'name' attribute for the current atlas feature

attributes

Returns a map containing all attributes from a feature, with field names as map keys.

Variant 1

Returns a map of all attributes from the current feature.

语法	attributes()
Examples	<ul style="list-style-type: none"> • <code>attributes() ['name']</code> → value stored in 'name' attribute for the current feature

Variant 2

Allows the target feature to be specified.

语法	attributes(feature)
参数	<ul style="list-style-type: none"> • feature - a feature
Examples	<ul style="list-style-type: none"> • <code>attributes(@atlas_feature) ['name']</code> → value stored in 'name' attribute for the current atlas feature

Further reading: *Maps Functions*

\$currentfeature

Returns the current feature being evaluated. This can be used with the 'attribute' function to evaluate attribute values from the current feature.

语法	\$currentfeature
Examples	<ul style="list-style-type: none"> • <code>attribute(\$currentfeature, 'name')</code> → value stored in 'name' attribute for the current feature

display_expression

Returns the display expression for a given feature in a layer. The expression is evaluated by default. Can be used with zero, one or more arguments, see below for details.

No parameters

If called with no parameters, the function will evaluate the display expression of the current feature in the current layer.

语法	display_expression()
Examples	<ul style="list-style-type: none"> • <code>display_expression()</code> → The display expression of the current feature in the current layer.

One 'feature' parameter

If called with a 'feature' parameter only, the function will evaluate the specified feature from the current layer.

语法	display_expression(feature)
参数	<ul style="list-style-type: none"> • feature - The feature which should be evaluated.
Examples	<ul style="list-style-type: none"> • <code>display_expression(@atlas_feature)</code> → The display expression of the current atlas feature.

Layer and feature parameters

If the function is called with both a layer and a feature, it will evaluate the specified feature from the specified layer.

语法	display_expression(layer, feature, [evaluate=true]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - The layer (or its ID or name) • feature - The feature which should be evaluated. • evaluate - If the expression must be evaluated. If false, the expression will be returned as a string literal only (which could potentially be later evaluated using the 'eval' function).
Examples	<ul style="list-style-type: none"> • <code>display_expression('streets', get_feature_by_id('streets', 1))</code> → The display expression of the feature with the ID 1 on the layer 'streets'. • <code>display_expression('a_layer_id', \$currentfeature, 'False')</code> → The display expression of the given feature not evaluated.

get_feature

Returns the first feature of a layer matching a given attribute value.

语法	get_feature(layer, attribute, value)
参数	<ul style="list-style-type: none"> • layer - layer name or ID • attribute - attribute name • value - attribute value to match
Examples	<ul style="list-style-type: none"> • <code>get_feature('streets', 'name', 'main st')</code> → first feature found in "streets" layer with "main st" value in the "name" field

get_feature_by_id

Returns the feature with an id on a layer.

语法	get_feature_by_id(layer, feature_id)
参数	<ul style="list-style-type: none"> • layer - layer, layer name or layer id • feature_id - the id of the feature which should be returned
Examples	<ul style="list-style-type: none"> • <code>get_feature_by_id('streets', 1)</code> → the feature with the id 1 on the layer "streets"

Further reading: *\$id*

\$id

Returns the feature id of the current row.

语法	\$id
Examples	<ul style="list-style-type: none"> • <code>\$id</code> → 42

is_selected

Returns True if a feature is selected. Can be used with zero, one or two arguments, see below for details.

No parameters

If called with no parameters, the function will return true if the current feature in the current layer is selected.

语法	is_selected()
Examples	<ul style="list-style-type: none"> • <code>is_selected()</code> → True if the current feature in the current layer is selected.

One 'feature' parameter

If called with a 'feature' parameter only, the function returns true if the specified feature from the current layer is selected.

语法	is_selected(feature)
参数	<ul style="list-style-type: none"> • feature - The feature which should be checked for selection.
Examples	<ul style="list-style-type: none"> • <code>is_selected(@atlas_feature)</code> → True if a selected feature on the current layer is the active atlas feature. • <code>is_selected(get_feature(@layer, 'name', 'Main St.'))</code> → True if the unique named "Main St." feature on the current layer is selected. • <code>is_selected(get_feature_by_id(@layer, 1))</code> → True if the feature with the id 1 on the current layer is selected.

Two parameters

If the function is called with both a layer and a feature, it will return true if the specified feature from the specified layer is selected.

语法	is_selected(layer, feature)
参数	<ul style="list-style-type: none"> • layer - The layer (its ID or name) on which the selection will be checked. • feature - The feature which should be checked for selection.
Examples	<ul style="list-style-type: none"> • <code>is_selected('streets', get_feature('streets', 'name', 'street_name'))</code> → True if the current building's street is selected (assuming the building layer has a field named 'street_name' and the 'streets' layer has a field called 'name' with unique values). • <code>is_selected('streets', get_feature_by_id('streets', 1))</code> → True if the feature with the id 1 on the "streets" layer is selected.

maptip

Returns the maptip for a given feature in a layer. The expression is evaluated by default. Can be used with zero, one or more arguments, see below for details.

No parameters

If called with no parameters, the function will evaluate the maptip of the current feature in the current layer.

语法	maptip()
Examples	<ul style="list-style-type: none"> • <code>maptip()</code> → The maptip of the current feature in the current layer.

One 'feature' parameter

If called with a 'feature' parameter only, the function will evaluate the specified feature from the current layer.

语法	maptip(feature)
参数	<ul style="list-style-type: none"> • feature - The feature which should be evaluated.
Examples	<ul style="list-style-type: none"> • <code>maptip(@atlas_feature)</code> → The maptip of the current atlas feature.

Layer and feature parameters

If the function is called with both a layer and a feature, it will evaluate the specified feature from the specified layer.

语法	maptip(layer, feature, [evaluate=true]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - The layer (or its ID or name) • feature - The feature which should be evaluated. • evaluate - If the expression must be evaluated. If false, the expression will be returned as a string literal only (which could potentially be later evaluated using the 'eval_template' function).
Examples	<ul style="list-style-type: none"> • maptip('streets', get_feature_by_id('streets', 1)) → The maptip of the feature with the ID 1 on the layer 'streets'. • maptip('a_layer_id', \$currentfeature, 'False') → The maptip of the given feature not evaluated.

num_selected

Returns the number of selected features on a given layer. By default works on the layer on which the expression is evaluated.

语法	num_selected([layer=current layer]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • layer - The layer (or its id or name) on which the selection will be checked.
Examples	<ul style="list-style-type: none"> • num_selected() → The number of selected features on the current layer. • num_selected('streets') → The number of selected features on the layer streets

represent_value

Returns the configured representation value for a field value. It depends on the configured widget type. Often, this is useful for 'Value Map' widgets.

语法	represent_value(value, fieldName)
参数	<ul style="list-style-type: none"> • value - The value which should be resolved. Most likely a field. • fieldName - The field name for which the widget configuration should be loaded. (Optional)
Examples	<ul style="list-style-type: none"> • represent_value("field_with_value_map") → Description for value • represent_value('static value', 'field_name') → Description for static value

Further reading: [widget types](#)

sqlite_fetch_and_increment

Manage autoincrementing values in sqlite databases.

SQLite default values can only be applied on insert and not prefetched.

This makes it impossible to acquire an incremented primary key via AUTO_INCREMENT before creating the row in the database. Sidenote: with postgres, this works via the option *evaluate default values*.

When adding new features with relations, it is really nice to be able to already add children for a parent, while the parents form is still open and hence the parent feature uncommitted.

To get around this limitation, this function can be used to manage sequence values in a separate table on sqlite based formats like gpkg.

The sequence table will be filtered for a sequence id (filter_attribute and filter_value) and the current value of the id_field will be incremented by 1 and the incremented value returned.

If additional columns require values to be specified, the default_values map can be used for this purpose.

Note

This function modifies the target sqlite table. It is intended for usage with default value configurations for attributes.

When the database parameter is a layer and the layer is in transaction mode, the value will only be retrieved once during the lifetime of a transaction and cached and incremented. This makes it unsafe to work on the same database from several processes in parallel.

语法	<pre>sqlite_fetch_and_increment(database, table, id_field, filter_attribute, filter_value, [default_values])</pre> <p>[] marks optional arguments</p>
参数	<ul style="list-style-type: none"> • database - Path to the sqlite file or geopackage layer • table - Name of the table that manages the sequences • id_field - Name of the field that contains the current value • filter_attribute - Name the field that contains a unique identifier for this sequence. Must have a UNIQUE index. • filter_value - Name of the sequence to use. • default_values - Map with default values for additional columns on the table. The values need to be fully quoted. Functions are allowed.
Examples	<ul style="list-style-type: none"> • <code>sqlite_fetch_and_increment(@layer, 'sequence_table', 'last_unique_id', 'sequence_id', 'global', map('last_change', 'date('now)'), 'user', '' @user_account_name '')) → 0</code> • <code>sqlite_fetch_and_increment(layer_property(@layer, 'path'), 'sequence_table', 'last_unique_id', 'sequence_id', 'global', map('last_change', 'date('now)'), 'user', '' @user_account_name '')) → 0</code>

Further reading: [数据源属性](#), [Creating one or many to many relations](#)

uuid

Generates a Universally Unique Identifier (UUID) for each row using the Qt `QUuid::createUuid` method. Each UUID is 38 characters long.

语法	uuid()
Examples	<ul style="list-style-type: none"> • <code>uuid()</code> → '{0bd2f60f-f157-4a6d-96af-d4ba4cb366a1}'

14.3.22 Relations

This group contains the list of the *relations* available in the current project, with their description. It provides a quick access to the relation ID for writing an expression (with e.g. the *relation_aggregate* function) or customizing a form.

14.3.23 String Functions

This group contains functions that operate on strings (e.g., that replace, convert to upper case).

- *ascii*
- *char*
- *concat*
- *format*
- *format_date*
- *format_number*
- *left*
- *length*
- *lower*
- *lpad*
- *regexp_match*
- *regexp_replace*
- *regexp_substr*
- *replace*
- *right*
- *rpad*
- *strpos*
- *substr*
- *title*
- *to_string*
- *trim*
- *upper*
- *wordwrap*

ascii

Returns the unicode code associated with the first character of a string.

语法	ascii(string)
参数	<ul style="list-style-type: none"> • string - the string to convert to unicode code
Examples	<ul style="list-style-type: none"> • <code>ascii('Q') → 81</code>

char

Returns the character associated with a unicode code.

语法	char(code)
参数	<ul style="list-style-type: none"> • code - a unicode code number
Examples	<ul style="list-style-type: none"> • <code>char(81) → 'Q'</code>

concat

Concatenates several strings to one. NULL values are converted to empty strings. Other values (like numbers) are converted to strings.

语法	concat(string1, string2, ...)
参数	<ul style="list-style-type: none"> • string - a string value
Examples	<ul style="list-style-type: none"> • <code>concat('sun', 'set') → 'sunset'</code> • <code>concat('a', 'b', 'c', 'd', 'e') → 'abcde'</code> • <code>concat('Anno ', 1984) → 'Anno 1984'</code> • <code>concat('The Wall', NULL) → 'The Wall'</code>

About fields concatenation

You can also concatenate strings or field values using either `||` or `+` operators, with some special characteristics:

- The `+` operator also means sum up expression, so if you have an integer (field or numeric value) operand, this can be error prone and you better use the others:

```
'My feature id is: ' + "gid" => triggers an error as gid returns an integer
```

- When any of the arguments is a NULL value, either `||` or `+` will return a NULL value. To return the other arguments regardless the NULL value, you may want to use the `concat` function:

```
'My feature id is: ' + NULL ==> NULL
'My feature id is: ' || NULL => NULL
concat('My feature id is: ', NULL) => 'My feature id is: '
```

format

Format a string using supplied arguments.

语法	<code>format(string, arg1, arg2, ...)</code>
参数	<ul style="list-style-type: none">• string - A string with place holders for the arguments. Use %1, %2, etc for placeholders. Placeholders can be repeated.• arg - any type. Any number of arguments.
Examples	<ul style="list-style-type: none">• <code>format('This %1 a %2', 'is', 'test') → 'This is a test'</code>

format_date

Formats a date type or string into a custom string format. Uses Qt date/time format strings. See [QDateTime::toString](#).

语法	<code>format_date(datetime, format, [language])</code> [] marks optional arguments																																																
参数	<ul style="list-style-type: none"> • datetime - date, time or datetime value • format - String template used to format the string. <table border="1" data-bbox="459 387 1385 748"> <thead> <tr> <th>Expression</th> <th>Output</th> </tr> </thead> <tbody> <tr> <td>d</td> <td>the day as number without a leading zero (1 to 31)</td> </tr> <tr> <td>dd</td> <td>the day as number with a leading zero (01 to 31)</td> </tr> <tr> <td>ddd</td> <td>the abbreviated localized day name (e.g. 'Mon' to 'Sun')</td> </tr> <tr> <td>dddd</td> <td>the long localized day name (e.g. 'Monday' to 'Sunday')</td> </tr> <tr> <td>M</td> <td>the month as number without a leading zero (1-12)</td> </tr> <tr> <td>MM</td> <td>the month as number with a leading zero (01-12)</td> </tr> <tr> <td>MMM</td> <td>the abbreviated localized month name (e.g. 'Jan' to 'Dec')</td> </tr> <tr> <td>MMMM</td> <td>the long localized month name (e.g. 'January' to 'December')</td> </tr> <tr> <td>yy</td> <td>the year as two digit number (00-99)</td> </tr> <tr> <td>yyyy</td> <td>the year as four digit number</td> </tr> </tbody> </table> <p>These expressions may be used for the time part of the format string:</p> <table border="1" data-bbox="459 826 1385 1254"> <thead> <tr> <th>Expression</th> <th>Output</th> </tr> </thead> <tbody> <tr> <td>h</td> <td>the hour without a leading zero (0 to 23 or 1 to 12 if AM/PM display)</td> </tr> <tr> <td>hh</td> <td>the hour with a leading zero (00 to 23 or 01 to 12 if AM/PM display)</td> </tr> <tr> <td>H</td> <td>the hour without a leading zero (0 to 23, even with AM/PM display)</td> </tr> <tr> <td>HH</td> <td>the hour with a leading zero (00 to 23, even with AM/PM display)</td> </tr> <tr> <td>m</td> <td>the minute without a leading zero (0 to 59)</td> </tr> <tr> <td>mm</td> <td>the minute with a leading zero (00 to 59)</td> </tr> <tr> <td>s</td> <td>the second without a leading zero (0 to 59)</td> </tr> <tr> <td>ss</td> <td>the second with a leading zero (00 to 59)</td> </tr> <tr> <td>z</td> <td>the milliseconds without trailing zeroes (0 to 999)</td> </tr> <tr> <td>zzz</td> <td>the milliseconds with trailing zeroes (000 to 999)</td> </tr> <tr> <td>AP or A</td> <td>interpret as an AM/PM time. <i>AP</i> must be either 'AM' or 'PM'.</td> </tr> <tr> <td>ap or a</td> <td>Interpret as an AM/PM time. <i>ap</i> must be either 'am' or 'pm'.</td> </tr> </tbody> </table> <ul style="list-style-type: none"> • language - language (lowercase, two- or three-letter, ISO 639 language code) used to format the date into a custom string 	Expression	Output	d	the day as number without a leading zero (1 to 31)	dd	the day as number with a leading zero (01 to 31)	ddd	the abbreviated localized day name (e.g. 'Mon' to 'Sun')	dddd	the long localized day name (e.g. 'Monday' to 'Sunday')	M	the month as number without a leading zero (1-12)	MM	the month as number with a leading zero (01-12)	MMM	the abbreviated localized month name (e.g. 'Jan' to 'Dec')	MMMM	the long localized month name (e.g. 'January' to 'December')	yy	the year as two digit number (00-99)	yyyy	the year as four digit number	Expression	Output	h	the hour without a leading zero (0 to 23 or 1 to 12 if AM/PM display)	hh	the hour with a leading zero (00 to 23 or 01 to 12 if AM/PM display)	H	the hour without a leading zero (0 to 23, even with AM/PM display)	HH	the hour with a leading zero (00 to 23, even with AM/PM display)	m	the minute without a leading zero (0 to 59)	mm	the minute with a leading zero (00 to 59)	s	the second without a leading zero (0 to 59)	ss	the second with a leading zero (00 to 59)	z	the milliseconds without trailing zeroes (0 to 999)	zzz	the milliseconds with trailing zeroes (000 to 999)	AP or A	interpret as an AM/PM time. <i>AP</i> must be either 'AM' or 'PM'.	ap or a	Interpret as an AM/PM time. <i>ap</i> must be either 'am' or 'pm'.
Expression	Output																																																
d	the day as number without a leading zero (1 to 31)																																																
dd	the day as number with a leading zero (01 to 31)																																																
ddd	the abbreviated localized day name (e.g. 'Mon' to 'Sun')																																																
dddd	the long localized day name (e.g. 'Monday' to 'Sunday')																																																
M	the month as number without a leading zero (1-12)																																																
MM	the month as number with a leading zero (01-12)																																																
MMM	the abbreviated localized month name (e.g. 'Jan' to 'Dec')																																																
MMMM	the long localized month name (e.g. 'January' to 'December')																																																
yy	the year as two digit number (00-99)																																																
yyyy	the year as four digit number																																																
Expression	Output																																																
h	the hour without a leading zero (0 to 23 or 1 to 12 if AM/PM display)																																																
hh	the hour with a leading zero (00 to 23 or 01 to 12 if AM/PM display)																																																
H	the hour without a leading zero (0 to 23, even with AM/PM display)																																																
HH	the hour with a leading zero (00 to 23, even with AM/PM display)																																																
m	the minute without a leading zero (0 to 59)																																																
mm	the minute with a leading zero (00 to 59)																																																
s	the second without a leading zero (0 to 59)																																																
ss	the second with a leading zero (00 to 59)																																																
z	the milliseconds without trailing zeroes (0 to 999)																																																
zzz	the milliseconds with trailing zeroes (000 to 999)																																																
AP or A	interpret as an AM/PM time. <i>AP</i> must be either 'AM' or 'PM'.																																																
ap or a	Interpret as an AM/PM time. <i>ap</i> must be either 'am' or 'pm'.																																																
Examples	<ul style="list-style-type: none"> • <code>format_date('2012-05-15', 'dd.MM.yyyy')</code> → '15.05.2012' • <code>format_date('2012-05-15', 'd MMMM yyyy', 'fr')</code> → '15 mai 2012' • <code>format_date('2012-05-15', 'dddd')</code> → 'Tuesday' • <code>format_date('2012-05-15 13:54:20', 'dd.MM.yy')</code> → '15.05.12' • <code>format_date('13:54:20', 'hh:mm AP')</code> → '01:54 PM' 																																																

format_number

Returns a number formatted with the locale separator for thousands. Also truncates the decimal places to the number of supplied places.

语法	format_number(number, places, [language]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • number - number to be formatted • places - integer representing the number of decimal places to truncate the string to. • language - language (lowercase, two- or three-letter, ISO 639 language code) used to format the number into a string
Examples	<ul style="list-style-type: none"> • <code>format_number(10000000.332, 2) → '10,000,000.33'</code> • <code>format_number(10000000.332, 2, 'fr') → '10 000 000,33'</code>

left

Returns a substring that contains the *n* leftmost characters of the string.

语法	left(string, length)
参数	<ul style="list-style-type: none"> • string - a string • length - integer. The number of characters from the left of the string to return.
Examples	<ul style="list-style-type: none"> • <code>left('Hello World', 5) → 'Hello'</code>

length

Returns the number of characters in a string or the length of a geometry linestring.

String variant

Returns the number of characters in a string.

语法	length(string)
参数	<ul style="list-style-type: none"> • string - string to count length of
Examples	<ul style="list-style-type: none"> • <code>length('hello') → 5</code>

Geometry variant

Calculate the length of a geometry line object. Calculations are always planimetric in the Spatial Reference System (SRS) of this geometry, and the units of the returned length will match the units for the SRS. This differs from the calculations performed by the \$length function, which will perform ellipsoidal calculations based on the project's ellipsoid and distance unit settings.

语法	length(geometry)
参数	<ul style="list-style-type: none"> • geometry - line geometry object
Examples	<ul style="list-style-type: none"> • <code>length(geom_from_wkt('LINESTRING(0 0, 4 0)')) → 4.0</code>

lower

Converts a string to lower case letters.

语法	lower(string)
参数	<ul style="list-style-type: none"> • string - the string to convert to lower case
Examples	<ul style="list-style-type: none"> • <code>lower('HELLO World') → 'hello world'</code>

lpad

Returns a string padded on the left to the specified width, using a fill character. If the target width is smaller than the string's length, the string is truncated.

语法	lpad(string, width, fill)
参数	<ul style="list-style-type: none"> • string - string to pad • width - length of new string • fill - character to pad the remaining space with
Examples	<ul style="list-style-type: none"> • <code>lpad('Hello', 10, 'x') → 'xxxxxHello'</code> • <code>lpad('Hello', 3, 'x') → 'Hel'</code>

regexp_match

Return the first matching position matching a regular expression within an unicode string, or 0 if the substring is not found.

语法	regexp_match(input_string, regex)
参数	<ul style="list-style-type: none"> • input_string - the string to test against the regular expression • regex - The regular expression to test against. Backslash characters must be double escaped (e.g., <code>"\s"</code> to match a white space character or <code>"\b"</code> to match a word boundary).
Examples	<ul style="list-style-type: none"> • <code>regexp_match('QGIS ROCKS', '\\sROCKS') → 5</code> • <code>regexp_match('Budač', 'udač\\b') → 2</code>

regexp_replace

Returns a string with the supplied regular expression replaced.

语法	regexp_replace(input_string, regex, replacement)
参数	<ul style="list-style-type: none"> • input_string - the string to replace matches in • regex - The regular expression to replace. Backslash characters must be double escaped (e.g., "\s" to match a white space character). • replacement - The string that will replace any matching occurrences of the supplied regular expression. Captured groups can be inserted into the replacement string using \1, \2, etc.
Examples	<ul style="list-style-type: none"> • <code>regexp_replace('QGIS SHOULD ROCK', '\\sSHOULD\\s', ' DOES')</code> → 'QGIS DOES ROCK' • <code>regexp_replace('ABC123', '\\d+', '')</code> → 'ABC' • <code>regexp_replace('my name is John', '(.*) is (.*)', '\\2 is \\1')</code> → 'John is my name'

regexp_substr

Returns the portion of a string which matches a supplied regular expression.

语法	regexp_substr(input_string, regex)
参数	<ul style="list-style-type: none"> • input_string - the string to find matches in • regex - The regular expression to match against. Backslash characters must be double escaped (e.g., "\s" to match a white space character).
Examples	<ul style="list-style-type: none"> • <code>regexp_substr('abc123', '(\\d+)')</code> → '123'

replace

Returns a string with the supplied string, array, or map of strings replaced.

String & array variant

Returns a string with the supplied string or array of strings replaced by a string or an array of strings.

语法	replace(string, before, after)
参数	<ul style="list-style-type: none"> • string - the input string • before - the string or array of strings to replace • after - the string or array of strings to use as a replacement
Examples	<ul style="list-style-type: none"> • <code>replace('QGIS SHOULD ROCK', 'SHOULD', 'DOES')</code> → 'QGIS DOES ROCK' • <code>replace('QGIS ABC', array('A', 'B', 'C'), array('X', 'Y', 'Z'))</code> → 'QGIS XYZ' • <code>replace('QGIS', array('Q', 'S'), '')</code> → 'GI'

Map variant

Returns a string with the supplied map keys replaced by paired values.

语法	<code>replace(string, map)</code>
参数	<ul style="list-style-type: none"> • string - the input string • map - the map containing keys and values
Examples	<ul style="list-style-type: none"> • <code>replace('APP SHOULD ROCK', map('APP', 'QGIS', 'SHOULD', 'DOES'))</code> → 'QGIS DOES ROCK'

right

Returns a substring that contains the *n* rightmost characters of the string.

语法	<code>right(string, length)</code>
参数	<ul style="list-style-type: none"> • string - a string • length - integer. The number of characters from the right of the string to return.
Examples	<ul style="list-style-type: none"> • <code>right('Hello World', 5)</code> → 'World'

rpadd

Returns a string padded on the right to the specified width, using a fill character. If the target width is smaller than the string's length, the string is truncated.

语法	<code>rpadd(string, width, fill)</code>
参数	<ul style="list-style-type: none"> • string - string to pad • width - length of new string • fill - character to pad the remaining space with
Examples	<ul style="list-style-type: none"> • <code>rpadd('Hello', 10, 'x')</code> → 'Helloxxxxx' • <code>rpadd('Hello', 3, 'x')</code> → 'Hel'

strpos

Return the first matching position of a substring within another string, or 0 if the substring is not found.

语法	<code>strpos(haystack, needle)</code>
参数	<ul style="list-style-type: none"> • haystack - string that is to be searched • needle - string to search for
Examples	<ul style="list-style-type: none"> • <code>strpos('HELLO WORLD', 'WORLD')</code> → 7 • <code>strpos('HELLO WORLD', 'GOODBYE')</code> → 0

substr

Returns a part of a string.

语法	substr(string, start, [length]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - the full input string • start - integer representing start position to extract beginning with 1; if start is negative, the return string will begin at the end of the string minus the start value • length - integer representing length of string to extract; if length is negative, the return string will omit the given length of characters from the end of the string
Examples	<ul style="list-style-type: none"> • substr('HELLO WORLD', 3, 5) → 'LLO W' • substr('HELLO WORLD', 6) → 'WORLD' • substr('HELLO WORLD', -5) → 'WORLD' • substr('HELLO', 3, -1) → 'LL' • substr('HELLO WORLD', -5, 2) → 'WO' • substr('HELLO WORLD', -5, -1) → 'WORL'

title

Converts all words of a string to title case (all words lower case with leading capital letter).

语法	title(string)
参数	<ul style="list-style-type: none"> • string - the string to convert to title case
Examples	<ul style="list-style-type: none"> • title('hello wOrld') → 'Hello World'

to_string

Converts a number to string.

语法	to_string(number)
参数	<ul style="list-style-type: none"> • number - Integer or real value. The number to convert to string.
Examples	<ul style="list-style-type: none"> • to_string(123) → '123'

trim

Removes all leading and trailing whitespace (spaces, tabs, etc) from a string.

语法	trim(string)
参数	<ul style="list-style-type: none"> • string - string to trim
Examples	<ul style="list-style-type: none"> • trim(' hello world ') → 'hello world'

upper

Converts a string to upper case letters.

语法	upper(string)
参数	<ul style="list-style-type: none"> • string - the string to convert to upper case
Examples	<ul style="list-style-type: none"> • upper('hello wOrld') → 'HELLO WORLD'

wordwrap

Returns a string wrapped to a maximum/minimum number of characters.



语法	wordwrap(string, wrap_length, [delimiter_string]) [] marks optional arguments
参数	<ul style="list-style-type: none"> • string - the string to be wrapped • wrap_length - an integer. If wrap_length is positive the number represents the ideal maximum number of characters to wrap; if negative, the number represents the minimum number of characters to wrap. • delimiter_string - Optional delimiter string to wrap to a new line.
Examples	<ul style="list-style-type: none"> • wordwrap('UNIVERSITY OF QGIS', 13) → 'UNIVERSITY OF
QGIS' • wordwrap('UNIVERSITY OF QGIS', -3) → 'UNIVERSITY
OF QGIS'

14.3.24 User Expressions

This group contains the expressions saved as *user expressions*.

14.3.25 Variables

This group contains dynamic variables related to the application, the project file and other settings. The availability of variables depends on the context:

- from the  Select by expression dialog
- from the  Field calculator dialog
- from the layer properties dialog
- from the print layout

To use these variables in an expression, they should be preceded by the @ character (e.g, @row_number).

Variable	Description
algorithm_id	The unique ID of an algorithm
animation_end_time	End of the animation's overall temporal time range (as a datetime value)
animation_interval	Duration of the animation's overall temporal time range (as an interval value)
animation_start_time	Start of the animation's overall temporal time range (as a datetime value)
atlas_feature	The current atlas feature (as feature object)
atlas_featureid	The current atlas feature ID

下页继续

表 14.2 – 续上页

Variable	Description
atlas_featurenumber	The current atlas feature number in the layout
atlas_filename	The current atlas file name
atlas_geometry	The current atlas feature geometry
atlas_layerid	The current atlas coverage layer ID
atlas_layername	The current atlas coverage layer name
atlas_pagename	The current atlas page name
atlas_totalfeatures	The total number of features in atlas
canvas_cursor_point	The last cursor position on the canvas in the project's geographical coordinates
cluster_color	The color of symbols within a cluster, or NULL if symbols have mixed colors
cluster_size	The number of symbols contained within a cluster
current_feature	The feature currently being edited in the attribute form or table row
current_geometry	The geometry of the feature currently being edited in the form or the table row
current_parent_feature	represents the feature currently being edited in the parent form. Only usable in an embedded form context.
current_parent_geometry	represents the geometry of the feature currently being edited in the parent form. Only usable in an embedded form context.
form_mode	What the form is used for, like AddFeatureMode, SingleEditMode, MultiEditMode, SearchMode, AggregateSearchMode or IdentifyMode as string.
frame_duration	Temporal duration of each animation frame (as an interval value)
frame_number	Current frame number during animation playback
frame_rate	Number of frames per second during animation playback
fullextent_maxx	Maximum x value from full canvas extent (including all layers)
fullextent_maxy	Maximum y value from full canvas extent (including all layers)
fullextent_minx	Minimum x value from full canvas extent (including all layers)
fullextent_miny	Minimum y value from full canvas extent (including all layers)
geometry_part_count	The number of parts in rendered feature's geometry
geometry_part_num	The current geometry part number for feature being rendered
geometry_point_count	The number of points in the rendered geometry's part
geometry_point_num	The current point number in the rendered geometry's part
grid_axis	The current grid annotation axis (eg, 'x' for longitude, 'y' for latitude)
grid_number	The current grid annotation value
item_id	The layout item user ID (not necessarily unique)
item_uuid	The layout item unique ID
layer	The current layer
layer_id	The ID of current layer
layer_ids	The IDs of all the map layers in the current project as a list
layer_name	The name of current layer
layers	All the map layers in the current project as a list
layout_dpi	The composition resolution (DPI)
layout_name	The layout name
layout_numpages	The number of pages in the layout
layout_page	The page number of the current item in the layout
layout_pageheight	The active page height in the layout (in mm)
layout_pagewidth	The active page width in the layout (in mm)
legend_column_count	The number of columns in the legend
legend_filter_by_map	Indicates if the content of the legend is filtered by the map
legend_filter_out_atlas	Indicates if the atlas is filtered out of the legend
legend_split_layers	Indicates if layers can be split in the legend
legend_title	The title of the legend
legend_wrap_string	The character(s) used to wrap the legend text
map_crs	The Coordinate reference system of the current map
map_crs_acronym	The acronym of the Coordinate reference system of the current map
map_crs_definition	The full definition of the Coordinate reference system of the current map

下页继续

表 14.2 – 续上页

Variable	Description
map_crs_description	The name of the Coordinate reference system of the current map
map_crs_ellipsoid	The acronym of the ellipsoid of the Coordinate reference system of the current map
map_crs_proj4	The Proj4 definition of the Coordinate reference system of the current map
map_crs_wkt	The WKT definition of the Coordinate reference system of the current map
map_end_time	The end of the map's temporal time range (as a datetime value)
map_extent	The geometry representing the current extent of the map
map_extent_center	The point feature at the center of the map
map_extent_height	The current height of the map
map_extent_width	The current width of the map
map_id	The ID of current map destination. This will be 'canvas' for canvas renders, and the item ID for layout map renders
map_interval	The duration of the map's temporal time range (as an interval value)
map_layer_ids	The list of map layer IDs visible in the map
map_layers	The list of map layers visible in the map
map_rotation	The current rotation of the map
map_scale	The current scale of the map
map_start_time	The start of the map's temporal time range (as a datetime value)
map_units	The units of map measurements
model_path	Full path (including file name) of current model (or project path if model is embedded in a project).
model_folder	Folder containing current model (or project folder if model is embedded in a project).
model_name	Name of current model
model_group	Group for current model
notification_message	Content of the notification message sent by the provider (available only for actions triggered by provider notifications).
parent	Refers to the current feature in the parent layer, providing access to its attributes and geometry when filtering an <i>aggregate</i> function
project_abstract	The project abstract, taken from project metadata
project_area_units	The area unit for the current project, used when calculating areas of geometries
project_author	The project author, taken from project metadata
project_basename	The basename of current project's filename (without path and extension)
project_creation_date	The project creation date, taken from project metadata
project_crs	The Coordinate reference system of the project
project_crs_arconym	The acronym of the Coordinate reference system of the project
project_crs_definition	The full definition of the Coordinate reference system of the project
project_crs_description	The description of the Coordinate reference system of the project
project_crs_ellipsoid	The ellipsoid of the Coordinate reference system of the project
project_crs_proj4	The Proj4 representation of the Coordinate reference system of the project
project_crs_wkt	The WKT (well known text) representation of the coordinate reference system of the project
project_distance_units	The distance unit for the current project, used when calculating lengths of geometries and distances
project_ellipsoid	The name of the ellipsoid of the current project, used when calculating geodetic areas or lengths of geometries
project_filename	The filename of the current project
project_folder	The folder of the current project
project_home	The home path of the current project
project_identifier	The project identifier, taken from the project's metadata
project_keywords	The project keywords, taken from the project's metadata
project_last_saved	Date/time when project was last saved.
project_path	The full path (including file name) of the current project
project_title	The title of current project
project_units	The units of the project's CRS

下页继续

表 14.2 - 续上页

Variable	Description
qgis_locale	The current language of QGIS
qgis_os_name	The current Operating system name, eg 'windows', 'linux' or 'osx'
qgis_platform	The QGIS platform, eg 'desktop' or 'server'
qgis_release_name	The current QGIS release name
qgis_short_version	The current QGIS version short string
qgis_version	The current QGIS version string
qgis_version_no	The current QGIS version number
row_number	Stores the number of the current row
snapping_results	Gives access to snapping results while digitizing a feature (only available in add feature)
scale_value	The current scale bar distance value
symbol_angle	The angle of the symbol used to render the feature (valid for marker symbols only)
symbol_color	The color of the symbol used to render the feature
symbol_count	The number of features represented by the symbol (in the layout legend)
symbol_id	The Internal ID of the symbol (in the layout legend)
symbol_label	The label for the symbol (either a user defined label or the default autogenerated label - in the layout legend)
symbol_layer_count	Total number of symbol layers in the symbol
symbol_layer_index	Current symbol layer index
symbol_marker_column	Column number for marker (valid for point pattern fills only).
symbol_marker_row	Row number for marker (valid for point pattern fills only).
user_account_name	The current user's operating system account name
user_full_name	The current user's operating system user name
value	The current value
with_variable	Allows setting a variable for usage within an expression and avoid recalculating the same value repeatedly
zoom_level	Zoom level of the tile that is being rendered (derived from the current map scale). Normally in interval [0, 20].

Some examples:

- Return the X coordinate of a map item center in layout:

```
x( map_get( item_variables( 'map1' ), 'map_extent_center' ) )
```

- Return, for each feature in the current layer, the number of overlapping airport features:

```
aggregate( layer='airport', aggregate='count', expression="code",
           filter:=intersects( $geometry, geometry( @parent ) ) )
```

- Get the object_id of the first snapped point of a line:

```
with_variable(
  'first_snapped_point',
  array_first( @snapping_results ),
  attribute(
    get_feature_by_id(
      map_get( @first_snapped_point, 'layer' ),
      map_get( @first_snapped_point, 'feature_id' )
    ),
    'object_id'
  )
)
```


14.3.26 Recent Functions

This group contains recently used functions. Depending on the context of its usage (feature selection, field calculator, generic), recently applied expressions are added to the corresponding list (up to ten expressions), sorted from more to less recent. This makes it easy to quickly retrieve and reapply previously used expressions.

14.4 属性表操作

属性表展示的是所选图层中各个要素的属性信息。表中的每一行代表一个（几何或非几何）要素，每一列代表要素的某种属性。表中要素可以被选中，移动和编辑。

14.4.1 前言：空间与非空间属性表

QGIS 可以加载空间与非空间图层。当前支持的有符合 OGR 标准的表格，分隔文本，以及 PostgreSQL, MSSQL, SpatiaLite, DB2 and Oracle 数据库的表格。所有的加载图层都会显示在 `:guilabel:Layers` panel 中。图层在地图中的可交互性取决于它是否包含空间的信息。

非空间表格可以使用属性表查看器进行浏览与编辑。不仅如此，它们还可以用于属性查找。例如，你可以在创建矢量图层的时候定义非空间表格的某一列为属性值本身，或者用来限定其取值范围。详见编辑部件的文档：[ref:vector_attributes_menu](#)

14.4.2 属性表界面介绍

点击对应图层的 `:ref:label_legend` 按钮激活图层，选择 `:menuselection:'Layer'` 菜单下的 `lopenTable|:menuselection:'Open Attribute Table'` 打开图层属性表。你也可以右键单击图层，在下拉列表中选择 `lopenTable|:menuselection:'Open Attribute Table'`，或者在属性工具栏中点击 `lopenTable Open Attribute Table` 按钮。如果你想使用快捷键，按 `:kbd:'F6'` 打开属性表，`:kbd:'Shift+F6'` 打开所选要素的属性表，`:kbd:'Ctrl+F6'` 打开可见要素的属性表。

This will open a new window that displays the feature attributes for the layer ([figure_attributes_table](#)). According to the setting in `Settings ► Options ► Data sources` menu, the attribute table will open in a docked window or a regular window. The total number of features in the layer and the number of currently selected/filtered features are shown in the attribute table title, as well as if the layer is spatially limited.

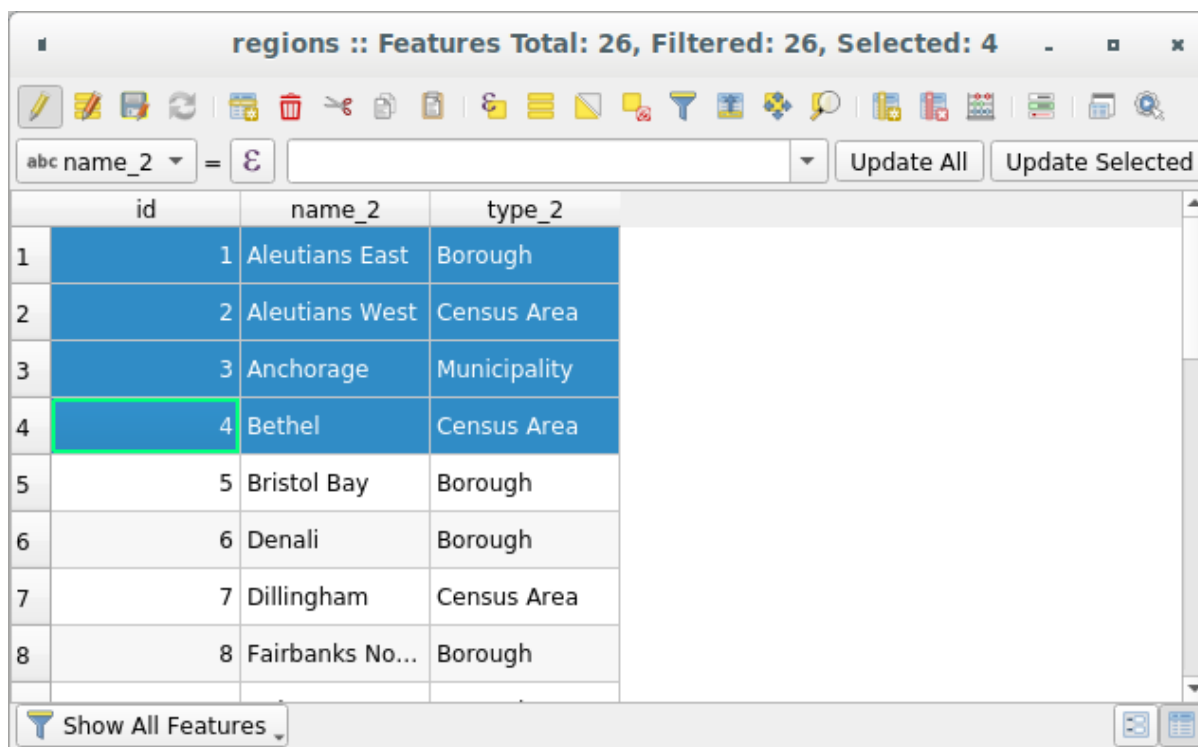


图 14.68: Attribute Table for regions layer








The buttons at the top of the attribute table window provide the following functionality:

表 14.3: Available Tools

Icon	Label	目标	Default Shortcut
	Toggle editing mode	Enable editing functionalities	Ctrl+E
	Toggle multi edit mode	Update multiple fields of many features	
	Save Edits	Save current modifications	
	Reload the table		
	Add feature	Add new geometryless feature	
	Delete selected features	Remove selected features from the layer	
	Cut selected features to clipboard		Ctrl+X
	Copy selected features to clipboard		Ctrl+C
	Paste features from clipboard	Insert new features from copied ones	Ctrl+V
	Select features using an Expression		
	Select All	Select all features in the layer	Ctrl+A
	Invert selection	Invert the current selection in the layer	Ctrl+R
	Deselect all	Deselect all features in the current layer	Ctrl+Shift+A
	Filter/Select features using form		Ctrl+F
	Move selected to top	Move selected rows to the top of the table	
	Pan map to the selected rows		Ctrl+P

下页继续

表 14.3 – 续上页

Icon	Label	目标	Default Shortcut
	Zoom map to the selected rows		Ctrl+J
	New field	Add a new field to the data source	Ctrl+W
	Delete field	Remove a field from the data source	
	Open field calculator	Update field for many features in a row	Ctrl+I
	Conditional formatting	Enable table formatting	
	Dock attribute table	Allows to dock/undock the attribute table	
	Actions	Lists the actions related to the layer	

注解: Depending on the format of the data and the OGR library built with your QGIS version, some tools may not be available.





Below these buttons is the Quick Field Calculation bar (enabled only in *edit mode*), which allows to quickly apply calculations to all or part of the features in the layer. This bar uses the same *expressions* as the  Field Calculator (see *Editing attribute values*).

Table view vs Form view

QGIS provides two view modes to easily manipulate data in the attribute table:

- The  Table view, displays values of multiple features in a tabular mode, each row representing a feature and each column a field.
- The  Form view shows *feature identifiers* in a first panel and displays only the attributes of the clicked identifier in the second one. There is a pull-down menu at the top of the first panel where the "identifier" can be specified using an attribute (*Column preview*) or an *Expression*. The pull-down also includes the last 10 expressions for re-use. Form view uses the layer fields configuration (see *Attributes Form Properties*). You can browse through the feature identifiers with the arrows on the bottom of the first panel. Once you marked the feature in yellow in the list it is selected in yellow on the canvas. Use the  on top of the attribute table to zoom to the feature. Clicking on an entry in the list (without using the rectangles) makes a feature flash in red color once so you can see where it is situated.

You can switch from one mode to the other by clicking the corresponding icon at the bottom right of the dialog.

You can also specify the *Default view* mode at the opening of the attribute table in *Settings ► Options ► Data Sources* menu. It can be 'Remember last view', 'Table view' or 'Form view'.

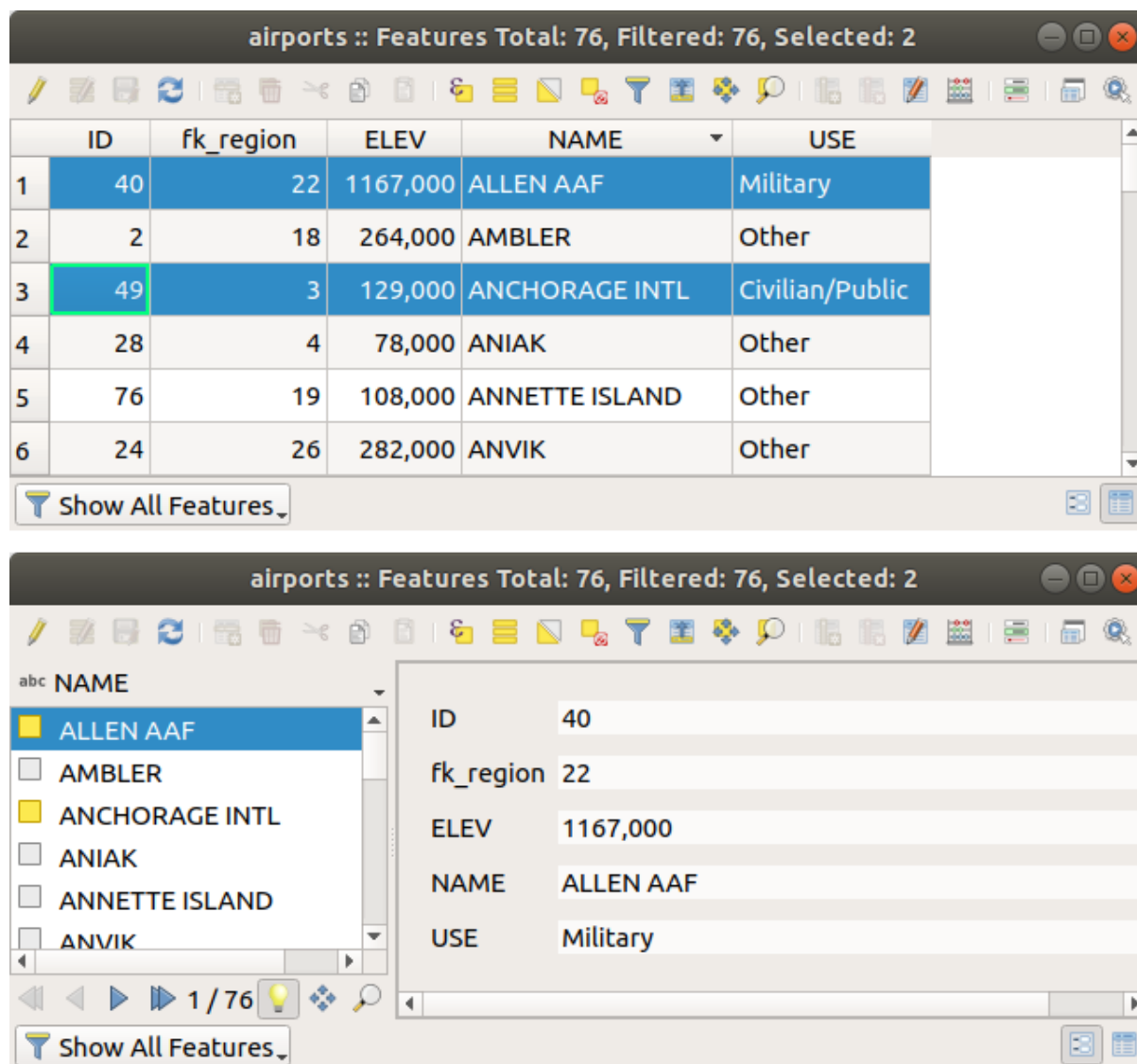


图 14.69: Attribute table in table view (top) vs form view (bottom)

Configuring the columns

Right-click in a column header when in table view to have access to tools that help you configure what can be displayed in the attribute table and how.

Hiding and organizing columns and enabling actions

By right-clicking in a column header, you can choose to hide it from the attribute table. To change several columns behavior at once, unhide a column or change the order of the columns, choose *Organize columns* In the new dialog, you can:

- check/uncheck columns you want to show or hide
- drag-and-drop items to reorder the columns in the attribute table. Note that this change is for the table rendering and does not alter the fields order in the layer datasource
- enable a new virtual *Actions* column that displays in each row a drop-down box or button list of actions for each row, see *Actions Properties* for more information about actions.

Resizing columns widths


Columns width can be set through a right-click on the column header and select either:

- *Set width...* to enter the desired value. By default, the current value is displayed in the widget
- *Autosize* to resize at the best fit the column.

It can also be changed by dragging the boundary on the right of the column heading. The new size of the column is maintained for the layer, and restored at the next opening of the attribute table.

Sorting columns

The table can be sorted by any column, by clicking on the column header. A small arrow indicates the sort order (downward pointing means descending values from the top row down, upward pointing means ascending values from the top row down). You can also choose to sort the rows with the *sort* option of the column header context menu and write an expression, e.g. to sort the row using multiple columns you can write `concat (col0, col1)`.

In form view, features identifier can be sorted using the  *Sort by preview expression* option.

小技巧: Sorting based on columns of different types



Trying to sort an attribute table based on columns of string and numeric types may lead to unexpected result because of the `concat ("USE", "ID")` expression returning string values (ie, 'Borough105' < 'Borough6'). You can workaround this by using eg `concat ("USE", lpad("ID", 3, 0))` which returns 'Borough105' > 'Borough006'.

Formatting of table cells using conditions

Conditional formatting settings can be used to highlight in the attribute table features you may want to put a particular focus on, using custom conditions on feature's:

- geometry (e.g., identifying multi-parts features, small area ones or in a defined map extent...);
- or field value (e.g., comparing values to a threshold, identifying empty cells...).

You can enable the conditional formatting panel clicking on  at the top right of the attributes window in table view (not available in form view).

The new panel allows user to add new rules to format rendering of  *Field* or  *Full row*. Adding new rule opens a form to define:

- the name of the rule;
- a condition using any of the *expression builder* functions;
- the formatting: it can be chosen from a list of predefined formats or created based on properties like:
 - background and text colors;
 - use of icon;
 - bold, italic, underline, or strikeout;
 - font.

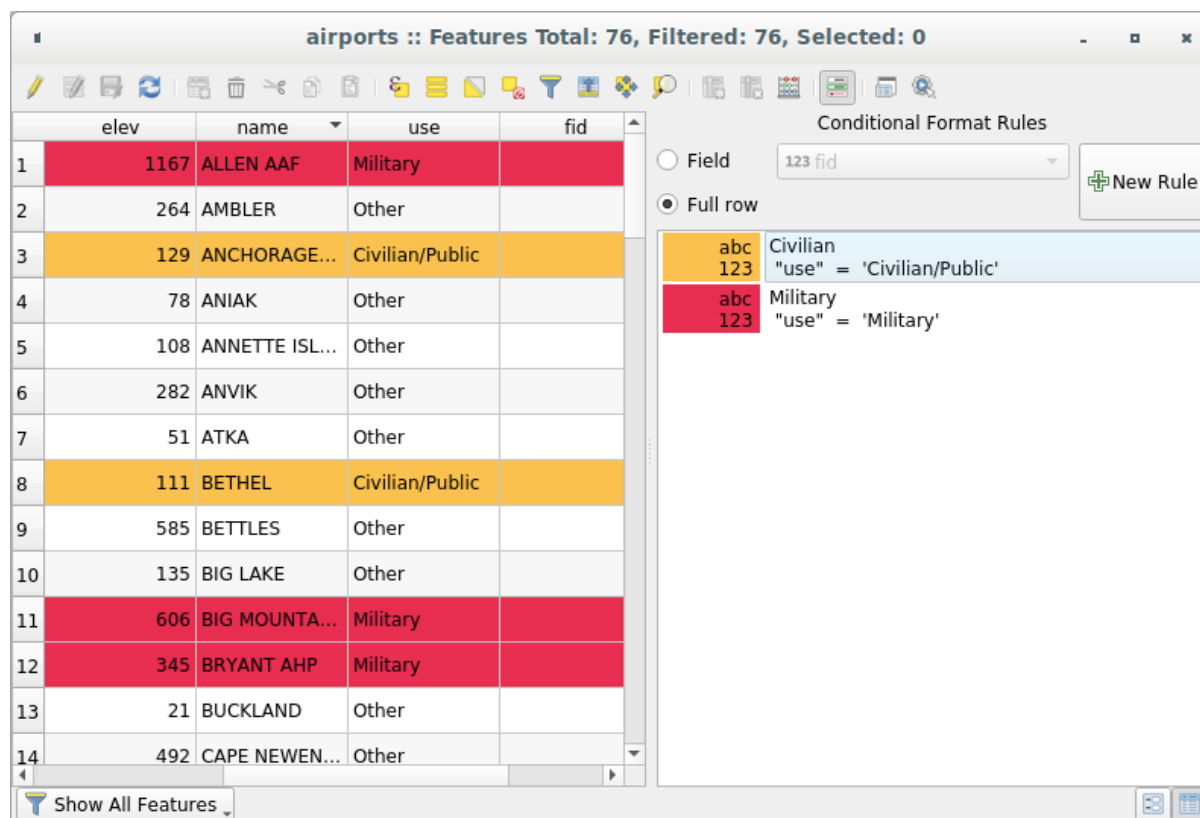


图 14.70: Conditional Formatting of an attribute table

14.4.3 Interacting with features in an attribute table

Selecting features

In table view, each row in the attribute table displays the attributes of a unique feature in the layer. Selecting a row selects the feature and likewise, selecting a feature in the map canvas (in case of geometry enabled layer) selects the row in the attribute table. If the set of features selected in the map canvas (or attribute table) is changed, then the selection is also updated in the attribute table (or map canvas) accordingly.





Rows can be selected by clicking on the row number on the left side of the row. **Multiple rows** can be marked by holding the `Ctrl` key. A **continuous selection** can be made by holding the `Shift` key and clicking on several row headers on the left side of the rows. All rows between the current cursor position and the clicked row are selected. Moving the cursor position in the attribute table, by clicking a cell in the table, does not change the row selection. Changing the selection in the main canvas does not move the cursor position in the attribute table.

In form view of the attribute table, features are by default identified in the left panel by the value of their displayed field (see *Display Properties*). This identifier can be replaced using the drop-down list at the top of the panel, either by selecting an existing field or using a custom expression. You can also choose to sort the list of features from the drop-down menu.

Click a value in the left panel to display the feature's attributes in the right one. To select a feature, you need to click inside the square symbol at the left of the identifier. By default, the symbol turns into yellow. Like in the table view, you can perform multiple feature selection using the keyboard combinations previously exposed.

Beyond selecting features with the mouse, you can perform automatic selection based on feature's attribute using tools available in the attribute table toolbar, such as (see section *Automatic selection* and following one for more information and use case):

-  *Select By Expression...*

-  *Select Features By Value...*
-  *Deselect All Features from the Layer*
-  *Select All Features*
-  *Invert Feature Selection.*

It is also possible to select features using the *Filtering and selecting features using forms*.

Filtering features

Once you have selected features in the attribute table, you may want to display only these records in the table. This can be easily done using the *Show Selected Features* item from the drop-down list at the bottom left of the attribute table dialog. This list offers the following filters:

- *Show All Features*
- *Show Selected Features*
- *Show Features visible on map*
- *Show Edited and New Features*
- *Field Filter* - allows the user to filter based on value of a field: choose a column from a list, type a value and press `Enter` to filter. Then, only the matching features are shown in the attribute table.
- *Advanced filter (Expression)* - Opens the expression builder dialog. Within it, you can create *complex expressions* to match table rows. For example, you can filter the table using more than one field. When applied, the filter expression will show up at the bottom of the form.


It is also possible to *filter features using forms*.

注解: Filtering records out of the attribute table does not filter features out of the layer; they are simply momentarily hidden from the table and can be accessed from the map canvas or by removing the filter. For filters that do hide features from the layer, use the *Query Builder*.

小技巧: Update datasource filtering with Show Features Visible on Map

When for performance reasons, features shown in attribute table are spatially limited to the canvas extent at its opening (see *Data Source Options* for a how-to), selecting *Show Features Visible on Map* on a new canvas extent updates the spatial restriction.

Filtering and selecting features using forms

Clicking the  Filter/Select features using form or pressing `Ctrl+F` will make the attribute table dialog switch to form view and replace each widget with its search variant.

From this point onwards, this tool functionality is similar to the one described in *Select Features By Value*, where you can find descriptions of all operators and selecting modes.

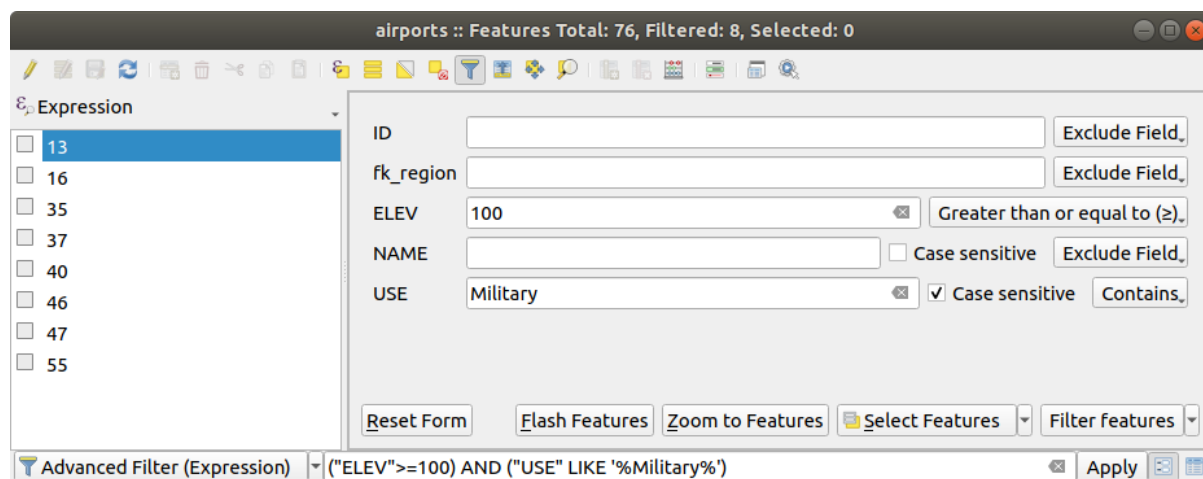


图 14.71: Attribute table filtered by the filter form

When selecting / filtering features from the attribute table, there is a *Filter features* button that allows defining and refining filters. Its use triggers the *Advanced filter (Expression)* option and displays the corresponding filter expression in an editable text widget at the bottom of the form.

If there are already filtered features, you can refine the filter using the drop-down list next to the *Filter features* button. The options are:

- *Filter within ("AND")*
- *Extend filter ("OR")*

To clear the filter, either select the *Show all features* option from the bottom left pull-down menu, or clear the expression and click *Apply* or press *Enter*.

14.4.4 Using action on features

Users have several possibilities to manipulate feature with the contextual menu like:

- *Select all (Ctrl+A)* the features;
- Copy the content of a cell in the clipboard with *Copy cell content*;
- *Zoom to feature* without having to select it beforehand;
- *Pan to feature* without having to select it beforehand;
- *Flash feature*, to highlight it in the map canvas;
- *Open form*: it toggles attribute table into form view with a focus on the clicked feature.

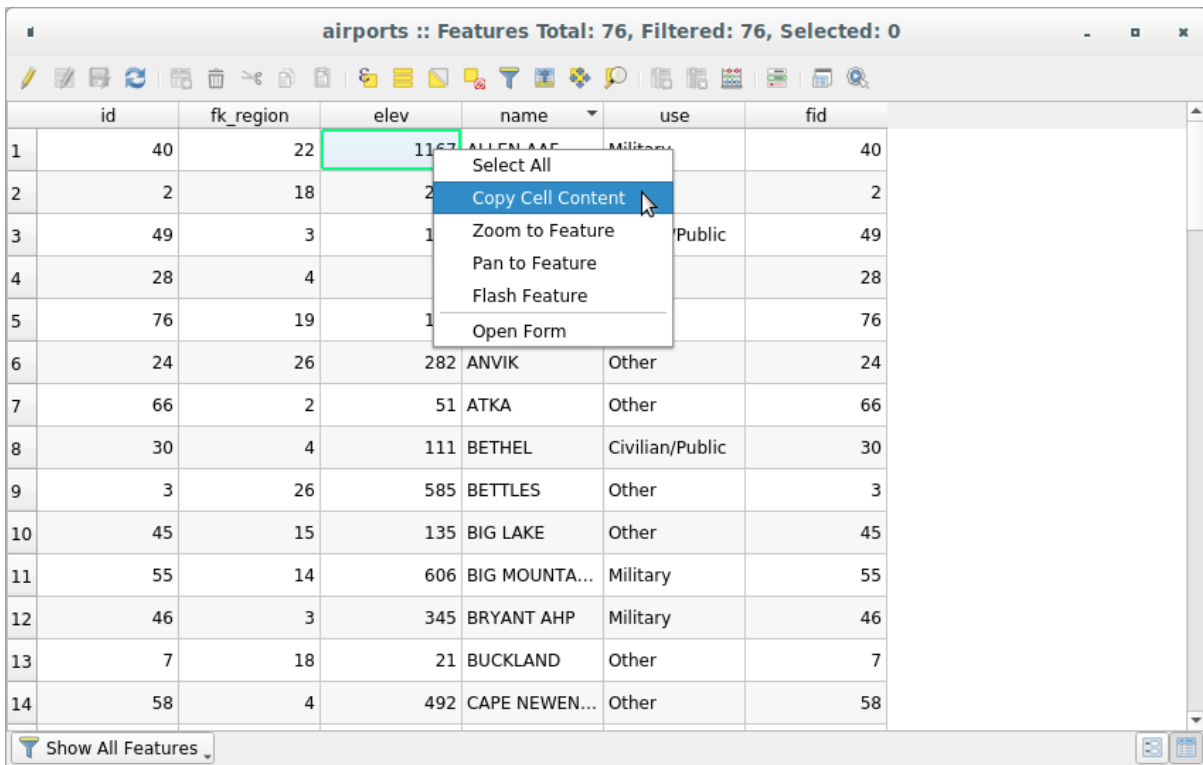



图 14.72: Copy cell content button

If you want to use attribute data in external programs (such as Excel, LibreOffice, QGIS or a custom web application), select one or more row(s) and use the  Copy selected rows to clipboard button or press `Ctrl+C`.

In *Settings* ► *Options* ► *Data Sources* menu you can define the format to paste to with *Copy features as* dropdown list:

- Plain text, no geometry,
- Plain text, WKT geometry,
- GeoJSON

You can also display a list of actions in this contextual menu. This is enabled in the *Layer properties* ► *Actions* tab. See *Actions Properties* for more information on actions.

Saving selected features as new layer


The selected features can be saved as any OGR-supported vector format and also transformed into another coordinate reference system (CRS). In the contextual menu of the layer, from the *Layers* panel, click on *Export* ► *Save selected features as...* to define the name of the output dataset, its format and CRS (see section *Creating new layers from an existing layer*). You'll notice that *Save only selected features* is checked. It is also possible to specify OGR creation options within the dialog.

14.4.5 Editing attribute values

Editing attribute values can be done by:

- typing the new value directly in the cell, whether the attribute table is in table or form view. Changes are hence done cell by cell, feature by feature;
- using the *field calculator*: update in a row a field that may already exist or to be created but for multiple features. It can be used to create virtual fields;
- using the quick field *calculation bar*: same as above but for only existing field;
- or using the *multi edit* mode: update in a row multiple fields for multiple features.

Using the Field Calculator

The  Field Calculator button in the attribute table allows you to perform calculations on the basis of existing attribute values or defined functions, for instance, to calculate length or area of geometry features. The results can be used to update an existing field, or written to a new field (that can be a *virtual* one).

The field calculator is available on any layer that supports edit. When you click on the field calculator icon the dialog opens (see [图 14.73](#)). If the layer is not in edit mode, a warning is displayed and using the field calculator will cause the layer to be put in edit mode before the calculation is made.

Based on the *Expression Builder* dialog, the field calculator dialog offers a complete interface to define an expression and apply it to an existing or a newly created field. To use the field calculator dialog, you must select whether you want to:

1. apply calculation on the whole layer or on selected features only
2. create a new field for the calculation or update an existing one.

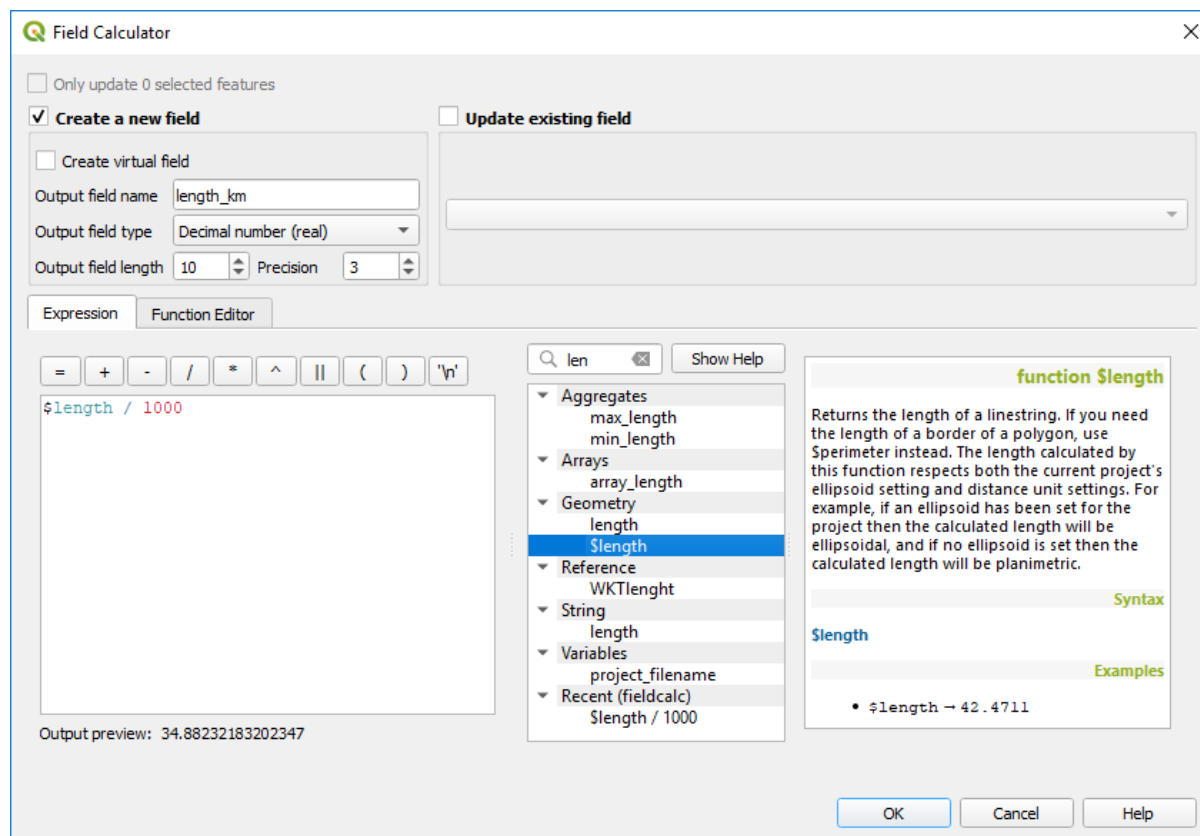






图 14.73: Field Calculator

If you choose to add a new field, you need to enter a field name, a field type (integer, real, date or string) and if needed, the total field length and the field precision. For example, if you choose a field length of 10 and a field precision of 3, it means you have 7 digits before the dot, and 3 digits for the decimal part.

A short example illustrates how field calculator works when using the *Expression* tab. We want to calculate the length in km of the `railroads` layer from the QGIS sample dataset:

1. Load the shapefile `railroads.shp` in QGIS and press  Open Attribute Table.
2. Click on  Toggle editing mode and open the  Field Calculator dialog.
3. Select the  *Create a new field* checkbox to save the calculations into a new field.
4. Set *Output field name* to `length_km`
5. Select `Decimal number (real)` as *Output field type*
6. Set the *Output field length* to 10 and the *Precision* to 3
7. Double click on `$length` in the *Geometry* group to add the length of the geometry into the Field calculator expression box.
8. Complete the expression by typing `/ 1000` in the Field calculator expression box and click *OK*.
9. You can now find a new `length_km` field in the attribute table.

Creating a Virtual Field


A virtual field is a field based on an expression calculated on the fly, meaning that its value is automatically updated as soon as an underlying parameter changes. The expression is set once; you no longer need to recalculate the field each time underlying values change. For example, you may want to use a virtual field if you need area to be evaluated as you digitize features or to automatically calculate a duration between dates that may change (e.g., using `now()` function).

注解: Use of Virtual Fields

- Virtual fields are not permanent in the layer attributes, meaning that they're only saved and available in the project file they've been created.
 - A field can be set virtual only at its creation. Virtual fields are marked with a purple background in the fields tab of the layer properties dialog to distinguish them from regular physical or joined fields. Their expression can be edited later by pressing the expression button in the Comment column. An expression editor window will be opened to adjust the expression of the virtual field.
-

Using the Quick Field Calculation Bar

While Field calculator is always available, the quick field calculation bar on top of the attribute table is only visible if the layer is in edit mode. Thanks to the expression engine, it offers a quicker access to edit an already existing field:

1. Select the field to update in the drop-down list.
2. Fill the textbox with a value, an expression you directly write or build using the  expression button.
3. Click on *Update All*, *Update Selected* or *Update Filtered* button according to your need.

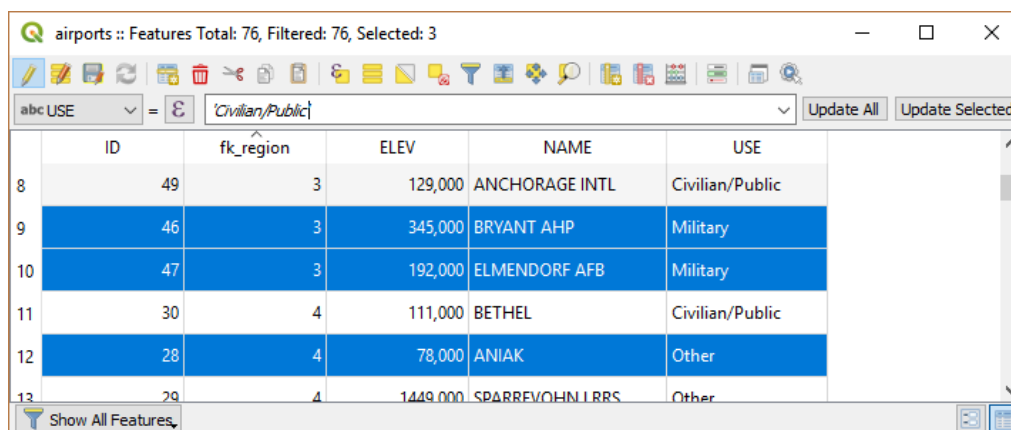




图 14.74: Quick Field Calculation Bar





Editing multiple fields

Unlike the previous tools, multi edit mode allows multiple attributes of different features to be edited simultaneously. When the layer is toggled to edit, multi edit capabilities are accessible:

- using the  Toggle multi edit mode button from the toolbar inside the attribute table dialog;
- or selecting *Edit* ►  *Modify attributes of selected features* menu.

注解: Unlike the tool from the attribute table, hitting the *Edit* ► *Modify Attributes of Selected Features* option provides you with a modal dialog to fill attributes changes. Hence, features selection is required before execution.

In order to edit multiple fields in a row:

1. Select the features you want to edit.
2. From the attribute table toolbar, click the  button. This will toggle the dialog to its form view. Feature selection could also be made at this step.
3. At the right side of the attribute table, fields (and values) of selected features are shown. New widgets appear next to each field allowing for display of the current multi edit state:
 -  The field contains different values for selected features. It's shown empty and each feature will keep its original value. You can reset the value of the field from the drop-down list of the widget.
 -  All selected features have the same value for this field and the value displayed in the form will be kept.
 -  The field has been edited and the entered value will be applied to all the selected features. A message appears at the top of the dialog, inviting you to either apply or reset your modification.

Clicking any of these widgets allows you to either set the current value for the field or reset to original value, meaning that you can roll back changes on a field-by-field basis.

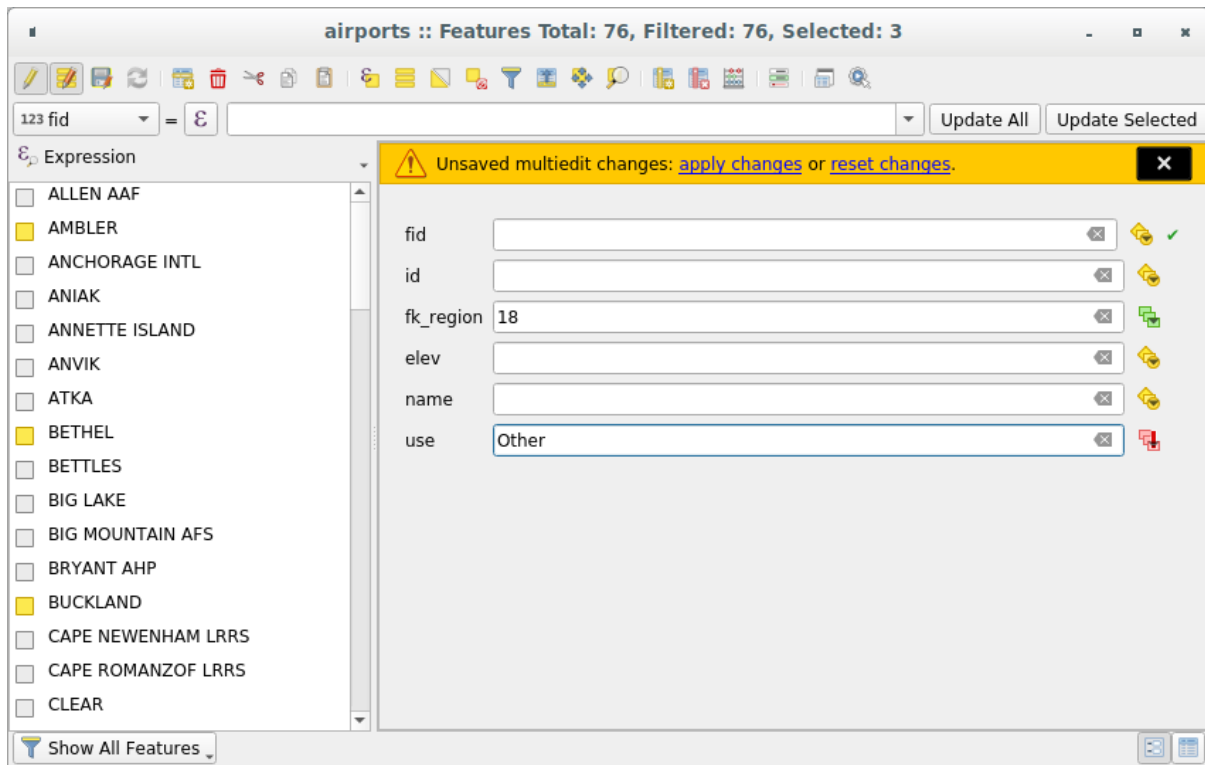



图 14.75: Editing fields of multiple features

4. Make the changes to the fields you want.
5. Click on **Apply changes** in the upper message text or any other feature in the left panel.

Changes will apply to **all selected features**. If no feature is selected, the whole table is updated with your changes.

Modifications are made as a single edit command. So pressing  Undo will rollback the attribute changes for all selected features at once.

注解: Multi edit mode is only available for auto generated and drag and drop forms (see *Customizing a form for your data*); it is not supported by custom ui forms.

14.4.6 Creating one or many to many relations

Relations are a technique often used in databases. The concept is that features (rows) of different layers (tables) can belong to each other.

Introducing 1-N relations

As an example you have a layer with all regions of alaska (polygon) which provides some attributes about its name and region type and a unique id (which acts as primary key).

Then you get another point layer or table with information about airports that are located in the regions and you also want to keep track of these. If you want to add them to the regions layer, you need to create a one to many relation using foreign keys, because there are several airports in most regions.



图 14.76: Alaska region with airports

Layers in 1-N relations

QGIS makes no difference between a table and a vector layer. Basically, a vector layer is a table with a geometry. So you can add your table as a vector layer. To demonstrate the 1-n relation, you can load the `regions` shapefile and the `airports` shapefile which has a foreign key field (`fk_region`) to the layer `regions`. This means, that each airport belongs to exactly one region while each region can have any number of airports (a typical one to many relation).


Foreign keys in 1-N relations

In addition to the already existing attributes in the `airports` attribute table, you'll need another field `fk_region` which acts as a foreign key (if you have a database, you will probably want to define a constraint on it).


This field `fk_region` will always contain an id of a region. It can be seen like a pointer to the region it belongs to. And you can design a custom edit form for editing and QGIS takes care of the setup. It works with different providers (so you can also use it with shape and csv files) and all you have to do is to tell QGIS the relations between your tables.

Defining 1-N relations

The first thing we are going to do is to let QGIS know about the relations between the layers. This is done in *Project*

► *Properties...* Open the *Relations* tab and click on  *Add Relation*.

- **Name** is going to be used as a title. It should be a human readable string, describing, what the relation is used for. We will just call say **airport_relation** in this case.
- **Referenced Layer (Parent)** also considered as parent layer, is the one with the primary key, pointed to, so here it is the `regions` layer. You need to define the primary key of the referenced layer, so it is `ID`.
- **Referencing Layer (Child)** also considered as child layer, is the one with the foreign key field on it. In our case, this is the `airports` layer. For this layer you need to add a referencing field which points to the other layer, so this is `fk_region`.

注解: Sometimes, you need more than a single field to uniquely identify features in a layer. Creating a relation with such a layer requires a **composite key**, ie more than a single pair of matching fields. Use the  button to add as many pairs as necessary.

- **Id** will be used for internal purposes and has to be unique. You may need it to build *custom forms*. If you leave it empty, one will be generated for you but you can assign one yourself to get one that is easier to handle
- **Relationship strength** sets the strength of the relation between the parent and the child layer. The default *Association* type means that the parent layer is *simply* linked to the child one while the *Composition* type allows you to duplicate also the child features when duplicating the parent ones.

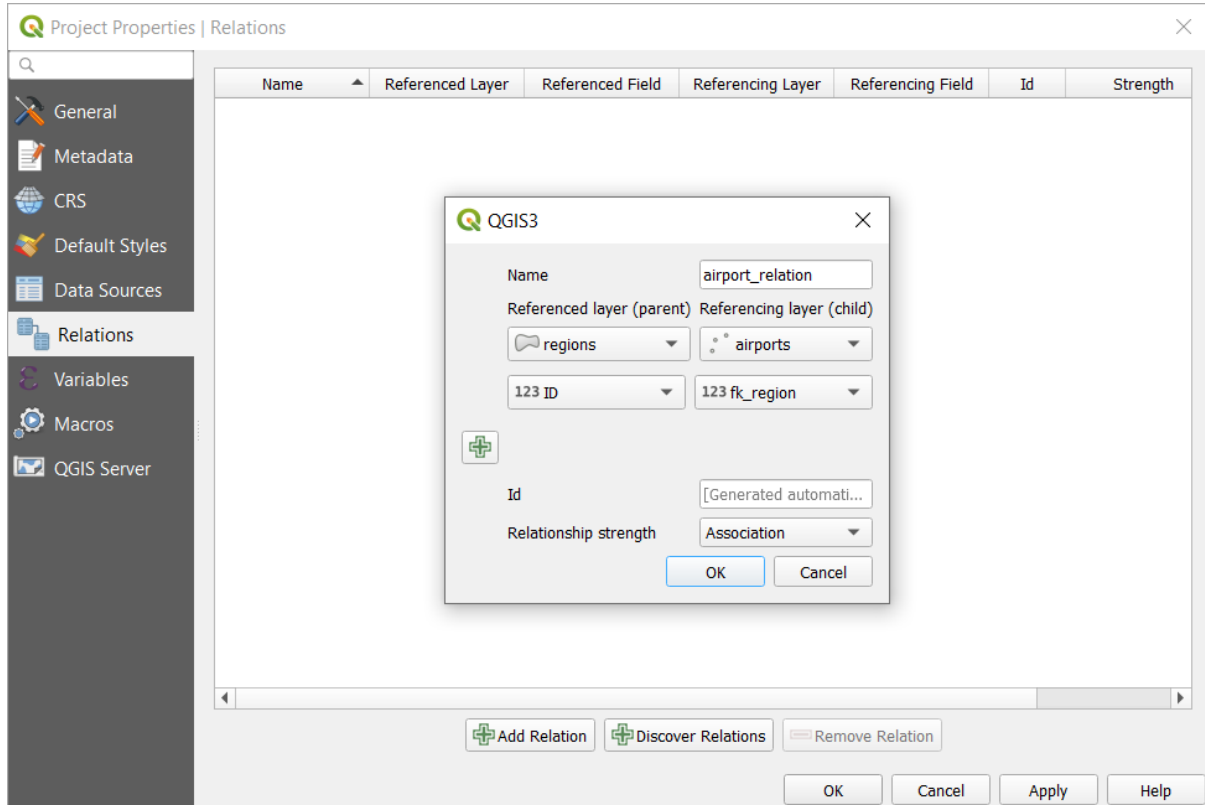



图 14.77: Adding a relation between regions and airports layers

From the *Relations* tab, you can also press the  *Discover Relation* button to fetch the relations available from the providers of the loaded layers. This is possible for layers stored in data providers like PostgreSQL or SpatiaLite.

Forms for 1-N relations

Now that QGIS knows about the relation, it will be used to improve the forms it generates. As we did not change the default form method (autogenerated) it will just add a new widget in our form. So let's select the layer region in the legend and use the identify tool. Depending on your settings, the form might open directly or you will have to choose to open it in the identification dialog under actions.

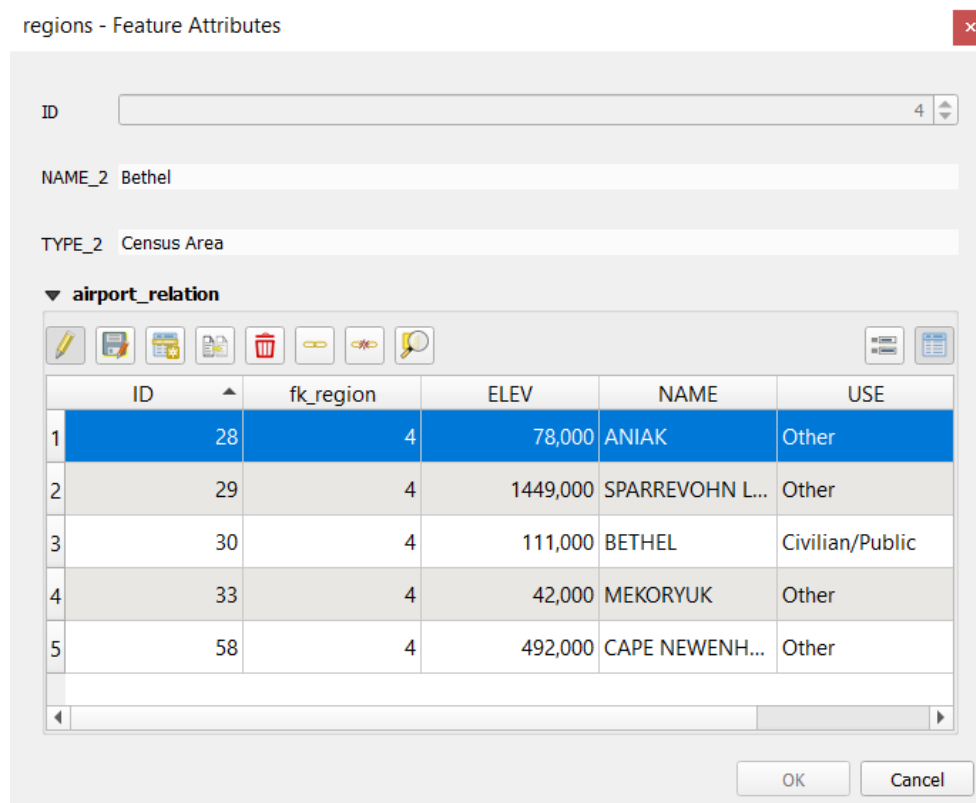
















图 14.78: Identification dialog regions with relation to airports


As you can see, the airports assigned to this particular region are all shown in a table. And there are also some buttons available. Let's review them shortly:

- The  button is for toggling the edit mode. Be aware that it toggles the edit mode of the airport layer, although we are in the feature form of a feature from the region layer. But the table is representing features of the airport layer.
- The  button is for saving all the edits.
- The  button will add a new record to the airport layer attribute table. And it will assign the new airport to the current region by default.
- The  is the same as  but lets you digitize the airport geometry in the map canvas beforehand. Note that the icon will change according to geometry type.
- The  button allows you to copy one or more child features.
- The  button will delete the selected airport permanently.
- The  symbol will open a new dialog where you can select any existing airport which will then be assigned to the current region. This may be handy if you created the airport on the wrong region by accident.
- The  symbol will unlink the selected airport from the current region, leaving them unassigned (the foreign key is set to NULL) effectively.
- With the  button you can zoom the map to the selected child features.
- The two buttons  and  to the right switch between table view and form view where the later let's you view all the airports in their respective form.

In the above example the referencing layer has geometries (so it isn't just an alphanumeric table) so the above steps will create an entry in the layer attribute table that has no corresponding geometric feature. To add the geometry:

1. Choose  *Open Attribute Table* for the referencing layer.
2. Select the record that has been added previously within the feature form of the referenced layer.
3. Use the  *Add Part* digitizing tool to attach a geometry to the selected attributes table record.

If you work on the airport table, the widget Relation Reference is automatically set up for the `fk_region` field (the one used to create the relation), see *Relation Reference widget*.

In the airport form you will see the  button at the right side of the `fk_region` field: if you click on the button the form of the region layer will be opened. This widget allows you to easily and quickly open the forms of the linked parent features.

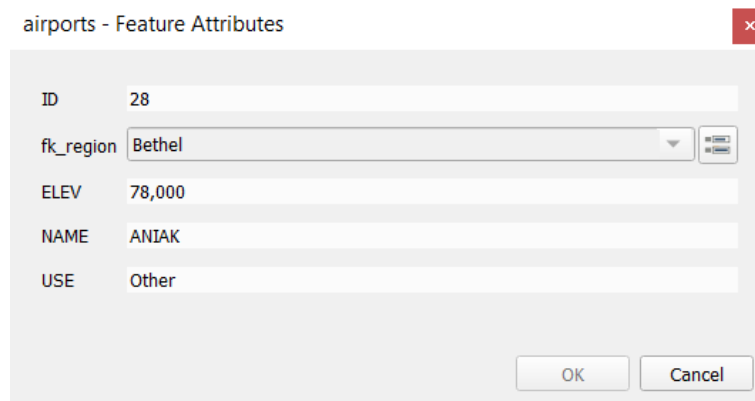
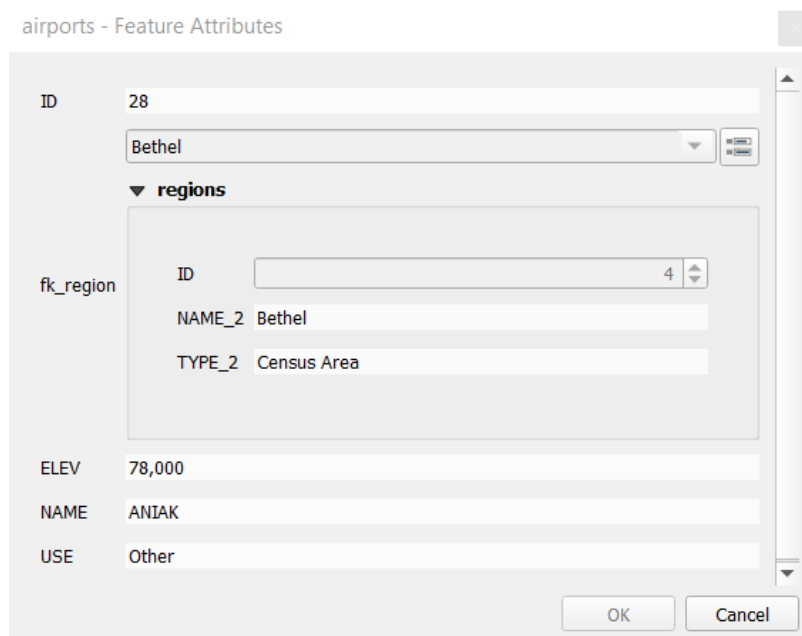



图 14.79: Identification dialog airport with relation to regions


The Relation Reference widget has also an option to embed the form of the parent layer within the child one. It is available in the *Properties* ► *Attributes Form* menu of the airport layer: select the `fk_region` field and check the *Show embedded form option*.

If you look at the feature dialog now, you will see, that the form of the region is embedded inside the airports form and will even have a combobox, which allows you to assign the current airport to another region.



Moreover if you toggle the editing mode of the airport layer, the `fk_region` field has also an autocompleter function: while typing you will see all the values of the `id` field of the region layer. Here it is possible to digitize a polygon for the region layer using the  button if you chose the option Allow adding new features in the *Properties* ► *Attributes Form* menu of the airport layer.

The child layer can also be used in the *Select Features By Value* tool in order to select features of the parent layer based on attributes of their children.

In , all the regions where the mean altitude of the airports is greater than 500 meters above sea level are selected.

You will find that many different aggregation functions are available in the form.

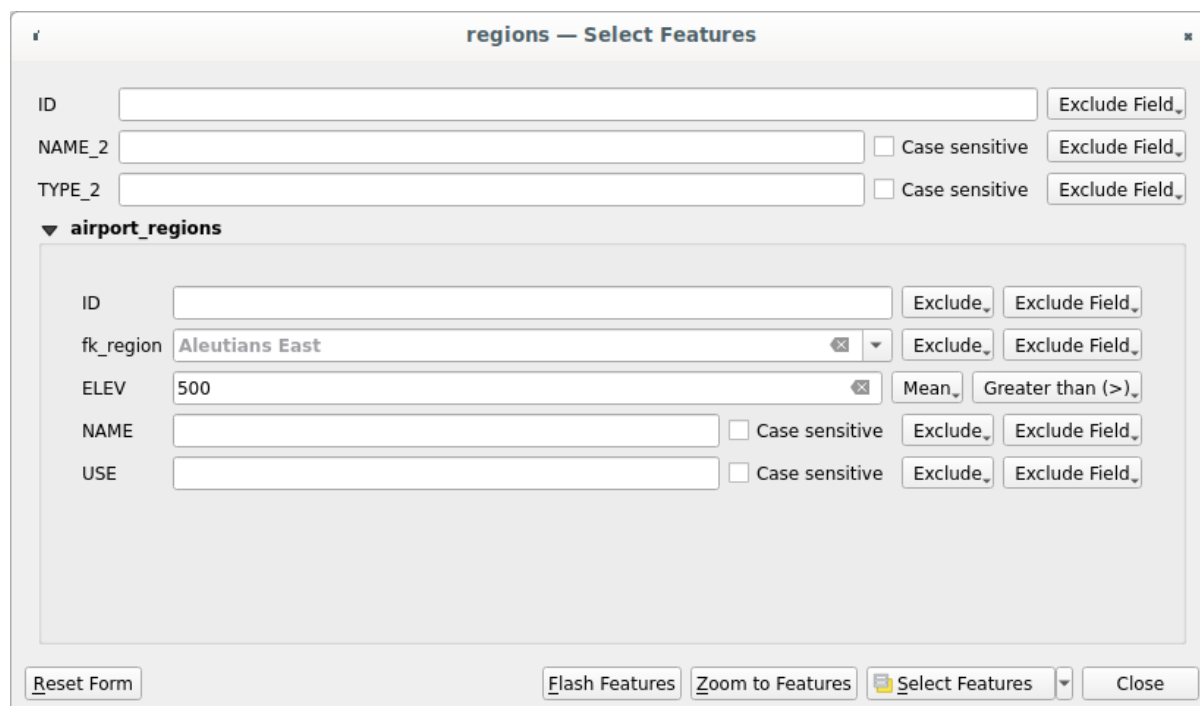


图 14.80: Select parent features with child values

Introducing many-to-many (N-M) relations

N-M relations are many-to-many relations between two tables. For instance, the `airports` and `airlines` layers: an airport receives several airline companies and an airline company flies to several airports.

This SQL code creates the three tables we need for an N-M relationship in a PostgreSQL/PostGIS schema named `locations`. You can run the code using the *Database* ► *DB Manager*... for PostGIS or external tools such as [pgAdmin](#). The `airports` table stores the `airports` layer and the `airlines` table stores the `airlines` layer. In both tables few fields are used for clarity. The *tricky* part is the `airports_airlines` table. We need it to list all airlines for all airports (or vice versa). This kind of table is known as a *pivot table*. The *constraints* in this table force that an airport can be associated with an airline only if both already exist in their layers.

```
CREATE SCHEMA locations;

CREATE TABLE locations.airports
(
  id serial NOT NULL,
  geom geometry(Point, 4326) NOT NULL,
  airport_name text NOT NULL,
  CONSTRAINT airports_pkey PRIMARY KEY (id)
```

(下页继续)

(续上页)

```

);

CREATE INDEX airports_geom_idx ON locations.airports USING gist (geom);

CREATE TABLE locations.airlines
(
  id serial NOT NULL,
  geom geometry(Point, 4326) NOT NULL,
  airline_name text NOT NULL,
  CONSTRAINT airlines_pkey PRIMARY KEY (id)
);

CREATE INDEX airlines_geom_idx ON locations.airlines USING gist (geom);

CREATE TABLE locations.airports_airlines
(
  id serial NOT NULL,
  airport_fk integer NOT NULL,
  airline_fk integer NOT NULL,
  CONSTRAINT airports_airlines_pkey PRIMARY KEY (id),
  CONSTRAINT airports_airlines_airport_fk_fkey FOREIGN KEY (airport_fk)
    REFERENCES locations.airports (id)
    ON DELETE CASCADE
    ON UPDATE CASCADE
    DEFERRABLE INITIALLY DEFERRED,
  CONSTRAINT airports_airlines_airline_fk_fkey FOREIGN KEY (airline_fk)
    REFERENCES locations.airlines (id)
    ON DELETE CASCADE
    ON UPDATE CASCADE
    DEFERRABLE INITIALLY DEFERRED
);

```

Instead of PostgreSQL you can also use GeoPackage. In this case, the three tables can be created manually using the *Database ► DB Manager...*. In GeoPackage there are no schemas so the *locations* prefix is not needed.

Foreign key constraints in *airports_airlines* table can't be created using *Table ► Create Table...* or *Table ► Edit Table...* so they should be created using *Database ► SQL Window...*. GeoPackage doesn't support *ADD CONSTRAINT* statements so the *airports_airlines* table should be created in two steps:

1. Set up the table only with the *id* field using *Table ► Create Table...*
2. Using *Database ► SQL Window...*, type and execute this SQL code:

```

ALTER TABLE airports_airlines
  ADD COLUMN airport_fk INTEGER
  REFERENCES airports (id)
  ON DELETE CASCADE
  ON UPDATE CASCADE
  DEFERRABLE INITIALLY DEFERRED;

ALTER TABLE airports_airlines
  ADD COLUMN airline_fk INTEGER
  REFERENCES airlines (id)
  ON DELETE CASCADE
  ON UPDATE CASCADE
  DEFERRABLE INITIALLY DEFERRED;

```

Then in QGIS, you should set up two *one-to-many relations* as explained above:

- a relation between *airlines* table and the pivot table;
- and a second one between *airports* table and the pivot table.

An easier way to do it (only for PostgreSQL) is using the *Discover Relations* in *Project ► Properties ► Relations*.

QGIS will automatically read all relations in your database and you only have to select the two you need. Remember to load the three tables in the QGIS project first.

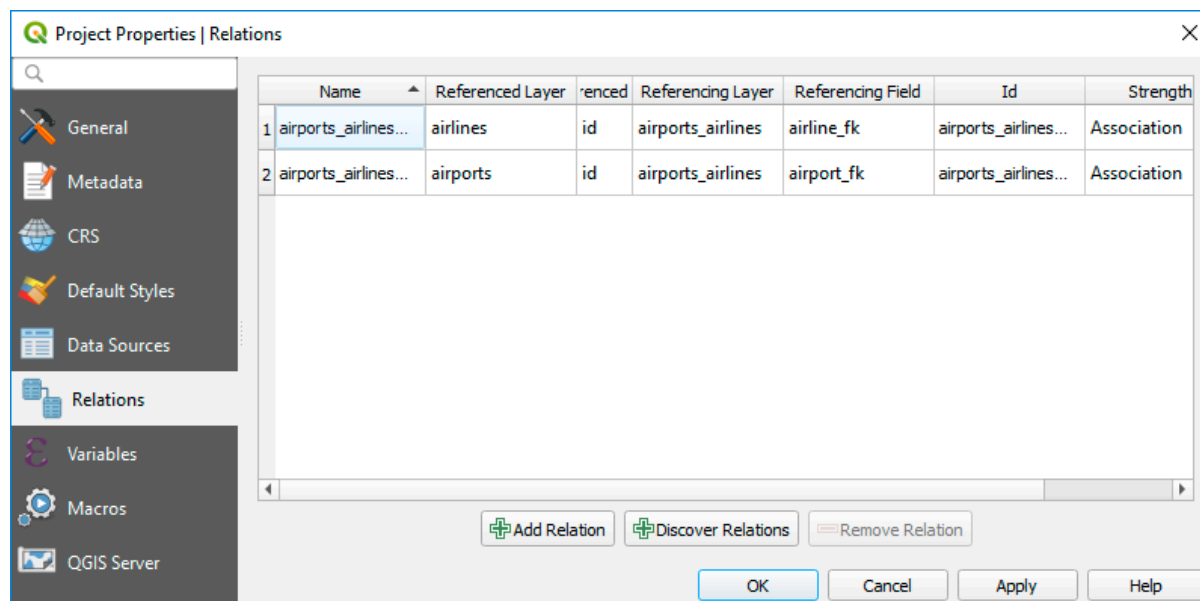


图 14.81: Relations and autodiscover

In case you want to remove an airport or an airline, QGIS won't remove the associated record(s) in `airports_airlines` table. This task will be made by the database if we specify the right *constraints* in the pivot table creation as in the current example.

注解: Combining N-M relation with automatic transaction group

You should enable the transaction mode in *Project Properties* ► *Data Sources* ► when working on such context. QGIS should be able to add or update row(s) in all tables (airlines, airports and the pivot tables).

Finally we have to select the right cardinality in the *Layer Properties* ► *Attributes Form* for the airports and airlines layers. For the first one we should choose the **airlines (id)** option and for the second one the **airports (id)** option.

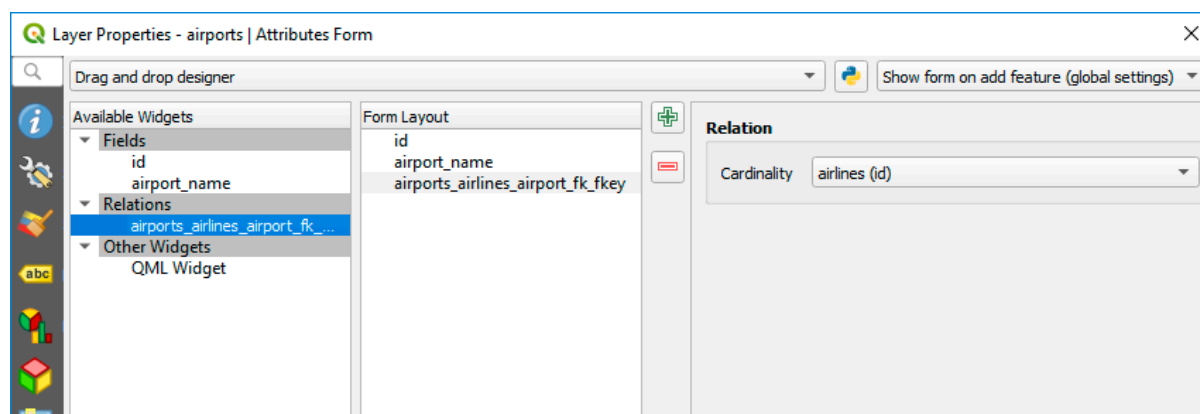


图 14.82: Set relationship cardinality

Now you can associate an airport with an airline (or an airline with an airport) using *Add child feature* or *Link existing child feature* in the subforms. A record will automatically be inserted in the `airports_airlines` table.

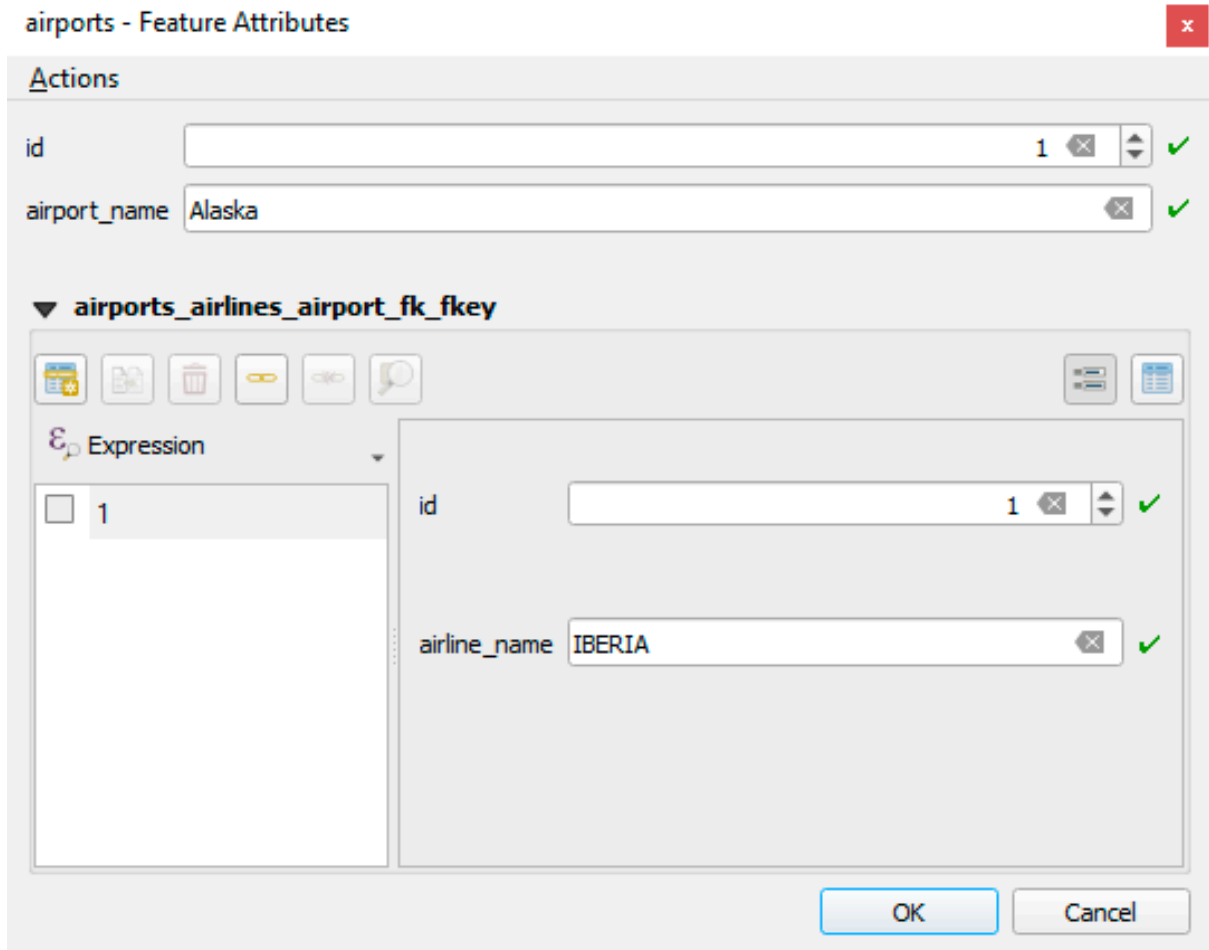


图 14.83: N-M relationship between airports and airlines

注解: Using **Many to one relation** cardinality

Sometimes hiding the pivot table in an N-M relationship is not desirable. Mainly because there are attributes in the relationship that can only have values when a relationship is established. If your tables are layers (have a geometry field) it could be interesting to activate the *On map identification* option (*Layer Properties* ► *Attributes Form* ► *Available widgets* ► *Fields*) for the foreign key fields in the pivot table.

注解: **Pivot table primary key**

Avoid using multiple fields in the primary key in a pivot table. QGIS assumes a single primary key so a constraint like `constraint airports_airlines_pkey primary key (airport_fk, airline_fk)` will not work.

14.5 Editing

QGIS has various capabilities for editing OGR, SpatiaLite, PostGIS, MSSQL Spatial and Oracle Spatial vector layers and tables.

注解: The procedure for editing GRASS layers is different - see section *Digitizing and editing a GRASS vector layer* for details.

小技巧: Concurrent Edits

This version of QGIS does not track if somebody else is editing the same feature at the same time as you are. The last person to save the edits wins.


14.5.1 Setting the Snapping Tolerance and Search Radius

For optimal and accurate editing of vector layer geometries, we need to set an appropriate value of snapping tolerance and search radius for features vertices.

Snapping tolerance

When you add a new vertex or move an existing one, the snapping tolerance is the distance QGIS uses to search for the closest vertex or segment you are trying to connect to. If you are not within the snapping tolerance, QGIS will leave the vertex where you release the mouse button, instead of snapping it to an existing vertex or segment.

The snapping tolerance setting affects all tools that work with tolerance.

You can enable / disable snapping by using the  Enable snapping button on the *Snapping Toolbar* or pressing `s`. The snapping mode, tolerance value, and units can also be configured in this toolbar.

The snapping configuration can also be set in *Project ► Snapping Options...*

There are three options to select the layer(s) to snap to:

- *All layers*: quick setting for all visible layers in the project so that the pointer snaps to all vertices and/or segments. In most cases, it is sufficient to use this snapping mode, but beware when using it for projects with many vector layers, as it may affect performance.
- *Current layer*: only the active layer is used, a convenient way to ensure topological consistency within the layer being edited.
- *Advanced Configuration*: allows you to enable and adjust snapping mode and tolerance on a layer basis (see [图 14.84](#)). If you need to edit a layer and snap its vertices to another, make sure that the target layer is checked and increase the snapping tolerance to a higher value. Snapping will not occur to a layer that is not checked in the snapping options dialog.

As for snapping mode, you can choose between `To vertex`, `To segment`, and `To vertex and segment`.

The tolerance values can be set either in the project's map units or in pixels. The advantage of choosing pixels is that it keeps the snapping constant at different map scales. 10 to 12 pixels is normally a good value, but it depends on the DPI of your screen. Using map units allows the tolerance to be related to real ground distances. For example, if you have a minimum distance between elements, this option can be useful to ensure that you don't add vertices too close to each other.

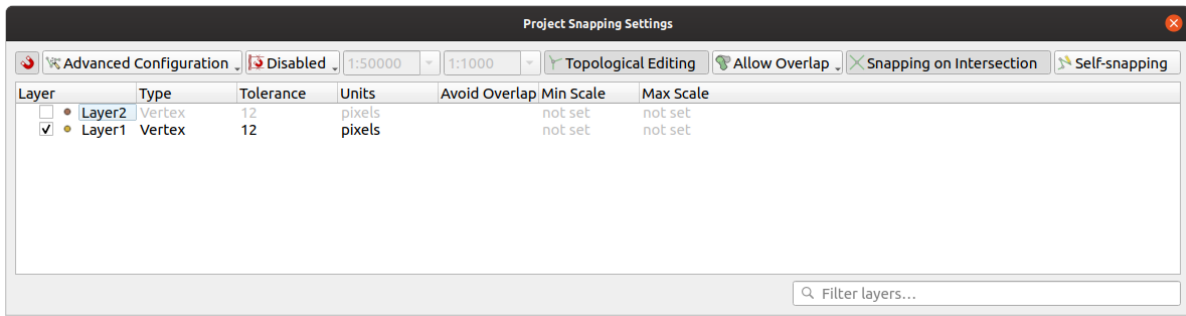


图 14.84: Snapping options (Advanced Configuration mode)

注解: By default, only visible features (the features whose style is displayed, except for layers where the symbology is "No symbols") can be snapped. You can enable the snapping on invisible features by checking *Enable snapping on invisible features* under the *Settings* ► *Options* ► *Digitizing* tab.

小技巧: Enable snapping by default

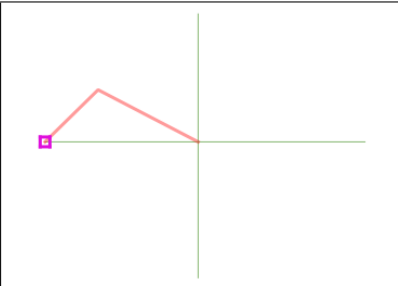
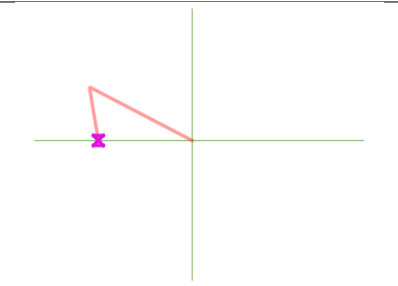
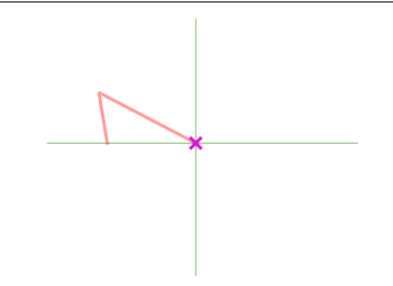
You can set snapping to be enabled by default on all new projects in the *Settings* ► *Options* ► *Digitizing* tab. You can also set the default snapping mode, tolerance value, and units, which will populate the *Snapping Options* dialog.

Enable snapping on intersections

Another available option is to use *snapping on intersection*, which allows you to snap to geometry intersections of snapping enabled layers, even if there are no vertices at the intersections.


Snapping icons

QGIS will show different *snap* icons depending on the kind of *snap*:

		
Snapping to a vertex: box icon	Snapping to a segment: hourglass icon	Snapping to an intersection: cross icon

Note that it is possible to change the color of these icons in the *Digitizing* part of your settings.

Search radius

Search radius for vertex edits is the distance QGIS uses to search for the vertex to select when you click on the map. If you are not within the search radius, QGIS will not find and select any vertex for editing. The search radius for vertex edits can be defined under the *Settings* ►  *Options* ► *Digitizing* tab (this is where you define the snapping default values).

Snap tolerance and search radius are set in `map units` or `pixels`. You may need to experiment to get them right. If you specify a too big tolerance, QGIS may snap to the wrong vertex, especially if you are dealing with a large number of vertices in close proximity. The smaller the search radius, the more difficult it will be to hit what you want to move.

Limit snapping to a scale range

In some cases snapping can become very slow. This is often caused by the amount of features in some layers that require a heavy index to compute and maintain. Some parameters exist to enable snapping only when the map view is inside a relevant scale range. This allows to only do the costly index computation related to snapping at a scale where drawing is relevant.

Scale limit to snapping is configured in *Project* ► *Snapping Options...* Limiting snapping to scale is only available in *Advanced Configuration* mode.

To limit snapping to a scale range you have three modes available:


- *Disabled*: Snapping is enabled whatever the current map scale is. This is the default mode.
- *Global*: Snapping is limited and only enabled when the current scale of the map is between a global minimum and a global maximum value. When selecting this mode two widgets become available to configure the range of scales in which snapping is enabled.
- *Per layer*: The snapping scale range limit is defined for each layer. When selecting this mode two columns become available to configure the minimum and maximum scales for each layer.

Please note that the minimum and maximum scales follow the QGIS convention: minimum scale is the most "zoomed out" scale while maximum scale is the most "zoomed in". A minimum or maximum scale that is set to "0" or "not set" is considered not limiting.

14.5.2 Topological editing


In addition to these snapping options, the *Snapping options...* dialog (*Project* ► *Snapping options*) and the *Snapping* toolbar allow you to enable / disable some other topological functionalities.

Enable topological editing

The  *Topological editing* button helps when editing and maintaining features with common boundaries. With this option enabled, QGIS 'detects' shared boundaries. When you move common vertices/segments, QGIS will also move them in the geometries of the neighboring features.

Topological editing works with features from different layers, as long as the layers are visible and in editing mode.

Avoid overlap of new polygons

When the snapping mode is set to *Advanced configuration*, for polygon layers, there's an option called  *Avoid overlap*. This option prevents you from drawing new features that overlap existing ones in the selected layer, speeding up digitizing of adjacent polygons.

With avoid overlap enabled, if you already have one polygon, you can digitize a second one such that they intersect. QGIS will then cut the second polygon to the boundary of the existing one. The advantage is that you don't have to digitize all vertices of the common boundary.

注解: If the new geometry is totally covered by existing ones, it gets cleared, and QGIS will show an error message.

警告: Use cautiously the *Avoid overlap* option


Since this option will cut new overlapping geometries of any polygon layer, you can get unexpected geometries if you forget to uncheck it when no longer needed.

Geometry Checker

A core plugin can help the user to find the geometry invalidity. You can find more information on this plugin at *Geometry Checker Plugin*.


Automatic Tracing

Usually, when using capturing map tools (add feature, add part, add ring, reshape and split), you need to click each vertex of the feature. With the automatic tracing mode, you can speed up the digitization process as you no longer need to manually place all the vertices during digitization:

1. Enable the  *Tracing* tool (in the *Snapping* toolbar) by pushing the icon or pressing T key.
2. *Snap to* a vertex or segment of a feature you want to trace along.
3. Move the mouse over another vertex or segment you'd like to snap and, instead of the usual straight line, the digitizing rubber band represents a path from the last point you snapped to the current position. The tool also works with curved geometries.

QGIS actually uses the underlying features topology to build the shortest path between the two points. Tracing requires snapping to be activated in traceable layers to build the path. You should also snap to an existing vertex or segment while digitizing and ensure that the two nodes are topologically connectable through existing features edges, otherwise QGIS is unable to connect them and thus traces a single straight line.

4. Click and QGIS places the intermediate vertices following the displayed path.

Unfold the  *Enable Tracing* icon and set the *Offset* option to digitize a path parallel to the features instead of tracing along them. A positive value shifts the new drawing to the left side of the tracing direction and a negative value does the opposite.

注解: Adjust map scale or snapping settings for an optimal tracing

If there are too many features in map display, tracing is disabled to avoid potentially long tracing structure preparation and large memory overhead. After zooming in or disabling some layers the tracing is enabled again.

注解: Does not add topological points

This tool does not add points to existing polygon geometries even if *Topological editing* is enabled. If geometry precision is activated on the edited layer, the resulting geometry might not exactly follow an existing geometry.

小技巧: Quickly enable or disable automatic tracing by pressing the T key

By pressing the T key, tracing can be enabled/disabled anytime (even while digitizing a feature), so it is possible to digitize parts of the feature with tracing enabled and other parts with tracing disabled. Tools behave as usual when tracing is disabled.

小技巧: Convert tracing to curved geometries

By using *Settings* ► *Options* ► *Digitizing* ► *Tracing* you can create curved geometries while digitizing. See *digitizing options*.

14.5.3 Digitizing an existing layer

By default, QGIS loads layers read-only. This is a safeguard to avoid accidentally editing a layer if there is a slip of the mouse. However, you can choose to edit any layer as long as the data provider supports it (see *Exploring Data Formats and Fields*), and the underlying data source is writable (i.e., its files are not read-only).

小技巧: Restrict edit permission on layers within a project

From the *Project* ► *Properties...* ► *Data Sources* ► *Layers Capabilities* table, you can choose to set any layer read-only regardless the provider permission. This can be a handy way, in a multi-users environment to avoid unauthorized users to mistakenly edit layers (e.g., Shapefile), hence potentially corrupt data. Note that this setting only applies inside the current project.

In general, tools for editing vector layers are divided into a digitizing and an advanced digitizing toolbar, described in section *Advanced digitizing*. You can select and unselect both under *View* ► *Toolbars* ►.

Using the basic digitizing tools, you can perform the following functions:


















Icon	目标	Icon	目标
	Current edits		Toggle editing
	Save layer edits		
	Add new record		Add Feature: Capture Point
	Add Feature: Capture Line		Add Feature: Capture Polygon
	Vertex Tool (All Layers)		Vertex Tool (Current Layer)
	Modify the attributes of all selected features simultaneously		
	Delete Selected		Cut Features
	Copy Features		Paste Features
	Undo		Redo


Table Editing: Vector layer basic editing toolbar

Note that while using any of the digitizing tools, you can still *zoom or pan* in the map canvas without losing the focus on the tool.





All editing sessions start by choosing the  **Toggle editing** option found in the context menu of a given layer, from the attribute table dialog, the digitizing toolbar or the *Edit* menu.


Once the layer is in edit mode, additional tool buttons on the editing toolbar will become available and markers will appear at the vertices of all features unless *Show markers only for selected features* option under *Settings ► Options... ► Digitizing* menu is checked.

小技巧: Save Regularly

Remember to  **Save Layer Edits** regularly. This will also check that your data source can accept all the changes.

Adding Features

Depending on the layer type, you can use the , **Add Record**, , **Add Point Feature**, , **Add Line Feature** or , **Add Polygon Feature** icons on the toolbar to add new features into the current layer.

To add a geometryless feature, click on the  **Add Record** button and you can enter attributes in the feature form that opens. To create features with the spatially enabled tools, you first digitize the geometry then enter its attributes. To digitize the geometry:

1. Left-click on the map area to create the first point of your new feature. For point features, this should be enough and trigger, if required, the feature form to fill in their attributes. Having set the *geometry precision* in the layer properties you can use *snap to grid* here to create features based on a regular distance.
2. For line or polygon geometries, keep on left-clicking for each additional point you wish to capture or use *automatic tracing* capability to accelerate the digitization. This will create consecutive straight lines between the vertices you place.

注解: Pressing **Delete** or **Backspace** key reverts the last node you add.

3. When you have finished adding points, right-click anywhere on the map area to confirm you have finished entering the geometry of that feature.

注解: While digitizing line or polygon geometries, you can switch back and forth between the linear *Add feature* tools and *circular string tools* to create compound curved geometries.

小技巧: Customize the digitizing rubber band

While capturing polygon, the by-default red rubber band can hide underlying features or places you'd like to capture a point. This can be fixed by setting a lower opacity (or alpha channel) to the rubber band's *Fill Color* in *Settings ► Options ► Digitizing* menu. You can also avoid the use of the rubber band by checking *Don't update rubber band during node editing*.

4. The attribute window will appear, allowing you to enter the information for the new feature. [图 14.85](#) shows setting attributes for a fictitious new river in Alaska. However, in the *Digitizing* menu under the *Settings ► Options* menu, you can also activate:
 - *Suppress attributes pop-up windows after each created feature* to avoid the form opening;
 - or *Reuse last entered attribute values* to have fields automatically filled at the opening of the form and just have to type changing values.

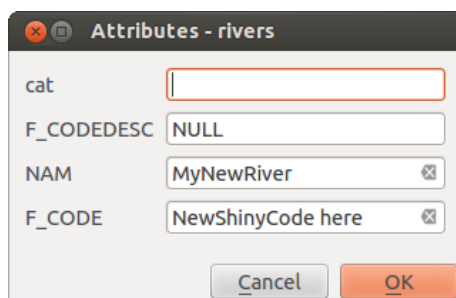




图 14.85: Enter Attribute Values Dialog after digitizing a new vector feature

Vertex tool


注解: QGIS 3 major changes

In QGIS 3, the node tool has been fully redesigned and renamed to *vertex tool*. It was previously working with "click and drag" ergonomics, and now uses a "click - click" workflow. This allows major improvements like taking profit of the advanced digitizing panel with the vertex tool while digitizing or editing objects of multiple layers at the same time.


For any editable vector layer, the  Vertex tool (Current Layer) provides manipulation capabilities of feature vertices similar to CAD programs. It is possible to simply select multiple vertices at once and to move, add or delete them altogether. The vertex tool also supports the topological editing feature. This tool is selection persistent, so when some operation is done, selection stays active for this feature and tool.

It is important to set the property *Settings* ►  *Options* ► *Digitizing* ► *Search Radius*: to a number greater than zero. Otherwise, QGIS will not be able to tell which vertex is being edited and will display a warning.

小技巧: Vertex Markers

The current version of QGIS supports three kinds of vertex markers: 'Semi-transparent circle', 'Cross' and 'None'. To change the marker style, choose  *Options* from the *Settings* menu, click on the *Digitizing* tab and select the appropriate entry.

Basic operations

Start by activating the  Vertex Tool (Current Layer). Red circles will appear when hovering vertices.

- **Selecting vertices:** You can select vertices by clicking on them one at a time holding *Shift* key pressed, or by clicking and dragging a rectangle around some vertices. When a vertex is selected, its color changes to blue. To add more vertices to the current selection, hold down the *Shift* key while clicking. To remove vertices from the selection, hold down *Ctrl*.
- **Batch vertex selection mode:** The batch selection mode can be activated by pressing *Shift+R*. Select a first node with one single click, and then hover **without clicking** another vertex. This will dynamically select all the nodes in between using the shortest path (for polygons).

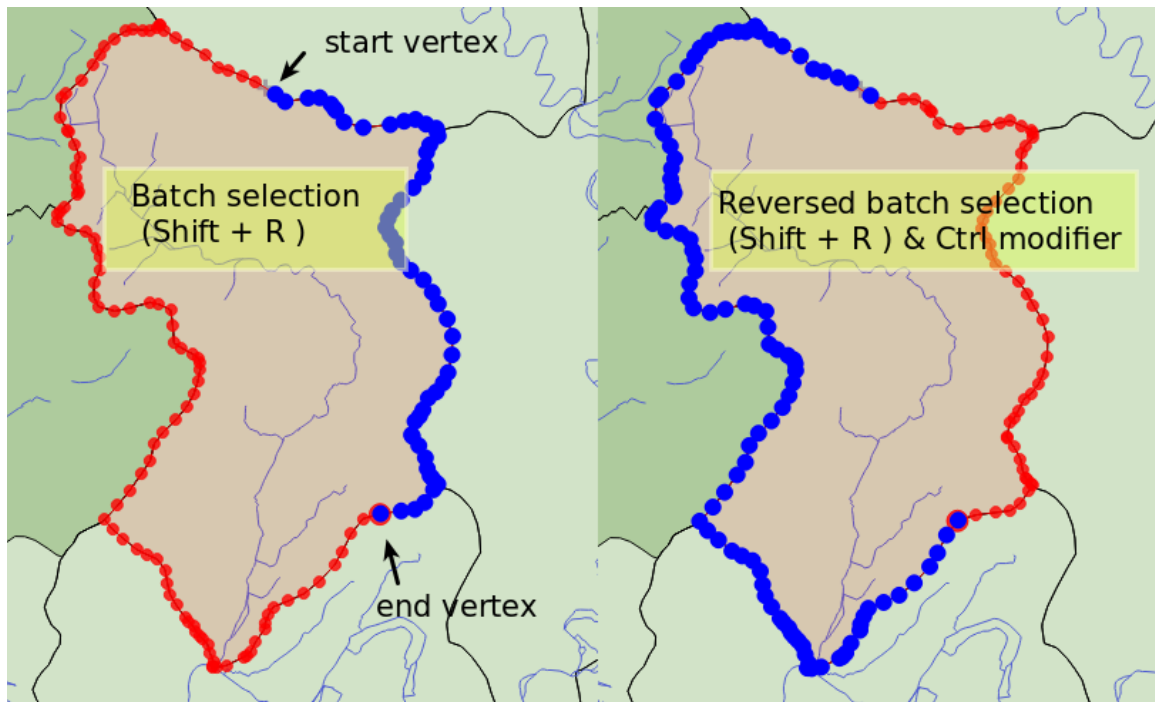


图 14.86: Batch vertex selection using Shift+R

Press `Ctrl` will invert the selection, selecting the longest path along the feature boundary. Ending your node selection with a second click, or pressing `Esc` will escape the batch mode.

- **Adding vertices:** To add a vertex, a virtual new node appears on the segment center. Simply grab it to add a new vertex. A double-click on any location of the boundary also creates a new node. For lines, a virtual node is also proposed at both extremities of a line to extend it.

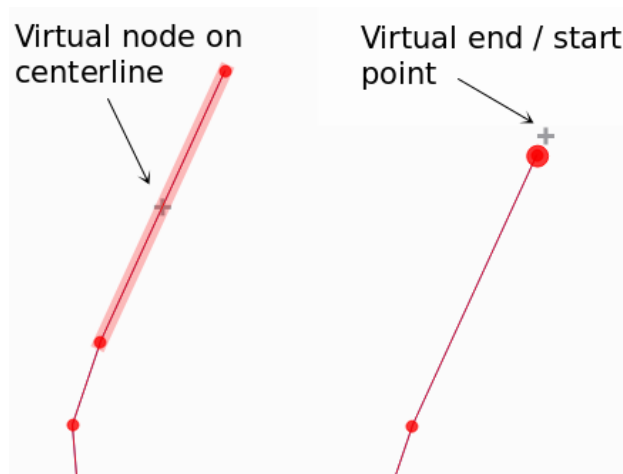



图 14.87: Virtual nodes for adding vertices

- **Deleting vertices:** Select the vertices and click the `Delete` key. Deleting all the vertices of a feature generates, if compatible with the datasource, a geometryless feature. Note that this doesn't delete the complete feature, just the geometry part. To delete a complete feature use the  `Delete Selected` tool.
- **Moving vertices:** Select all the vertices you want to move, click on a selected vertex or edge, and click again on the desired new location. All the selected vertices will move together. If snapping is enabled, the whole selection can jump to the nearest vertex or line. You can use Advanced Digitizing Panel constraints for distance, angles, exact X Y location before the second click.

Here you can use the snap-to-grid feature. Having set a value for the *geometry precision* in the layer properties, a grid appears on a zoom level according to the Geometry precision.

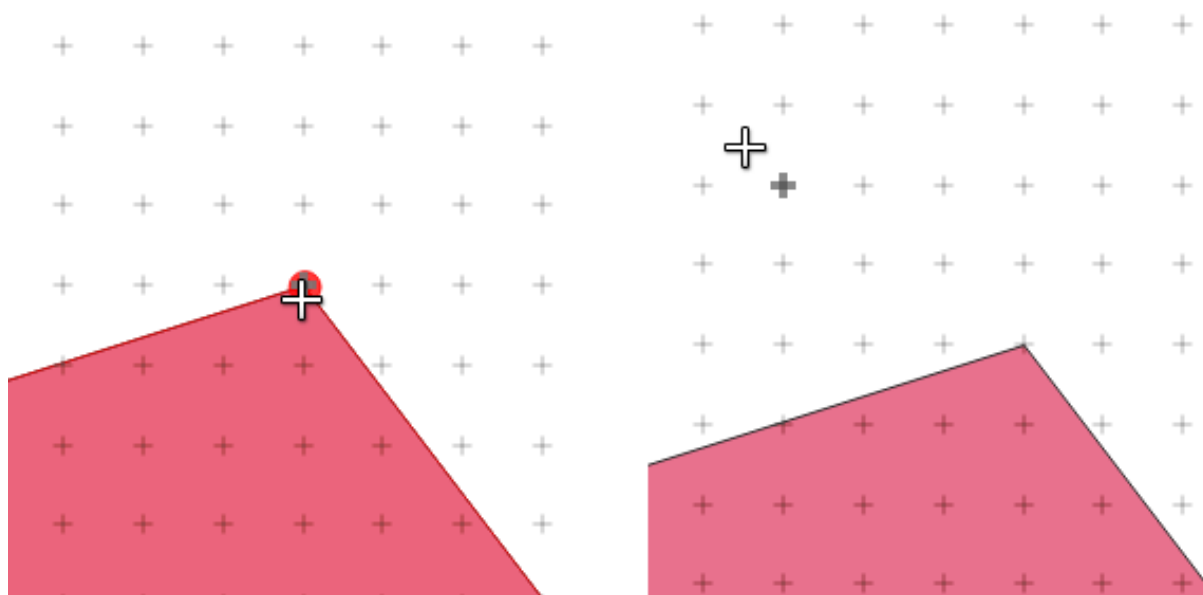


图 14.88: Selecting a vertex and moving the vertices to grid

Each change made with the vertex is stored as a separate entry in the *Undo* dialog. Remember that all operations support topological editing when this is turned on. On-the-fly projection is also supported, and the vertex tool provides tooltips to identify a vertex by hovering the pointer over it.

The Vertex Editor Panel

When using the *Vertex tool* on a feature, it is possible to right click to open the *Vertex Editor* panel listing all the vertices of the feature with their x , y (z , m if applicable) coordinates and r (for the radius, in case of circular geometry). Simply select a row in the table does select the corresponding vertex in the map canvas, and vice versa. Simply change a coordinate in the table and your vertex position is updated. You can also select multiple rows and delete them altogether.

注解: Changed behavior in QGIS 3.4

Right click on a feature will immediately show the vertex editor and lock this feature, thus disabling the editing of any other features. While being locked, a feature is exclusive for editing: Selecting and moving of vertices and segments by clicking or dragging is only possible for this feature. New vertices can only be added to the locked feature. Also, the vertex editor panel now opens itself automatically upon activating the vertex tool, and its position/docked state remembered across uses.

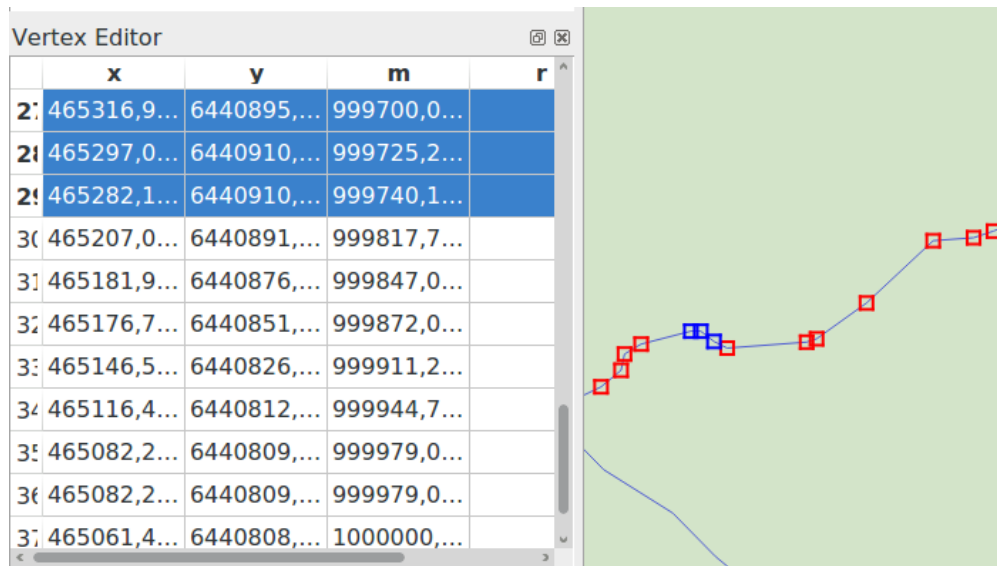



图 14.89: Vertex editor panel showing selected nodes

Cutting, Copying and Pasting Features

Selected features can be cut, copied and pasted between layers in the same QGIS project, as long as destination layers are set to  Toggle editing beforehand.




小技巧: Transform polygon into line and vice-versa using copy/paste

Copy a line feature and paste it in a polygon layer: QGIS pastes in the target layer a polygon whose boundary corresponds to the closed geometry of the line feature. This is a quick way to generate different geometries of the same data.

Features can also be pasted to external applications as text. That is, the features are represented in CSV format, with the geometry data appearing in the OGC Well-Known Text (WKT) format. WKT and GeoJSON features from outside QGIS can also be pasted to a layer within QGIS.

When would the copy and paste function come in handy? Well, it turns out that you can edit more than one layer at a time and copy/paste features between layers. Why would we want to do this? Say we need to do some work on a new layer but only need one or two lakes, not the 5,000 on our `big_lakes` layer. We can create a new layer and use copy/paste to plop the needed lakes into it.

As an example, we will copy some lakes to a new layer:

1. Load the layer you want to copy from (source layer)
2. Load or create the layer you want to copy to (target layer)
3. Start editing for target layer
4. Make the source layer active by clicking on it in the legend
5. Use the  Select Features by area or single click tool to select the feature(s) on the source layer
6. Click on the  Copy Features tool
7. Make the destination layer active by clicking on it in the legend
8. Click on the  Paste Features tool
9. Stop editing and save the changes

What happens if the source and target layers have different schemas (field names and types are not the same)? QGIS populates what matches and ignores the rest. If you don't care about the attributes being copied to the target layer, it doesn't matter how you design the fields and data types. If you want to make sure everything - the feature and its attributes - gets copied, make sure the schemas match.



注解: Congruency of Pasted Features



If your source and destination layers use the same projection, then the pasted features will have geometry identical to the source layer. However, if the destination layer is a different projection, then QGIS cannot guarantee the geometry is identical. This is simply because there are small rounding-off errors involved when converting between projections.

小技巧: Copy string attribute into another



If you have created a new column in your attribute table with type 'string' and want to paste values from another attribute column that has a greater length the length of the column size will be extended to the same amount. This is because the GDAL Shapefile driver starting with GDAL/OGR 1.10 knows to auto-extend string and integer fields to dynamically accommodate for the length of the data to be inserted.

Deleting Selected Features

If we want to delete an entire feature (attribute and geometry), we can do that by first selecting the geometry using the regular  Select Features by area or single click tool. Selection can also be done from the attribute table. Once you have the selection set, press **Delete** or **Backspace** key or use the  Delete Selected tool to delete the features. Multiple selected features can be deleted at once.

The  Cut Features tool on the digitizing toolbar can also be used to delete features. This effectively deletes the feature but also places it on a "spatial clipboard". So, we cut the feature to delete. We could then use the  Paste Features tool to put it back, giving us a one-level undo capability. Cut, copy, and paste work on the currently selected features, meaning we can operate on more than one at a time.

Undo and Redo

The  Undo and  Redo tools allows you to undo or redo vector editing operations. There is also a dockable widget, which shows all operations in the undo/redo history (see [图 14.90](#)). This widget is not displayed by default; it can be displayed by right-clicking on the toolbar and activating the *Undo/Redo Panel* checkbox. The Undo/Redo capability is however active, even if the widget is not displayed.

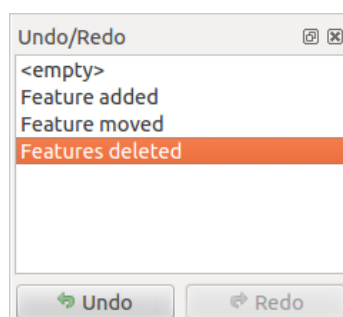




图 14.90: Redo and Undo digitizing steps

When Undo is hit or **Ctrl+Z** (or **Cmd+Z**) pressed, the state of all features and attributes are reverted to the state before the reverted operation happened. Changes other than normal vector editing operations (for example, changes

done by a plugin) may or may not be reverted, depending on how the changes were performed.

To use the undo/redo history widget, simply click to select an operation in the history list. All features will be reverted to the state they were in after the selected operation.

Saving Edited Layers




When a layer is in editing mode, any changes remain in the memory of QGIS. Therefore, they are not committed/saved immediately to the data source or disk. If you want to save edits to the current layer but want to continue editing without leaving the editing mode, you can click the  Save Layer Edits button. When you turn editing mode off with  Toggle editing (or quit QGIS for that matter), you are also asked if you want to save your changes or discard them.

If the changes cannot be saved (e.g., disk full, or the attributes have values that are out of range), the QGIS in-memory state is preserved. This allows you to adjust your edits and try again.

小技巧: Data Integrity

It is always a good idea to back up your data source before you start editing. While the authors of QGIS have made every effort to preserve the integrity of your data, we offer no warranty in this regard.

Saving multiple layers at once

This feature allows the digitization of multiple layers. Choose  Save for Selected Layers to save all changes you made in multiple layers. You also have the opportunity to  Rollback for Selected Layers, so that the digitization may be withdrawn for all selected layers. If you want to stop editing the selected layers,  Cancel for Selected Layer(s) is an easy way.

The same functions are available for editing all layers of the project.

小技巧: Use transaction group to edit, save or rollback multiple layers changes at once

When working with layers from the same PostgreSQL database, activate the *Automatically create transaction groups where possible* option in *Project ► Properties... ► Data Sources* to sync their behavior (enter or exit the edit mode, save or rollback changes at the same time).

14.5.4 Advanced digitizing























Icon	目标	Icon	目标
	Enable Advanced Digitizing Tools		Enable Tracing
	Move Feature(s)		Copy and Move Feature(s)
	Rotate Feature(s)		Simplify Feature
	Add Ring		Add Part
	Fill Ring		Swap direction
	Delete Ring		Delete Part
	Offset Curve		Reshape Features
	Split Parts		Split Features
	合并 (merge) 选中要素的属性		合并 (merge) 选中的要素
	Rotate Point Symbols		Offset Point Symbols
	Trim or Extend Feature		



Table Advanced Editing: Vector layer advanced editing toolbar


Move Feature(s)

The  Move Feature(s) tool allows you to move existing features:

1. Select the feature(s) to move.
2. Click on the map canvas to indicate the origin point of the displacement; you can rely on snapping capabilities to select an accurate point.


You can also take advantages of the *advanced digitizing constraints* to accurately set the origin point coordinates. In that case:


1. First click on the  button to enable the panel.
2. Type x and enter the corresponding value for the origin point you'd like to use. Then press the  button next to the option to lock the value.
3. Do the same for the y coordinate.
4. Click on the map canvas and your origin point is placed at the indicated coordinates.
3. Move over the map canvas to indicate the destination point of the displacement, still using snapping mode or, as above, use the advanced digitizing panel which would provide complementary distance and angle placement constraints to place the end point of the translation.
4. Click on the map canvas: the whole features are moved to new location.

Likewise, you can create a translated copy of the feature(s) using the  Copy and Move Feature(s) tool.

注解: If no feature is selected when you first click on the map canvas with any of the *Move Feature(s)* or *Copy and Move Feature(s)* tools, then only the feature under the mouse is affected by the action. So, if you want to move several features, they should be selected first.


Rotate Feature(s)

Use the  Rotate Feature(s) tool to rotate one or multiple features in the map canvas:


1. Press the  Rotate Feature(s) icon
2. Then click on the feature to rotate. The feature's centroid is referenced as rotation center, a preview of the rotated feature is displayed and a widget opens showing the current *Rotation* angle.
3. Click on the map canvas when you are satisfied with the new placement or manually enter the rotation angle in the text box. You can also use the *Snap to* ° box to constrain the rotation values.
4. If you want to rotate several features at once, they shall be selected first, and the rotation is by default around the centroid of their combined geometries.


You can also use an anchor point different from the default feature centroid: press the `Ctrl` button, click on the map canvas and that point will be used as the new rotation center.

If you hold `Shift` before clicking on the map, the rotation will be done in 45 degree steps, which can be modified afterwards in the user input widget.

To abort feature rotation, press the `ESC` button or click on the  Rotate Feature(s) icon.


Simplify Feature

The  Simplify Feature tool allows you to interactively reshape a line or polygon geometry by reducing or densifying the number of vertices, as long as the geometry remains valid:


1. Select the  Simplify Feature tool.
2. Click on the feature or drag a rectangle over the features.
3. A dialog pops up allowing you to define the *Method* to apply, ie whether you would like to:
 - *simplify the geometry*, meaning less vertices than the original. Available methods are Simplify by distance, Simplify by snapping to grid or simplify by area (Visvalingam). You'd then need to indicate the value of *Tolerance* in Layer units, Pixels or map units to use for simplification. The higher the tolerance is the more vertices can be deleted.
 - or *densify the geometries* with new vertices thanks to the `Smooth` option: for each existing vertex, two vertices are placed on each of the segments originated from it, at an *Offset* distance representing the percentage of the segment length. You can also set the number of *Iterations* the placement would be processed: the more iterations, the more vertices and smoother is the feature.



Settings that you used will be saved when leaving a project or an edit session. So you can go back to the same parameters the next time you simplify a feature.

4. A summary of the modifications that would apply is shown at the bottom of the dialog, listing number of features and number of vertices (before and after the operation and the ratio the change represents). Also, in the map canvas, the expected geometry is displayed over the existing one, using the rubberband color.
5. When the expected geometry fits your needs, click *OK* to apply the modification. Otherwise, to abort the operation, you can either press *Cancel* or right-click in the map canvas.


注解: Unlike the feature simplification option in *Settings* ► *Options* ► *Rendering* menu which simplifies the geometry just for rendering, the  Simplify Feature tool permanently modifies feature's geometry in data source.

Add Part


You can  Add Part to a selected feature generating a multipoint, multiline or multipolygon feature. The new part must be digitized outside the existing one which should be selected beforehand.

The  Add Part can also be used to add a geometry to a geometryless feature. First, select the feature in the attribute table and digitize the new geometry with the  Add Part tool.


Delete Part




The  Delete Part tool allows you to delete parts from multifeatures (e.g., to delete polygons from a multi-polygon feature). This tool works with all multi-part geometries: point, line and polygon. Furthermore, it can be used to totally remove the geometric component of a feature. To delete a part, simply click within the target part.

Add Ring

You can create ring polygons using the  Add Ring icon in the toolbar. This means that inside an existing area, it is possible to digitize further polygons that will occur as a 'hole', so only the area between the boundaries of the outer and inner polygons remains as a ring polygon.


Fill Ring

The  Fill Ring tool helps you create polygon feature that totally falls within another one without any overlapping area; that is the new feature covers a hole within the existing one. To create such a feature:

1. Select the  Fill Ring tool.
2. Draw a new polygon over the existing feature: QGIS adds a ring to its geometry (like if you used the  Add Ring tool) and creates a new feature whose geometry matches the ring (like if you *traced* over the interior boundaries with the  Add polygon feature tool).
3. Or alternatively, if the ring already exists on the feature, place the mouse over the ring and left-click while pressing **Shift**: a new feature filling the hole is drawn at that place.

The *Feature Attributes* form of the new feature opens, pre-filled with values of the "parent" feature and/or *fields constraints*.

Delete Ring

The  Delete Ring tool allows you to delete rings within an existing polygon, by clicking inside the hole. This tool only works with polygon and multi-polygon features. It doesn't change anything when it is used on the outer ring of the polygon.

Reshape Features

You can reshape line and polygon features using the  Reshape Features tool on the toolbar. For lines, it replaces the line part from the first to the last intersection with the original line.

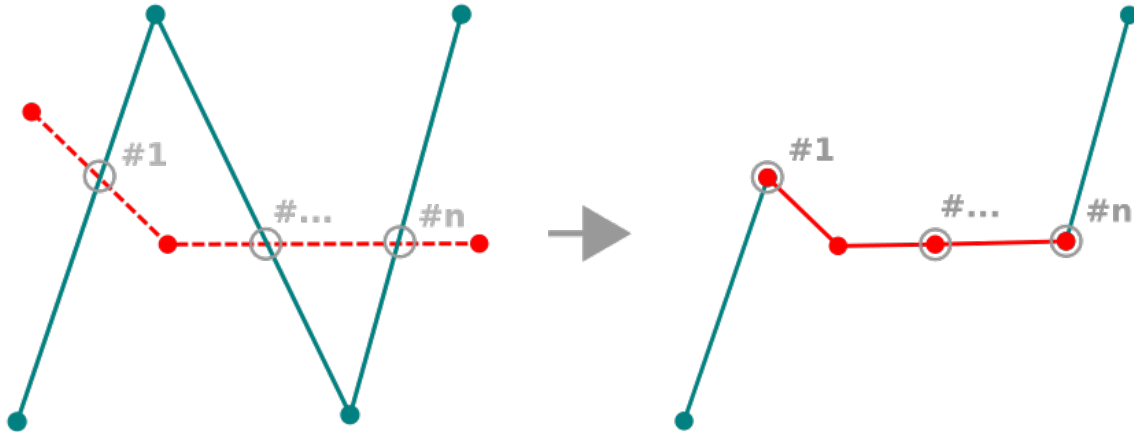



图 14.91: Reshape line

小技巧: Extend linestring geometries with reshape tool

Use the  Reshape Features tool to extend existing linestring geometries: snap to the first or last vertex of the line and draw a new one. Validate and the feature's geometry becomes the combination of the two lines.

For polygons, it will reshape the polygon's boundary. For it to work, the reshape tool's line must cross the polygon's boundary at least twice. To draw the line, click on the map canvas to add vertexes. To finish it, just right-click. Like with the lines, only the segment between the first and the last intersections is considered. The reshape line's segments that are inside the polygon will result in cropping it, where the ones outside the polygon will extend it.

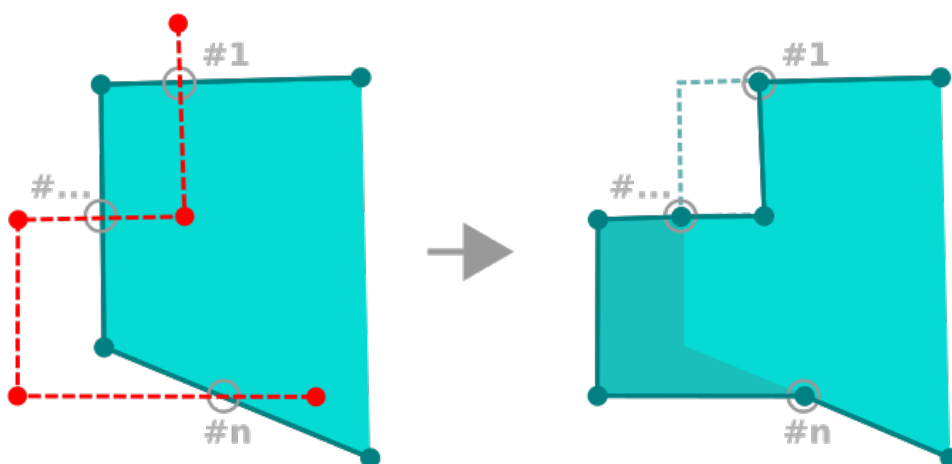



图 14.92: Reshape polygon


With polygons, reshaping can sometimes lead to unintended results. It is mainly useful to replace smaller parts of


a polygon, not for major overhauls, and the reshape line is not allowed to cross several polygon rings, as this would generate an invalid polygon.

注解: The reshape tool may alter the starting position of a polygon ring or a closed line. So, the point that is represented 'twice' will not be the same any more. This may not be a problem for most applications, but it is something to consider.

Offset Curves

The  Offset Curve tool creates parallel shifts of line layers. The tool can be applied to the edited layer (the geometries are modified) or also to background layers (in which case it creates copies of the lines / rings and adds them to the edited layer). It is thus ideally suited for the creation of distance line layers. The *User Input* dialog pops-up, showing the displacement distance.

To create a shift of a line layer, you must first go into editing mode and activate the  Offset Curve tool. Then click on a feature to shift it. Move the mouse and click where wanted or enter the desired distance in the user input widget.


Your changes may then be saved with the  Save Layer Edits tool.

QGIS options dialog (Digitizing tab then **Curve offset tools** section) allows you to configure some parameters like **Join style**, **Quadrant segments**, **Miter limit**.


Reverse Line

Changing the direction of a line geometry can be useful for cartographical purposes or when preparing for network analysis.


To change a line direction:

1. Activate the reverse line tool by clicking  Reverse line.
2. Click on the line. The direction of the line is reversed.


Split Features

Use the  Split Features tool to split a feature into two or more new and independent features, ie. each geometry corresponding to a new row in the attribute table.


To split line or polygon features:

1. Select the  Split Features tool.
2. Draw a line across the feature(s) you want to split. If a selection is active, only selected features are split. When set, *default values or clauses* are applied to corresponding fields and other attributes of the parent feature are by default copied to the new features.
3. You can then as usually modify any of the attributes of any resulting feature.


小技巧: Split a polyline into new features in one-click

Using the  Split Features tool, snap and click on an existing vertex of a polyline feature to split that feature into two new features.


Split parts


In QGIS it is possible to split the parts of a multi part feature so that the number of parts is increased. Just draw a line across the part you want to split using the  Split Parts icon.

小技巧: Split a polyline into new parts in one-click

Using the  Split Parts tool, snap and click on an existing vertex of a polyline feature to split the feature into two new polylines belonging to the same feature.

Merge selected features


The  Merge Selected Features tool allows you to create a new feature by merging existing ones: their geometries are merged to generate a new one. If features don't have common boundaries, a multipolygon/multipolyline/multipoint feature is created.

1. First, select the features you'd like to combine.
2. Then press the  Merge Selected Features button.
3. In the new dialog, the *Merge* line at the bottom of the table shows the attributes of the resulting feature. You can alter any of these values either by:
 - manually replacing the value in the corresponding cell;
 - selecting a row in the table and pressing *Take attributes from selected feature* to use the values of this initial feature;
 - pressing *Skip all fields* to use empty attributes;
 - or, expanding the drop down menu at the top of the table, select any of the above options to apply to the corresponding field only. There, you can also choose to aggregate the initial features attributes (Minimum, Maximum, Median, Sum, Count, Concatenation... depending on the type of the field. see [Statistical Summary Panel](#) for the full list of functions).


注解: If the layer has default values or clauses present on fields, these are used as the initial value for the merged feature.

4. Press *OK* to apply the modifications. A single (multi)feature is created in the layer, replacing the previously selected ones.


Merge attributes of selected features

The  Merge Attributes of Selected Features tool allows you to apply same attributes to features without merging their boundaries. The dialog is the same as the Merge Selected Features tool's except that unlike that tool, selected objects are kept with their geometry while some of their attributes are made identical.

Rotate Point Symbols

The  Rotate Point Symbols allows you to individually change the rotation of point symbols in the map canvas.

1. First, you need to indicate the field to store the rotation value in. This is made by assigning a field to the symbol *data-defined* rotation property:

1. In the *Layer Properties* ► *Symbology* dialog, browse to the symbol editor dialog.
2. Click the  *Data-defined override* widget near the *Rotation* option of the top *Marker* level (preferably) of the symbol layers.
3. Choose a field in the *Field Type* combobox. Values of this field are hence used to rotate each feature's symbol accordingly.

You can also check the *Store data in project* entry to generate an *auxiliary data storage* field to control the rotation value.

注解: Make sure that the same field is assigned to all the symbol layers

Setting the data-defined rotation field at the topmost level of the symbol tree automatically propagates it to all the symbol layers, a prerequisite to perform graphical symbol rotation with the *Rotate Point Symbols* tool. Indeed, if a symbol layer does not have the same field attached to its rotation property, the tool will not work.

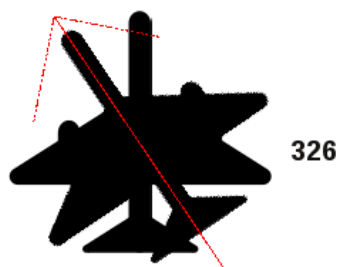





图 14.93: Rotating a point symbol


2. Then click on a point symbol in the map canvas with the  Rotate Point Symbols tool
3. Move the mouse around. A red arrow with the rotation value will be visualized (see 图 14.93). If you hold the `Ctrl` key while moving, the rotation will be done in 15 degree steps.
4. When you get the expected angle value, click again. The symbol is rendered with this new rotation and the associated field is updated accordingly.

You can right-click to abort symbol rotation.




Offset Point Symbols

The  Offset Point Symbols allows you to interactively change the rendered position of point symbols in the map canvas.

This tool behaves like the  Rotate Point Symbols tool except that it requires you to connect a field to the data-defined *Offset (X, Y)* property of each layer of the symbol. The field will then be populated with the offset coordinates for the features whose symbol is moved in the map canvas.


1. Associate a field to the data-defined widget of the *Offset (X, Y)* property of the symbol. If the symbol is made with many layers, you may want to assign the field to each of them
2. Select the  Offset Point Symbols tool
3. Click a point symbol
4. Move to a new location
5. Click again. The symbol is moved to the new place. Offset values from the original position are stored in the linked field.

You can right-click to abort symbol offset.

注解: The  Offset Point Symbols tool doesn't move the point feature itself; you should use the  Vertex Tool (Current Layer) or  Move Feature tool for this purpose.

Trim/Extend Feature

When a digitized line is too short or too long to snap to another line (missing or crossing the line), it is necessary to be able to extend or shorten the segment.

The  Trim/Extend tool allows you to also modify (multi)lines AND (multi)polygons. Moreover, it is not necessarily the end of the lines that is concerned; any segment of a geometry can be modified.

注解: This can lead to invalid geometries.

注解: You must activate segment snapping for this tool to work.

The tool asks you to select a limit (a segment) with respect to which another segment will be extended or trimmed. Unlike the vertex tool, a check is performed to modify only the layer being edited.



When both segments are in 3D, the tool performs an interpolation on the limit segment to get the Z value.

In the case of a trim, you must select the part that will be shortened by clicking on it.

14.5.5 Shape digitizing

The *Shape Digitizing* toolbar offers a set of tools to draw regular shapes and curved geometries.

Add Circular string

The  Add circular string or  Add circular string by radius buttons allow users to add line or polygon features with a circular geometry.

Creating features with these tools follow the same rule as of other digitizing tools: left-click to place vertices and right-click to finish the geometry. While drawing the geometry, you can switch from one tool to the other as well as to the *linear geometry tools*, creating some compound geometries.






注解: Curved geometries are stored as such only in compatible data provider

Although QGIS allows to digitize curved geometries within any editable data format, you need to be using a data provider (e.g. PostGIS, memory layer, GML or WFS) that supports curves to have features stored as curved, otherwise QGIS segmentizes the circular arcs.

Draw Circles

There is a set of tools for drawing circles. The tools are described below.





Circles are converted into circular strings. Therefore, as explained in *Add Circular string*, if allowed by the data provider, it will be saved as a curved geometry, if not, QGIS will segmentize the circular arcs.

-  Add circle from 2 points: The two points define the diameter and the orientation of the circle. (Left-click, right-click)
-  Add circle from 3 points: Draws a circle from three known points on the circle. (Left-click, left-click, right-click)
-  Add circle from center and a point: Draws a circle with a given center and a point on the circle (Left-click, right-click). When used with the *The Advanced Digitizing panel* this tool can become a "Add circle from center and radius" tool by setting and locking the distance value after first click.
-  Add circle from 3 tangents: Draws a circle that is tangential to three segments. **Note that you must activate snapping to segments** (See *Setting the Snapping Tolerance and Search Radius*). Click on a segment to add a tangent. If two tangents are parallel, an error message appears and the input is cleared. (Left-click, left-click, right-click)
-  Add circle from 2 tangents and a point: Similar to circle from 3 tangents, except that you have to select two tangents, enter a radius and select the desired center.

Draw Ellipses





There is a set of tools for drawing ellipses. The tools are described below.

Ellipses cannot be converted as circular strings, so they will always be segmented.

-  Add Ellipse from center and two points: Draws an ellipse with a given center, major axis and minor axis. (Left-click, left-click, right-click)
-  Add Ellipse from center and a point: Draws an ellipse into a bounding box with the center and a corner. (Left-click, right-click)
-  Add Ellipse from extent: Draws an ellipse into a bounding box with two opposite corners. (Left-click, right-click)
-  Add Ellipse from foci: Draws an ellipse by 2 points for foci and a point on the ellipse. (Left-click, left-click, right-click)

Draw Rectangles

There is a set of tools for drawing rectangles. The tools are described below.

-  Rectangle from center and a point: Draws a rectangle from the center and a corner. (Left-click, right-click)
-  Rectangle from extent: Draws a rectangle from two opposite corners. (Left-click, right-click)
-  Rectangle from 3 points (distance): Draws an oriented rectangle from three points. The first and second points determine the length and angle of the first edge. The third point determines the length of the other edge. One can use *The Advanced Digitizing panel* to set the length of the edges. (Left-click, left-click, right-click)
-  Rectangle from 3 points (projected): Same as the preceding tool, but the length of the second edge is computed from the projection of the third point on the first edge. (Left-click, left-click, right-click)

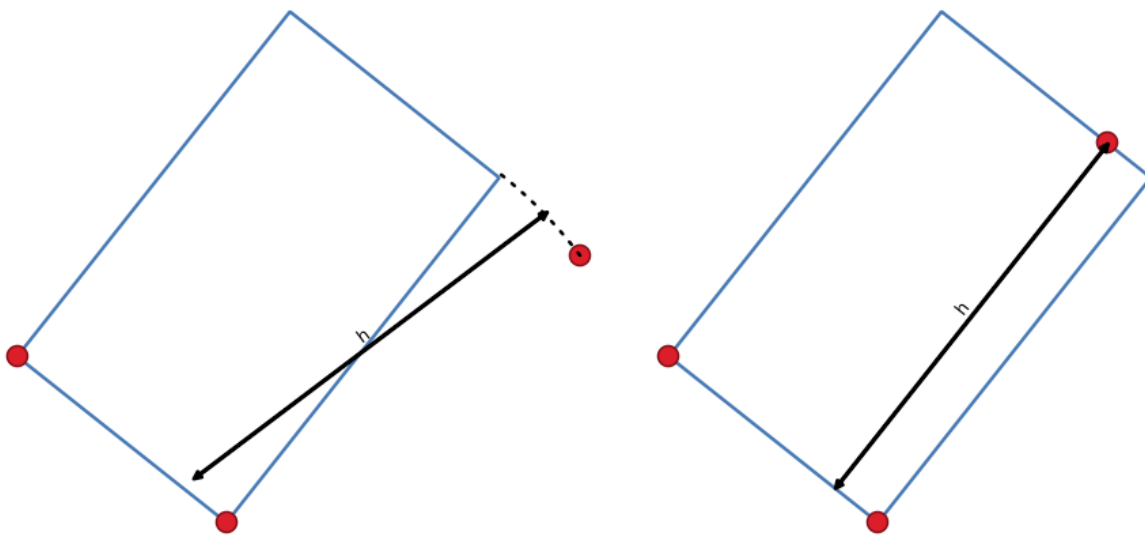





图 14.94: Draw rectangle from 3 points using distance (right) and projected (left)

Draw Regular Polygons

There is a set of tools for drawing regular polygons. The tools are described below. Left-click to place the first point. A dialog appears, where you can set the number of polygon edges. Right-click to finish the regular polygon.

-  Regular polygon from two points: Draws a regular polygon where the two points determine the length and angle of the first edge.
-  Regular polygon from center and a point: Draws a regular polygon from the provided center point. The second point determines the angle and distance to the middle of an edge.
-  Regular polygon from center and a corner: Same as the preceding tool, but the second point determines the angle and distance to a vertex.

14.5.6 The Advanced Digitizing panel

When capturing, reshaping, splitting new or existing geometries you also have the possibility to use the Advanced Digitizing panel. You can digitize lines exactly parallel or perpendicular to a particular angle or lock lines to specific angles. Furthermore, you can enter coordinates directly so that you can make a precise definition of your new geometry.

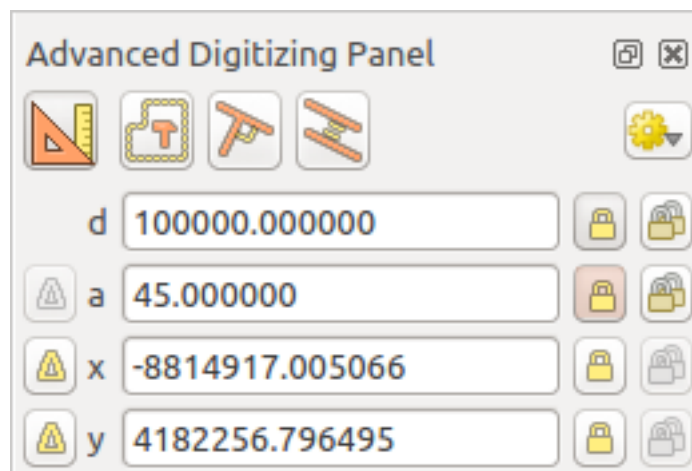



图 14.95: The Advanced Digitizing panel

The *Advanced Digitizing* panel can be open either with a right-click on the toolbar, from *View* ► *Panels* ► menu or pressing `Ctrl+4`. Once the panel is visible, click the  *Enable advanced digitizing tools* button to activate the set of tools.


注解: The tools are not enabled if the map view is in geographic coordinates.

Concepts

The aim of the Advanced Digitizing tool is to lock coordinates, lengths, and angles when moving the mouse during the digitalizing in the map canvas.

You can also create constraints with relative or absolute reference. Relative reference means that the next vertex constraints' values will be relative to the previous vertex or segment.

Snapping Settings

Click the  button to set the Advanced Digitizing Tool snapping settings. You can make the tool snap to common angles. The options are:

- *Do not snap to common angles*
- *Snap to 30° angles*
- *Snap to 45° angles*
- *Snap to 90° angles*

You can also control the snapping to features. The options are:

- *Do not snap to vertices or segments*
- *Snap according to project configuration*
- *Snap to all layers*


Keyboard shortcuts



To speed up the use of Advanced Digitizing Panel, there are a couple of keyboard shortcuts available:

Key	Simple	Ctrl+ or Alt+	Shift+
D	Set distance	Lock distance	
A	Set angle	Lock angle	Toggle relative angle to last segment
X	Set X coordinate	Lock X coordinate	Toggle relative X to last vertex
Y	Set Y coordinate	Lock Y coordinate	Toggle relative Y to last vertex
C	Toggle construction mode		
P	Toggle perpendicular and parallel modes		

Absolute reference digitizing

When drawing a new geometry from scratch, it is very useful to have the possibility to start digitizing vertices at given coordinates.

For example, to add a new feature to a polygonal layer, click the  button. You can choose the X and Y coordinates where you want to start editing the feature, then:

- Click the *x* text box (or use the X keyboard shortcut).
- Type the X coordinate value you want and press `Enter` or click the  button to their right to lock the mouse to the X axis on the map canvas.
- Click the *y* text box (or use the Y keyboard shortcut).
- Type the Y coordinate value you want and press `Enter` or click the  button to their right to lock the mouse to the Y axis on the map canvas.

Two blue dotted lines and a green cross identify the exact coordinates you entered. Start digitizing by clicking on the map canvas; the mouse position is locked at the green cross.

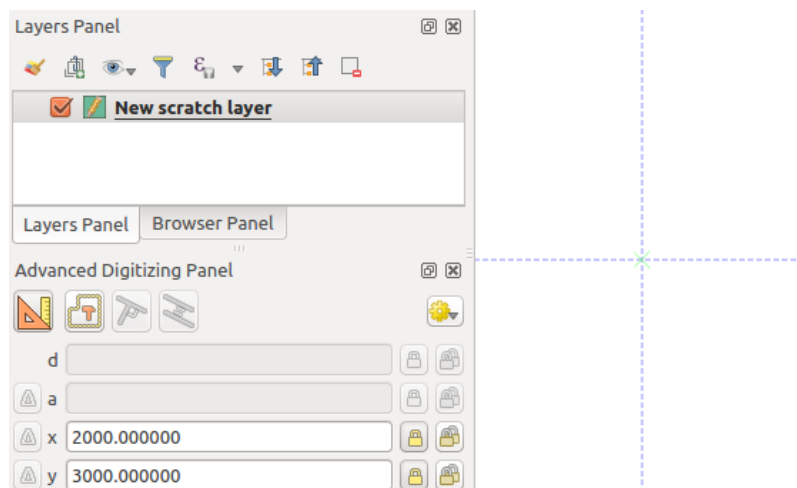



图 14.96: Start drawing at given coordinates

You can continue digitizing by free hand, adding a new pair of coordinates, or you can type the segment's **length** (distance) and **angle**.

If you want to draw a segment of a given length, click the *d* (*distance*) text box (keyboard shortcut D), type the distance value (in map units) and press `Enter` or click the  button on the right to lock the mouse in the map canvas to

the length of the segment. In the map canvas, the clicked point is surrounded by a circle whose radius is the value entered in the distance text box.

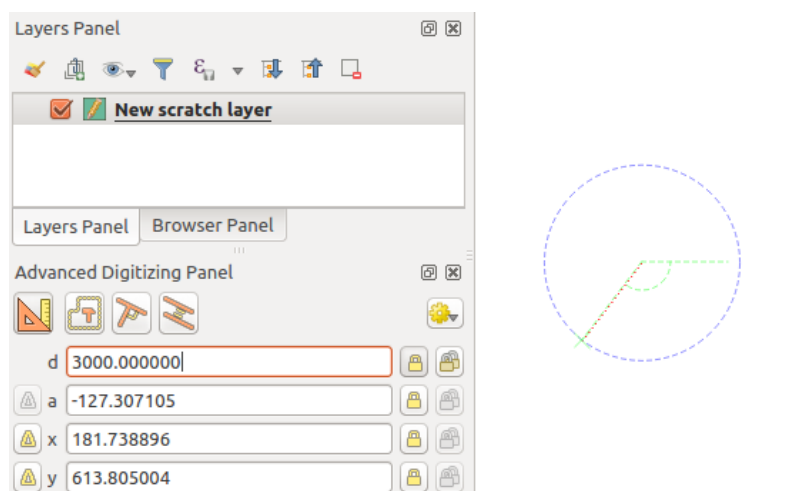



图 14.97: Fixed length segment

Finally, you can also choose the angle of the segment. As described before, click the *a* (angle) text box (keyboard shortcut **A**), type the angle value (in degrees), and press **Enter** or click the  buttons on the right to lock it. In this way the segment will follow the desired angle:

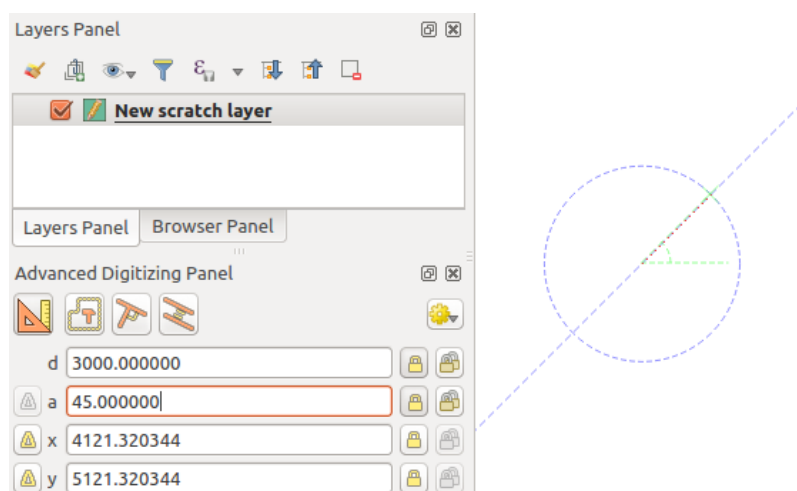




图 14.98: Fixed angle segment


Relative reference digitizing

Instead of using absolute values of angles or coordinates, you can also use values relative to the last digitized vertex or segment.



For angles, you can click the  button on the left of the *a* text box (or press **Shift+A**) to toggle relative angles to the previous segment. With that option on, angles are measured between the last segment and the mouse pointer.


For coordinates, click the  buttons to the left of the *x* or *y* text boxes (or press **Shift+X** or **Shift+Y**) to toggle relative coordinates to the previous vertex. With these options on, coordinates measurement will consider the last vertex to be the X and Y axes origin.

Continuous lock

Both in absolute or relative reference digitizing, angle, distance, X and Y constraints can be locked continuously by clicking the  *Continuous lock* buttons. Using continuous lock allows you to digitize several points or vertexes using the same constraints.

Parallel and perpendicular lines

All the tools described above can be combined with the  Perpendicular and  Parallel tools. These two tools allow drawing segments perfectly perpendicular or parallel to another segment.

To draw a *perpendicular* segment, during the editing click the  Perpendicular icon (keyboard shortcut P) to activate it. Before drawing the perpendicular line, click on the segment of an existing feature that you want to be perpendicular to (the line of the existing feature will be colored in light orange); you should see a blue dotted line where your feature will be snapped:

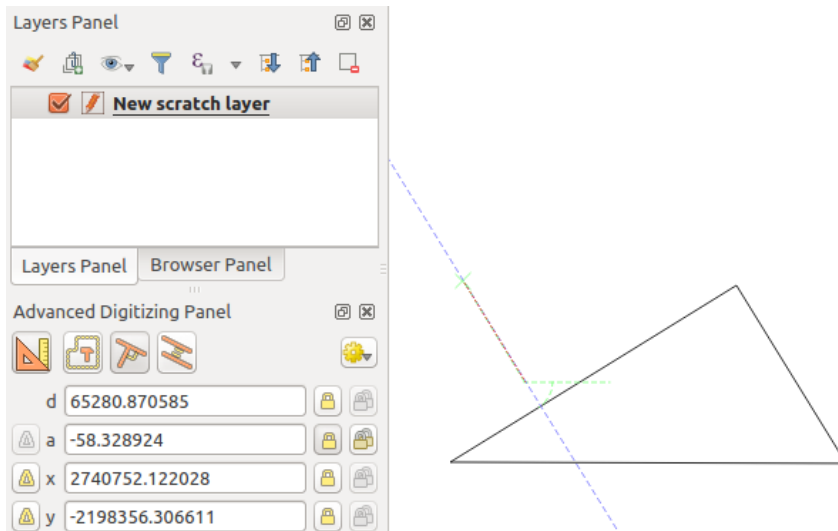



图 14.99: Perpendicular digitizing

To draw a *parallel* feature, the steps are the same: click on the  Parallel icon (keyboard shortcut P twice), click on the segment you want to use as reference and start drawing your feature:

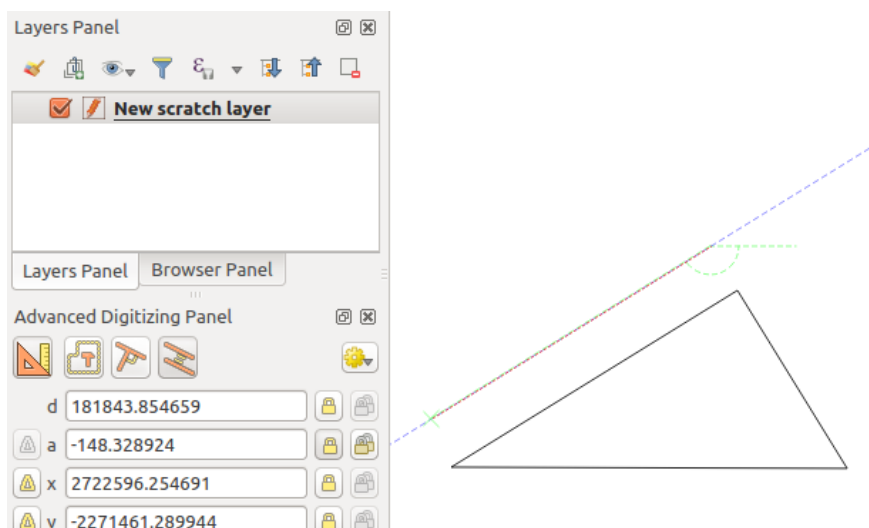





图 14.100: Parallel digitizing

These two tools just find the right angle of the perpendicular and parallel angle and lock this parameter during your editing.

Construction mode

You can enable and disable *construction* mode by clicking on the  *Construction* icon or with the **C** keyboard shortcut. While in construction mode, clicking the map canvas won't add new vertexes, but will capture the clicks' positions so that you can use them as reference points to then lock distance, angle or X and Y relative values.

As an example, the construction mode can be used to draw some point at an exact distance from an existing point.

With an existing point in the map canvas and the snapping mode correctly activated, you can easily draw other points at given distances and angles from it. In addition to the  button, you have to activate also the *construction* mode by clicking the  *Construction* icon or with the **C** keyboard shortcut.

Click next to the point from which you want to calculate the distance and click on the *d* box (**D** shortcut) type the desired distance and press **Enter** to lock the mouse position in the map canvas:

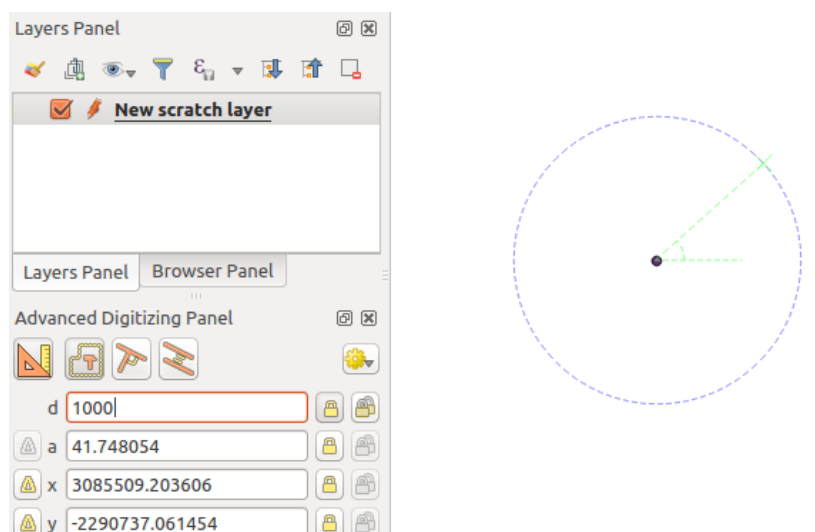



图 14.101: Distance from point

Before adding the new point, press **C** to exit the construction mode. Now, you can click on the map canvas, and the point will be placed at the distance entered.

You can also use the angle constraint to, for example, create another point at the same distance of the original one, but at a particular angle from the newly added point. Click the  Construction icon or with the **C** keyboard shortcut to enter construction mode. Click the recently added point, and then the other one to set a direction segment. Then, click on the *d* text box (**D** shortcut) type the desired distance and press **Enter**. Click the *a* text box (**A** shortcut) type the angle you want and press **Enter**. The mouse position will be locked both in distance and angle.

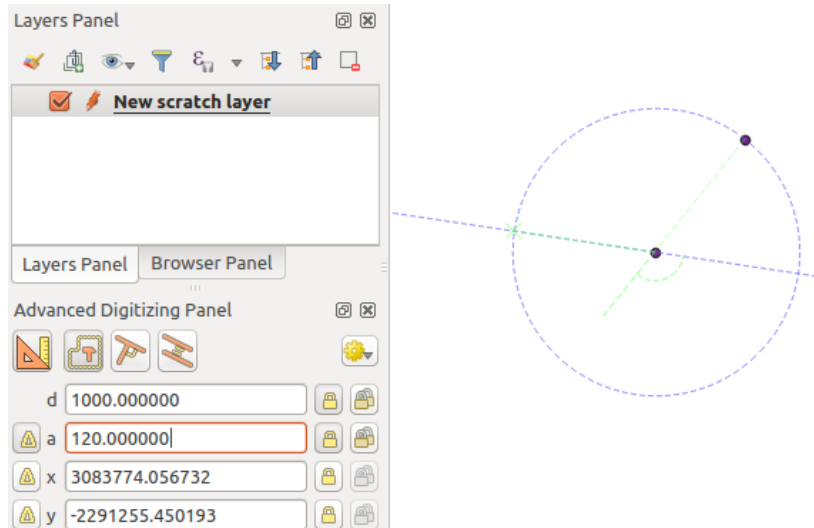


图 14.102: Distance and angle from points

Before adding the new point, press **C** to exit the construction mode. Now, you can click on the map canvas, and the point will be placed at the distance and angle entered. Repeating the process, several points can be added.

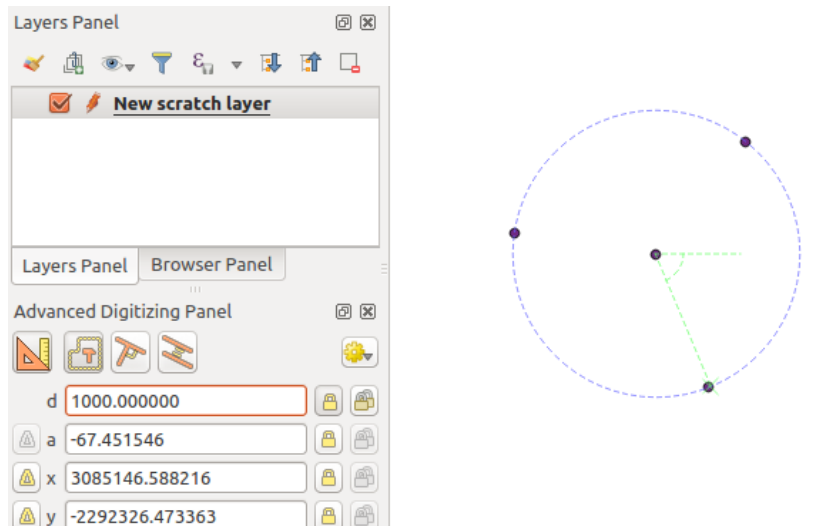



图 14.103: Points at given distance and angle

14.5.7 The Processing in-place layer modifier

The *Processing menu* provides access to a large set of tools to analyze and create new features based on the properties of the input features or their relations with other features (within the same layer or not). While the common behavior is to create new layers as outputs, some algorithms also allow modifications to the input layer. This is a handy way to automate multiple features modification using advanced and complex operations.

To edit features in-place:

1. Select the layer to edit in the *Layers* panel.
2. Select the concerned features. You can skip this step, in which case the modification will apply to the whole layer.
3. Press the  Edit Features In-Place button at the top of the *Processing toolbox*. The list of algorithms is filtered, showing only those compatible with in-place modifications, i.e.:
 - They work at the feature source and not at the layer level.
 - They do not change the layer structure, e.g. adding or removing fields.
 - They do not change the geometry type, e.g. from line to point layer.

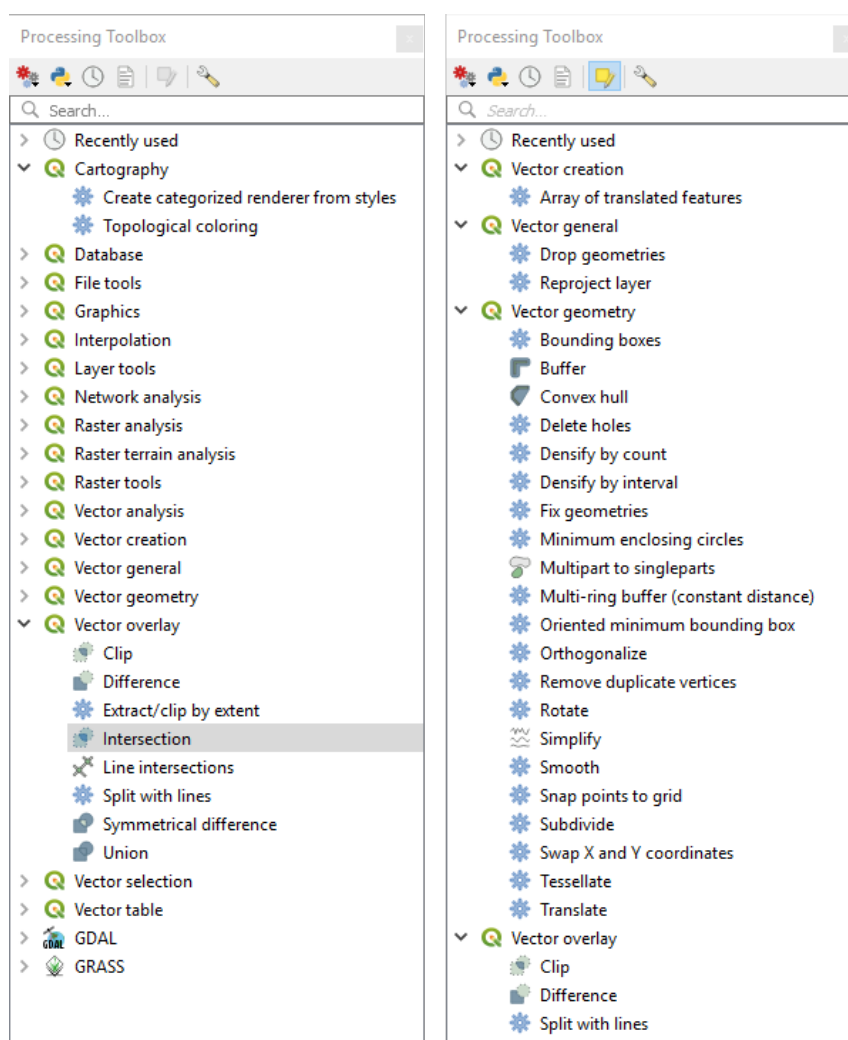





图 14.104: Processing algorithms: all (left) vs polygon in-place editors (right)

4. Find the algorithm you'd like to run and double-click it.

注解: If the algorithm does not need any additional user-set parameters (excluding the usual input and output layer parameters), then the algorithm is run immediately without any dialog popup.

1. If parameters other than the usual input or output layers are needed, the algorithm dialog pops up. Fill in the required information.
2. Click *Modify Selected Features* or *Modify All Features* depending on whether there's an active selection.

Changes are applied to the layer and placed in the edit buffer: the layer is indeed toggled to editing mode with unsaved modification as indicated by the  icon next to the layer name.

5. As usual, press  Save layer edits to commit the changes in the layer. You can also press  Undo to rollback the whole modification.

15.1 Raster Properties Dialog

To view and set the properties for a raster layer, double click on the layer name in the map legend, or right click on the layer name and choose *Properties* from the context menu. This will open the *Raster Layer Properties* dialog.

There are several tabs in the dialog:

-  *Information*
-  *Source*
-  *Symbology*
-  *Transparency*
-  *Histogram*
-  *Rendering*
-  *Pyramids*
-  *Metadata*
-  *Legend*
-  *QGIS Server*


小技巧: Live update rendering

The *Layer Styling Panel* provides you with some of the common features of the Layer properties dialog and is a good modeless widget that you can use to speed up the configuration of the layer styles and view your changes on the map canvas.

注解: Because properties (symbology, label, actions, default values, forms...) of embedded layers (see *Nesting Projects*) are pulled from the original project file, and to avoid changes that may break this behavior, the layer properties


dialog is made unavailable for these layers.


15.1.1 Information Properties

The  *Information* tab is read-only and represents an interesting place to quickly grab summarized information and metadata for the current layer. Provided information are:

- based on the provider of the layer (format of storage, path, data type, extent, width/height, compression, pixel size, statistics on bands, number of columns, rows and no-data values of the raster...);
- picked from the *provided metadata*: access, links, contacts, history... as well as dataset information (CRS, Extent, bands...).

15.1.2 Source Properties

The  *Source* tab displays basic information about the selected raster, including:

- the *Layer name* to display in the *Layers Panel*;
- the *Coordinate Reference System*: Displays the layer's *Coordinate Reference System (CRS)*. You can change the layer's CRS, by selecting a recently used one in the drop-down list or clicking on the  *Select CRS* button (see *Coordinate Reference System Selector*). Use this process only if the layer CRS is a wrong or not specified. If you wish to reproject your data, use a reprojection algorithm from Processing or *Save it as new dataset*.

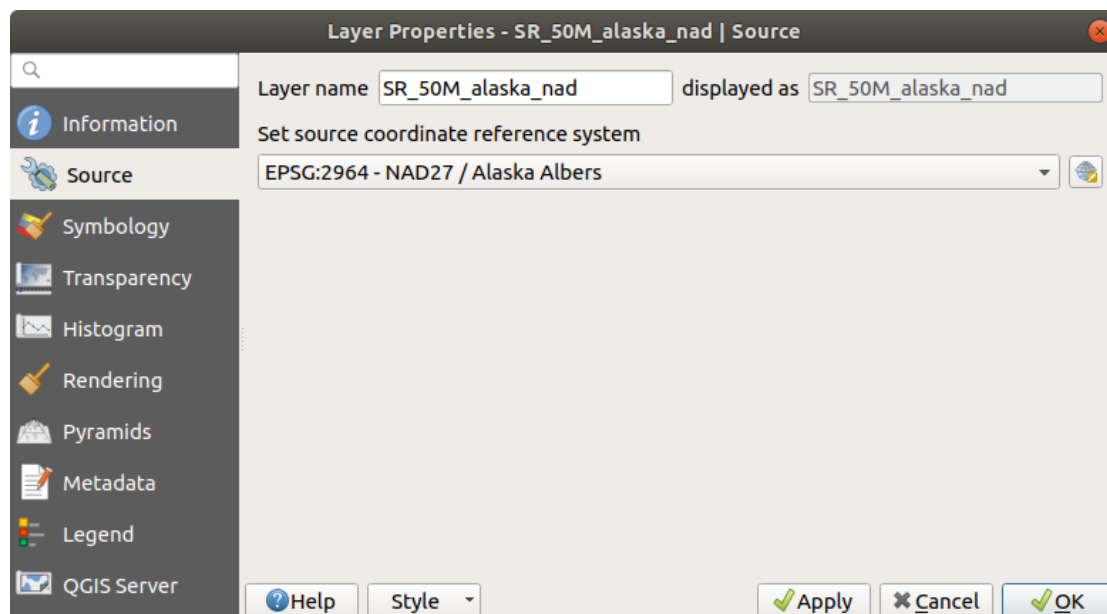


图 15.1: Raster Layer Properties - Source Dialog

15.1.3 Symbology Properties

Band rendering

QGIS offers many different *Render types*. The choice of renderer depends on the data type and the information you'd like to highlight.

1. *Multiband color* - if the file comes with several bands (e.g. a satellite image with several bands).
2. *Paletted/Unique values* - for single band files that come with an indexed palette (e.g. a digital topographic map) or for general use of palettes for rendering raster layers.
3. *Singleband gray* - (one band of) the image will be rendered as gray. QGIS will choose this renderer if the file is neither multiband nor paletted (e.g. a shaded relief map).
4. *Singleband pseudocolor* - this renderer can be used for files with a continuous palette or color map (e.g. an elevation map).
5. *Hillshade* - Creates hillshade from a band.
6. *Contours* - Generates contours on the fly for a source raster band.

Multiband color

With the multiband color renderer, three selected bands from the image will be used as the red, green or blue component of the color image. QGIS automatically fetches *Min* and *Max* values for each band of the raster and scales the coloring accordingly. You can control the value ranges in the *Min/Max Value Settings* section.

A *Contrast enhancement* method can be applied to the values: 'No enhancement', 'Stretch to MinMax', 'Stretch and clip to MinMax' and 'Clip to min max'.

注解: Contrast enhancement

When adding GRASS rasters, the option *Contrast enhancement* will always be set automatically to *stretch to min max*, even if this is set to another value in the QGIS general options.

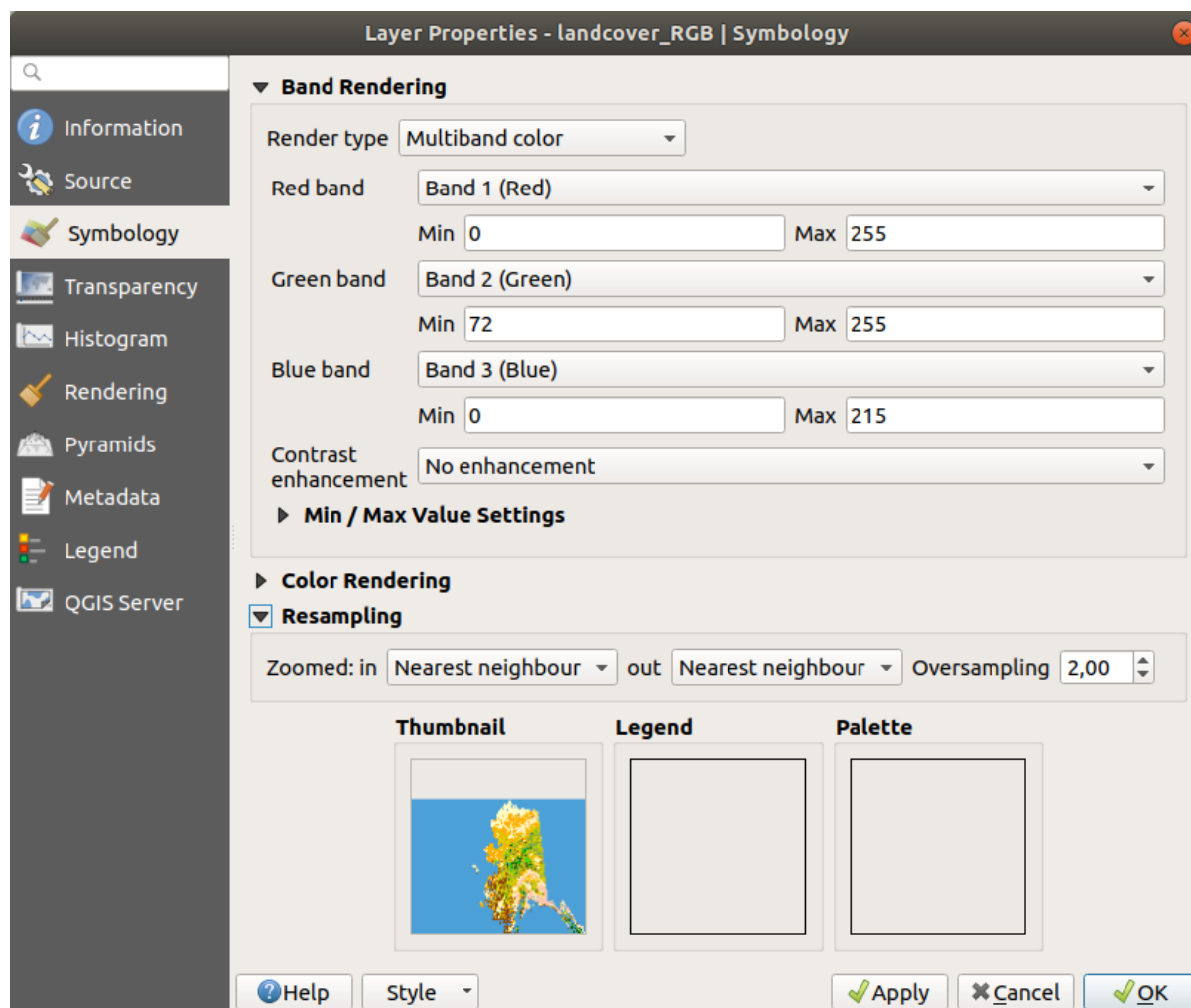


图 15.2: Raster Symbology - Multiband color rendering

小技巧: Viewing a Single Band of a Multiband Raster

If you want to view a single band of a multiband image (for example, Red), you might think you would set the Green and Blue bands to *Not Set*. But the preferred way of doing this is to set the image type to *Singleband gray*, and then select Red as the *Gray band* to use.

Paletted/Unique values

This is the standard render option for singleband files that include a color table, where a certain color is assigned to each pixel value. In that case, the palette is rendered automatically.

It can be used for all kinds of raster bands, assigning a color to each unique raster value.

If you want to change a color, just double-click on the color and the *Select color* dialog appears.

It is also possible to assign labels to the colors. The label will then appear in the legend of the raster layer.

Right-clicking over selected rows in the color table shows a contextual menu to:

- *Change Color...* for the selection
- *Change Opacity...* for the selection
- *Change Label...* for the selection

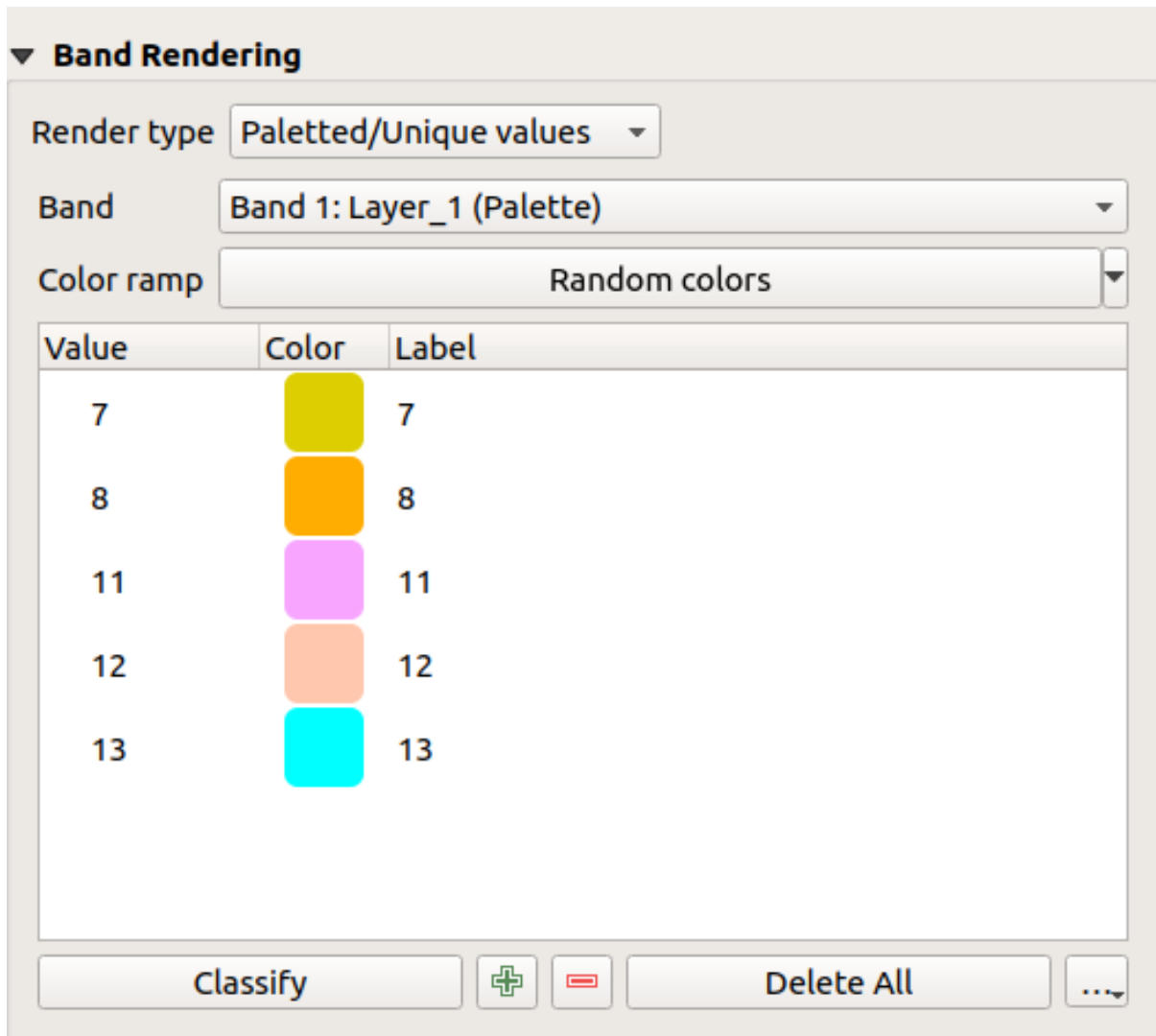


图 15.3: Raster Symbolology - Paletted unique value rendering

The pulldown menu, that opens when clicking the ... (*Advanced options*) button below the color map to the right, offers color map loading (*Load Color Map from File...*) and exporting (*Export Color Map to File...*), and loading of classes (*Load Classes from Layer*).

Singleband gray

This renderer allows you to render a single band layer with a *Color gradient*: 'Black to white' or 'White to black'. You can change the range of values to color (*Min* and *Max*) in the *Min/Max Value Settings*.

A *Contrast enhancement* method can be applied to the values: 'No enhancement', 'Stretch to MinMax', 'Stretch and clip to MinMax' and 'Clip to min max'.

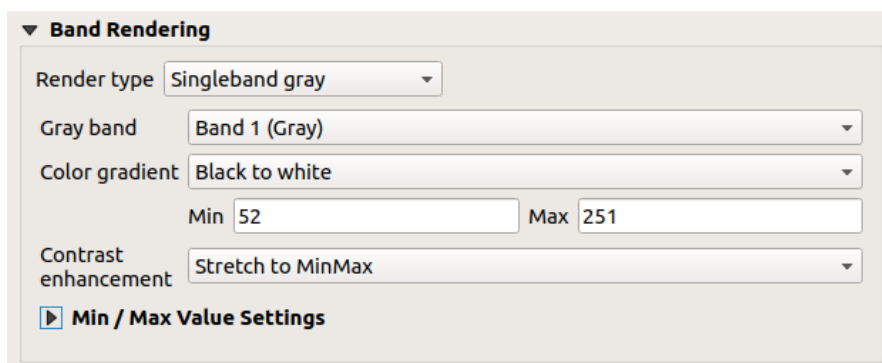


图 15.4: Raster Symbology - Singleband gray rendering

Singleband pseudocolor

This is a render option for single-band files that include a continuous palette. You can also create color maps for a bands of a multiband raster.

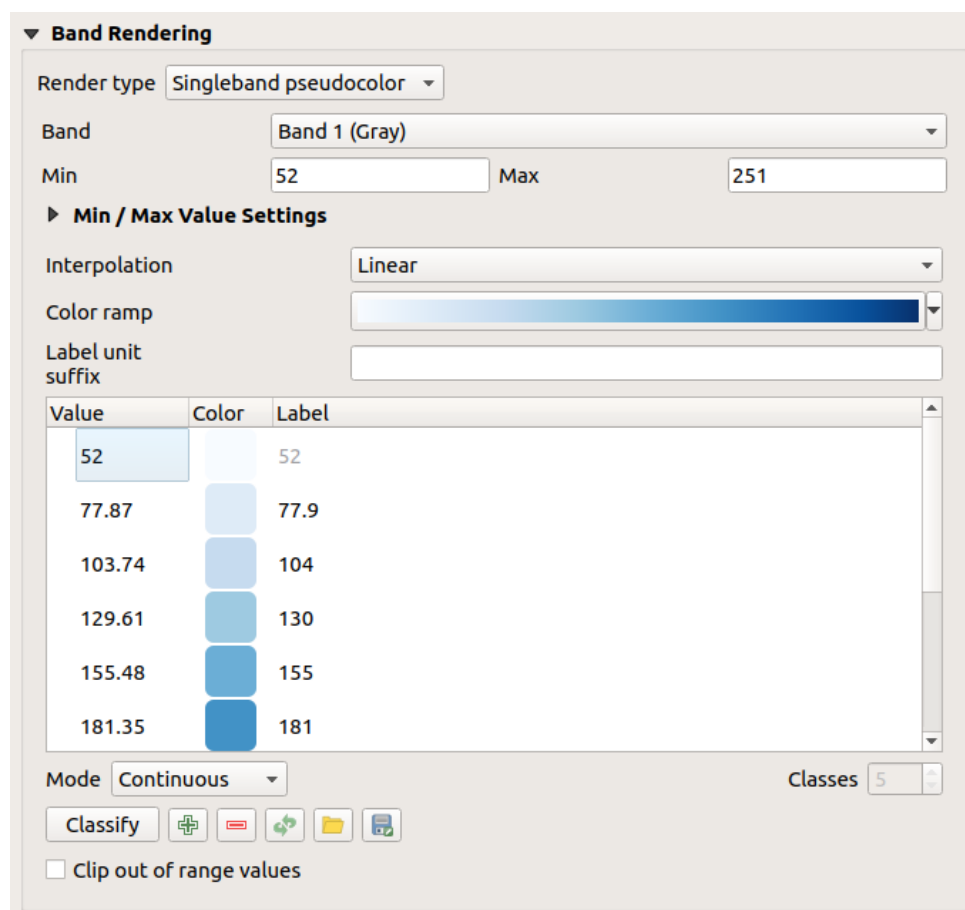


图 15.5: Raster Symbology - Singleband pseudocolor rendering




Using a *Band* of the layer and a *values range*, three types of color *Interpolation* are available:



- Discrete (a <= symbol appears in the header of the *Value* column)
- Linear

- Exact (an = symbol appears in the header of the *Value* column)

The *Color ramp* drop down lists the available color ramps. You can create a new one and edit or save the currently selected one. The name of the color ramp will be saved in the configuration and in the QML file.

The *Label unit suffix* is a label added after the value in the legend.


For classification *Mode*  'Equal interval', you only need to select the *number of classes*  and press the button *Classify*. For *Mode*  'Continuous', QGIS creates classes automatically depending on *Min* and *Max*.

The button  *Add values manually* adds a value to the table. The button  *Remove selected row* deletes a value from the table. Double clicking in the *Value* column lets you insert a specific value. Double clicking in the *Color* column opens the dialog *Change color*, where you can select a color to apply for that value. Further, you can also add labels for each color, but this value won't be displayed when you use the identify feature tool.

Right-clicking over selected rows in the color table shows a contextual menu to:

- *Change Color...* for the selection
- *Change Opacity...* for the selection

You can use the buttons  *Load color map from file* or  *Export color map to file* to load an existing color table or to save the color table for later use.

The  *Clip out of range values* allows QGIS to not render pixel greater than the *Max* value.

Hillshade

Render a band of the raster layer using hillshading.

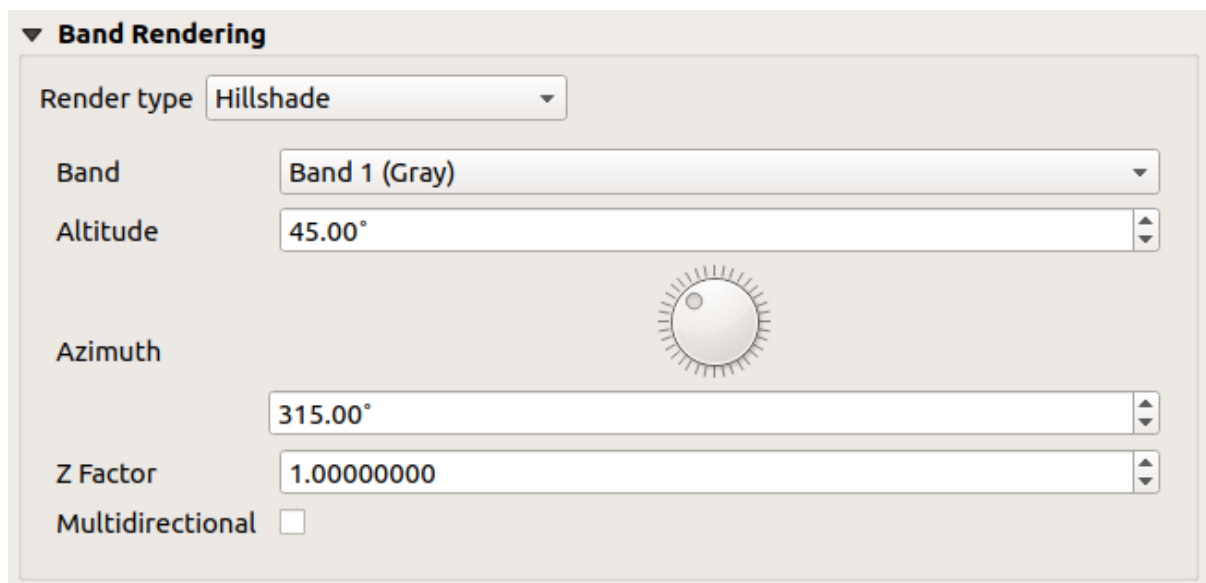


图 15.6: Raster Symbology - Hillshade rendering

Options:

- *Band*: The raster band to use.
- *Altitude*: The elevation angle of the light source (default is 45°).
- *Azimuth*: The azimuth of the light source (default is 315°).
- *Z Factor*: Scaling factor for the values of the raster band (default is 1).

- *Multidirectional*: Specify if multidirectional hillshading is to be used (default is `off`).

Contours

This renderer draws contour lines that are calculated on the fly from the source raster band.

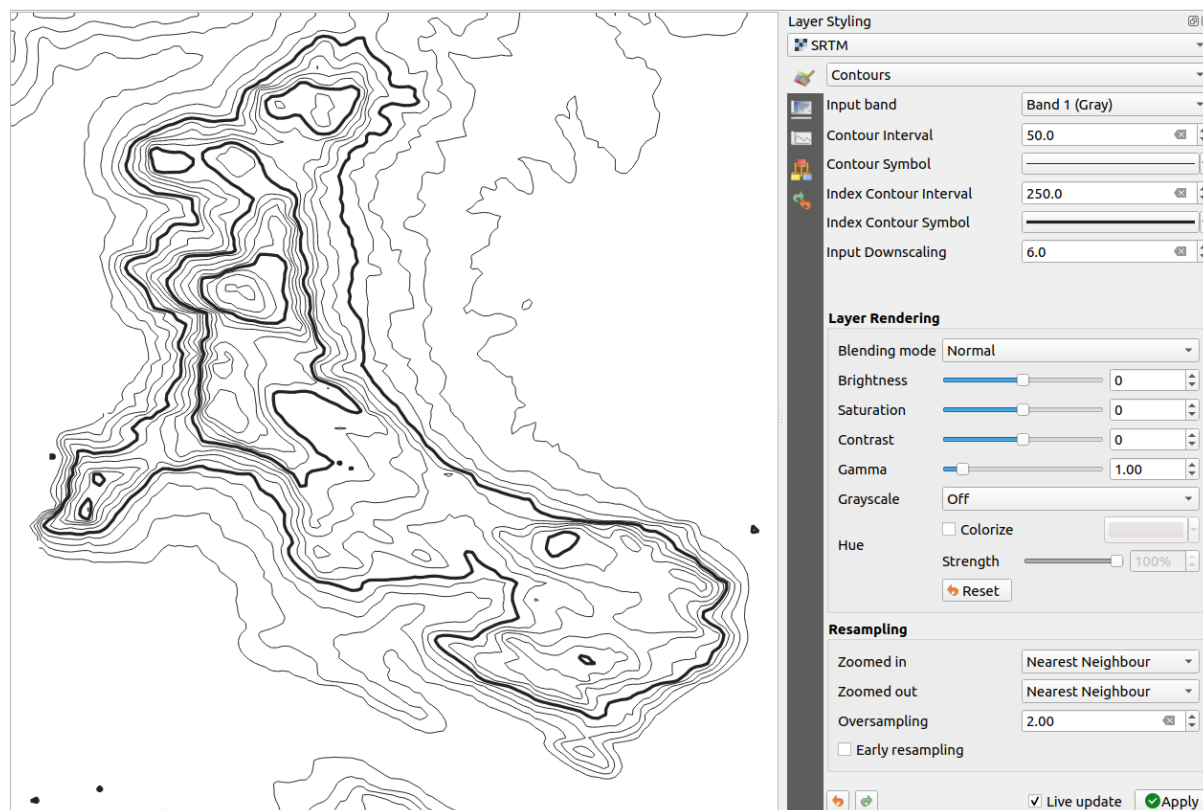


图 15.7: Raster Symbology - Contours rendering

Options:

- *Input band*: the raster band to use.
- *Contour interval*: the distance between two consecutive contour lines
- *Contour symbol*: the *symbol* to apply to the common contour lines.
- *Index contour interval*: the distance between two consecutive **index contours**, that is the lines shown in a distinctive manner for ease of identification, being commonly printed more heavily than other contour lines and generally labeled with a value along its course.
- *Index contour symbol*: the symbol to apply to the index contour lines
- *Input downscaling*: Indicates by how much the renderer will scale down the request to the data provider (default is 4.0).

For example, if you generate contour lines on input raster block with the same size as the output raster block, the generated lines would contain too much detail. This detail can be reduced by the "downscale" factor, requesting lower resolution of the source raster. For a raster block 1000x500 with downscale 10, the renderer will request raster 100x50 from provider. Higher downscale makes contour lines more simplified (at the expense of losing some detail).

Setting the min and max values

By default, QGIS reports the *Min* and *Max* values of the band(s) of the raster. A few very low and/or high values can have a negative impact on the rendering of the raster. The *Min/Max Value Settings* frame helps you control the rendering.

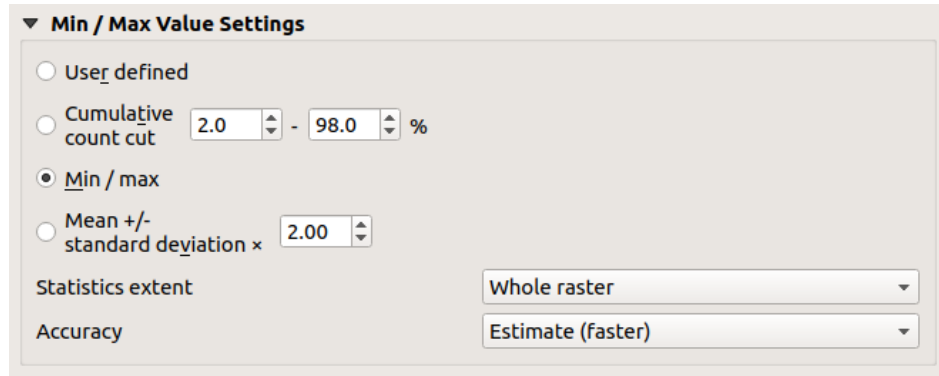


图 15.8: Raster Symbology - Min and Max Value Settings

Available options are:

- *User defined*: The default *Min* and *Max* values of the band(s) can be overridden
- *Cumulative count cut*: Removes outliers. The standard range of values is 2% to 98%, but it can be adapted manually.
- *Min / max*: Uses the whole range of values in the image band.
- *Mean +/- standard deviation x*: Creates a color table that only considers values within the standard deviation or within multiple standard deviations. This is useful when you have one or two cells with abnormally high values in a raster layer that impact the rendering of the raster negatively.

Calculations of the min and max values of the bands are made based on the:

- *Statistics extent*: it can be *Whole raster*, *Current canvas* or *Updated canvas*. *Updated canvas* means that min/max values used for the rendering will change with the canvas extent (dynamic stretching).
- *Accuracy*, which can be either *Estimate (faster)* or *Actual (slower)*.

注解: For some settings, you may need to press the *Apply* button of the layer properties dialog in order to display the actual min and max values in the widgets.

Color rendering

For all kinds of *Band rendering*, the *Color rendering* set.

You can achieve special rendering effects for your raster file(s) by using one of the blending modes (see [Blending Modes](#)).

Further settings can be made by modifying the *Brightness*, *Saturation*, *Gamma* and *Contrast*. You can also use a *Grayscale* option, where you can choose between 'Off', 'By lightness', 'By luminosity' and 'By average'. For one *Hue* in the color table, you can modify the 'Strength'.

Resampling

The *Resampling* option has effect when you zoom in and out of an image. Resampling modes can optimize the appearance of the map. They calculate a new gray value matrix through a geometric transformation.

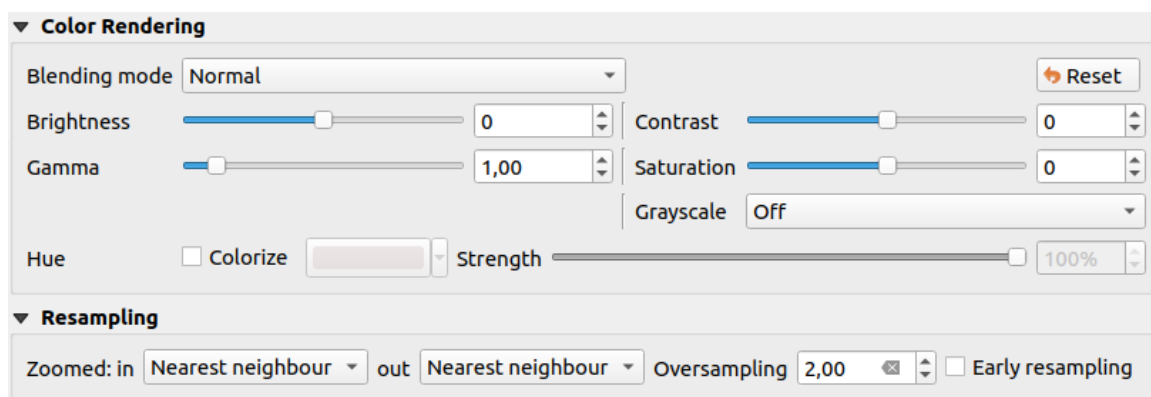




图 15.9: Raster Symbology - Color rendering and Resampling settings

When applying the 'Nearest neighbour' method, the map can get a pixelated structure when zooming in. This appearance can be improved by using the 'Bilinear' or 'Cubic' method, which cause sharp edges to be blurred. The effect is a smoother image. This method can be applied to for instance digital topographic raster maps.

At the bottom of the *Symbology* tab, you can see a thumbnail of the layer, its legend symbol, and the palette.

15.1.4 Transparency Properties

 QGIS has the ability to set the transparency level of a raster layer. Use the transparency slider  to set to what extent the underlying layers (if any) should be visible through the current raster layer. This is very useful if you overlay raster layers (e.g., a shaded relief map overlaid by a classified raster map). This will make the look of the map more three dimensional.

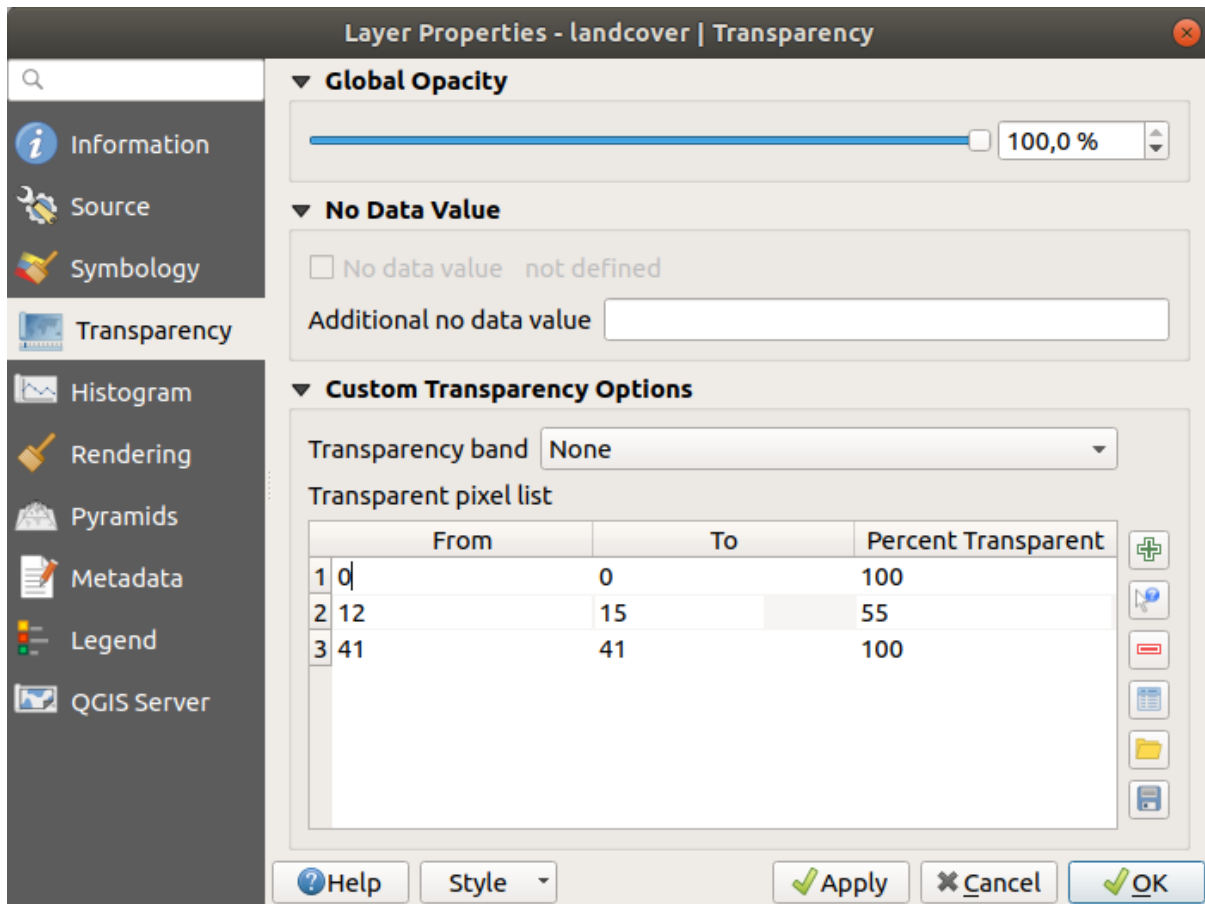






图 15.10: Raster Transparency



Additionally, you can enter a raster value that should be treated as an *Additional no data value*.


An even more flexible way to customize the transparency is available in the *Custom transparency options* section:


- Use *Transparency band* to apply transparency for an entire band.
- Provide a list of pixels to make transparent with corresponding levels of transparency:
 1. Click the  **Add values manually** button. A new row will appear in the pixel list.
 2. Enter the **Red**, **Green** and **Blue** values of the pixel and adjust the **Percent Transparent** to apply.
 3. Alternatively, you can fetch the pixel values directly from the raster using the  **Add values from display** button. Then enter the transparency value.
 4. Repeat the steps to adjust more values with custom transparency.
 5. Press the *Apply* button and have a look at the map.

As you can see, it is quite easy to set custom transparency, but it can be quite a lot of work. Therefore, you can use the button  **Export to file** to save your transparency list to a file. The button  **Import from file** loads your transparency settings and applies them to the current raster layer.

15.1.5 Histogram Properties

The  *Histogram* tab allows you to view the distribution of the values in your raster. The histogram is generated when you press the *Compute Histogram* button. All existing bands will be displayed together. You can save the histogram as an image with the  button.

At the bottom of the histogram, you can select a raster band in the drop-down menu and *Set min/max style* for it. The  *Prefs/Actions* drop-down menu gives you advanced options to customize the histogram:

- With the *Visibility* option, you can display histograms for individual bands. You will need to select the option  *Show selected band*.
- The *Min/max options* allow you to 'Always show min/max markers', to 'Zoom to min/max' and to 'Update style to min/max'.
- The *Actions* option allows you to 'Reset' or 'Recompute histogram' after you have changed the min or max values of the band(s).

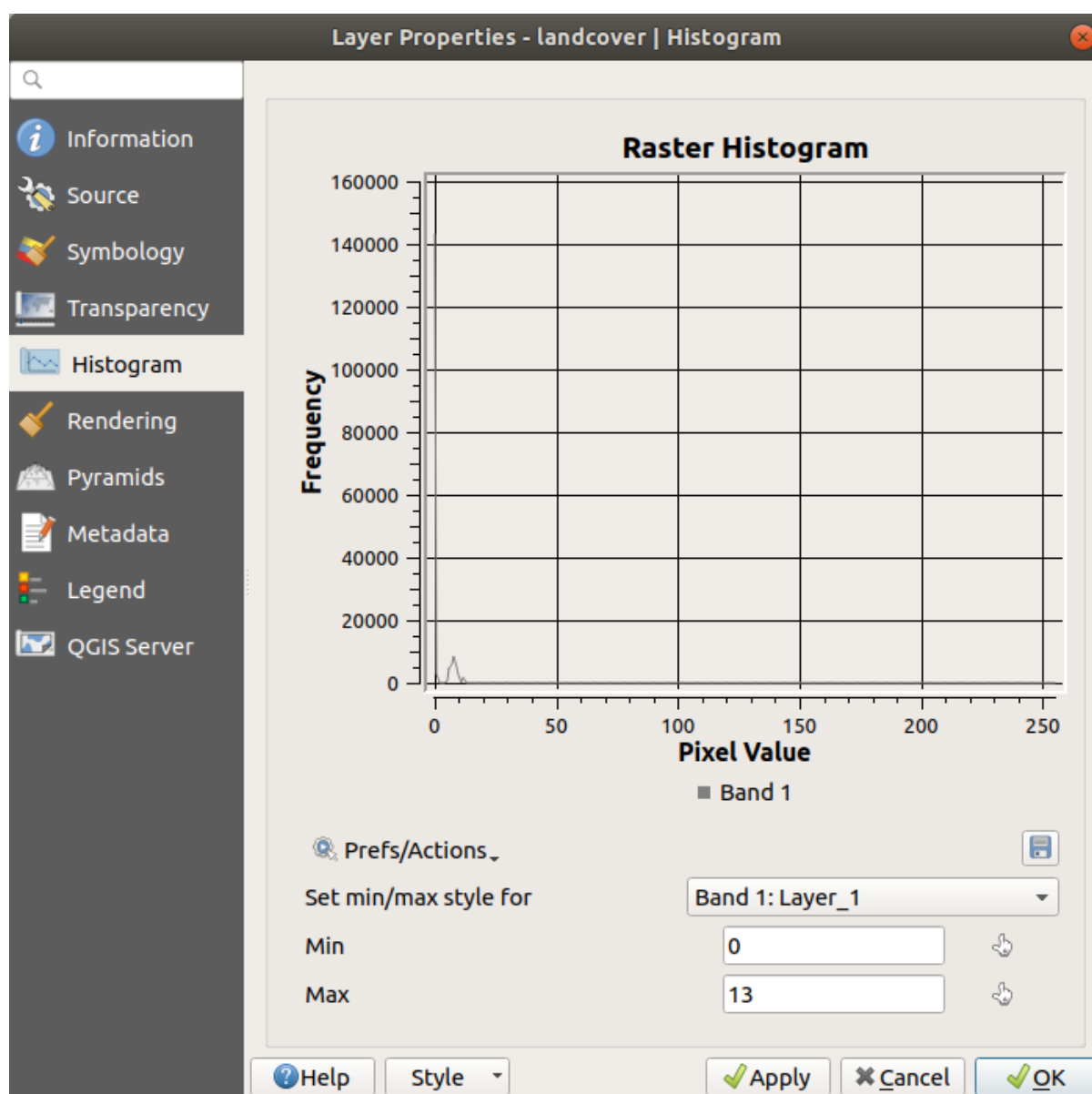




图 15.11: Raster Histogram

15.1.6 Rendering Properties

In the  *Rendering* tab, it's possible to:

- set *Scale dependent visibility* for the layer: You can set the *Maximum (inclusive)* and *Minimum (exclusive)* scale, defining a range of scales in which the layer will be visible. It will be hidden outside this range. The  *Set to current canvas scale* button helps you use the current map canvas scale as a boundary. See *Scale Dependent Rendering* for more information.
- *Refresh layer at interval (seconds)*: set a timer to automatically refresh individual layers. Canvas updates are deferred in order to avoid refreshing multiple times if more than one layer has an auto update interval set.

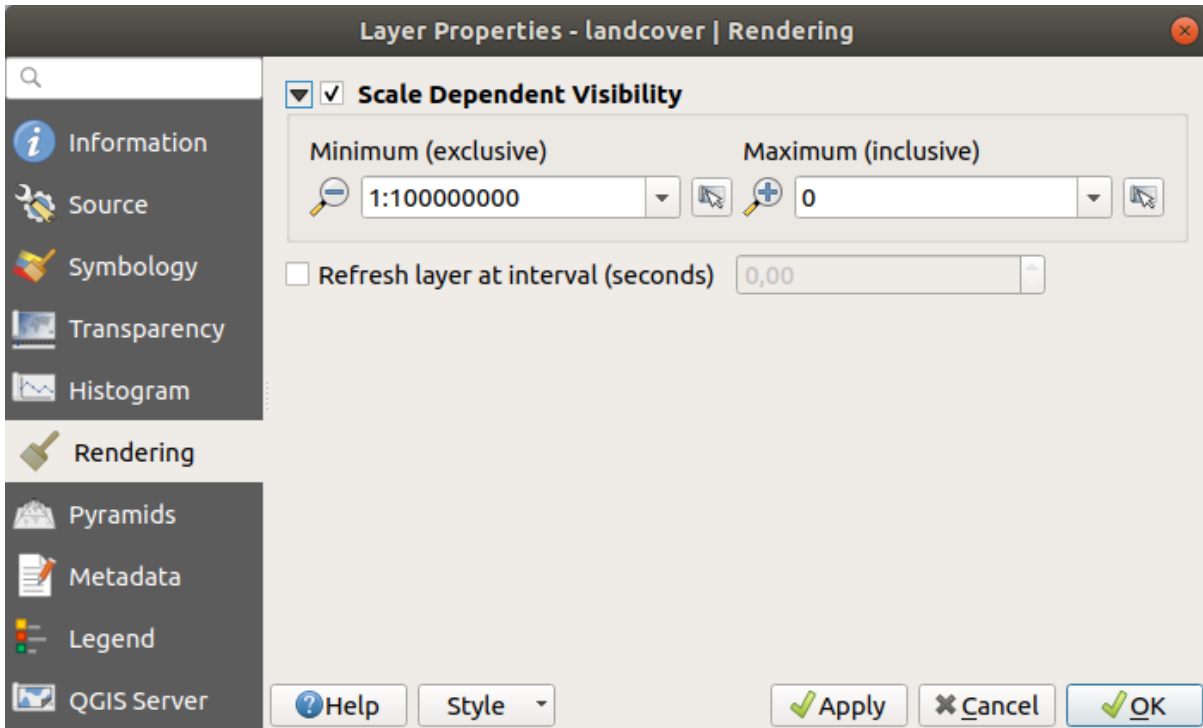


图 15.12: Raster Rendering

15.1.7 Pyramids Properties

High resolution raster layers can slow navigation in QGIS. By creating lower resolution copies of the data (pyramids), performance can be considerably improved, as QGIS selects the most suitable resolution to use depending on the zoom level.

You must have write access in the directory where the original data is stored to build pyramids.

From the *Resolutions* list, select resolutions at which you want to create pyramid levels by clicking on them.

If you choose **Internal (if possible)** from the *Overview format* drop-down menu, QGIS tries to build pyramids internally.

注解: Please note that building pyramids may alter the original data file, and once created they cannot be removed. If you wish to preserve a 'non-pyramided' version of your raster, make a backup copy prior to pyramid building.

If you choose **External** and **External (Erdas Imagine)** the pyramids will be created in a file next to the original raster with the same name and a `.ovr` extension.

Several *Resampling methods* can be used for pyramid calculation:

- 最邻近采样分析
- 平均
- Gauss
- 立方体
- Cubic Spline
- Laczos
- Mode
- None

Finally, click *Build Pyramids* to start the process.

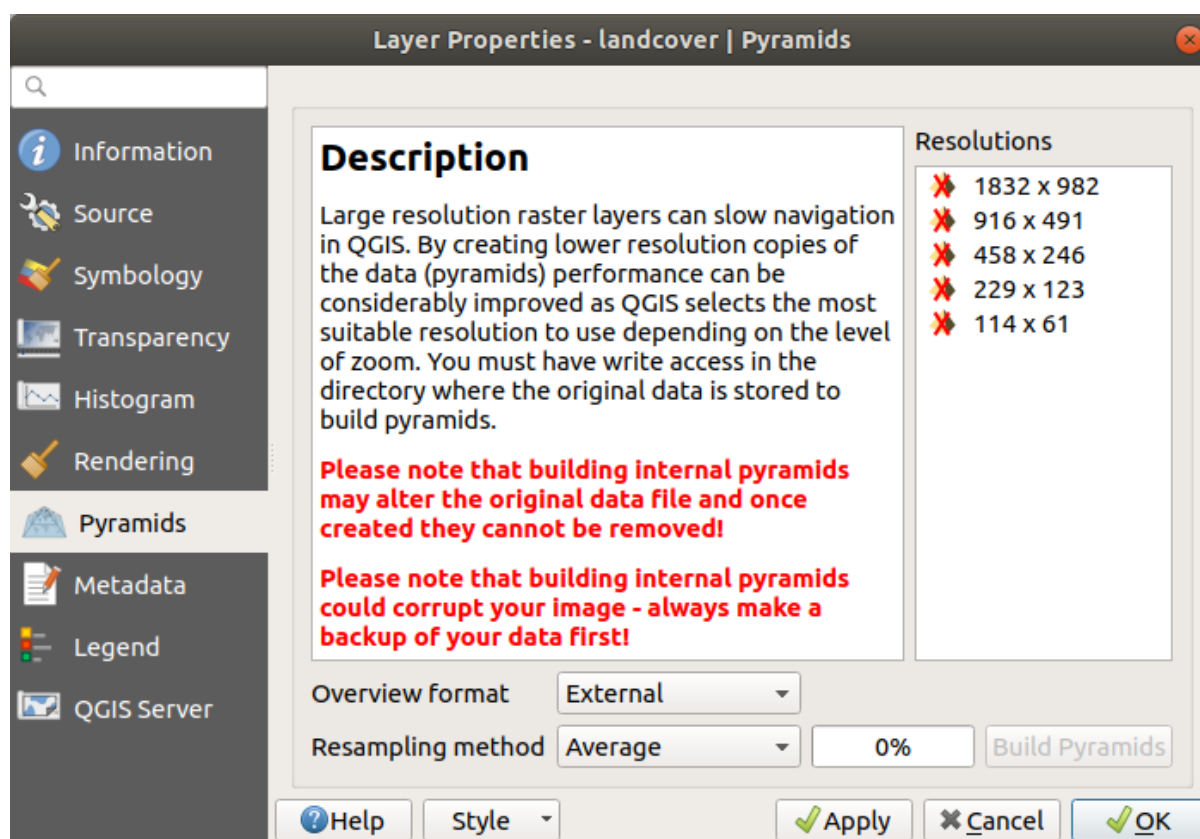



图 15.13: Raster Pyramids

15.1.8 Metadata Properties

The  *Metadata* tab provides you with options to create and edit a metadata report on your layer. See *vector layer metadata properties* for more information.

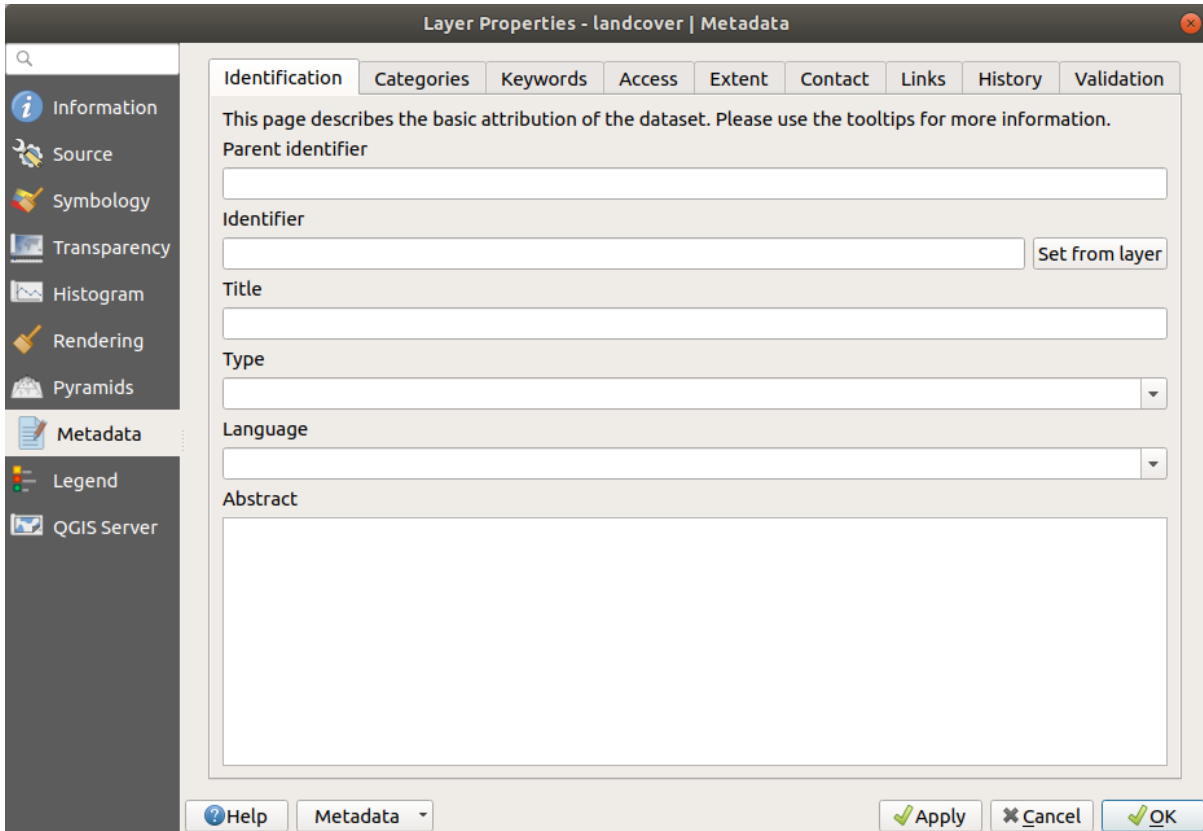



图 15.14: Raster Metadata

15.1.9 Legend Properties

The  *Legend* tab provides you with a list of widgets you can embed within the layer tree in the Layers panel. The idea is to have a way to quickly access some actions that are often used with the layer (setup transparency, filtering, selection, style or other stuff...).

By default, QGIS provides a transparency widget but this can be extended by plugins that register their own widgets and assign custom actions to layers they manage.

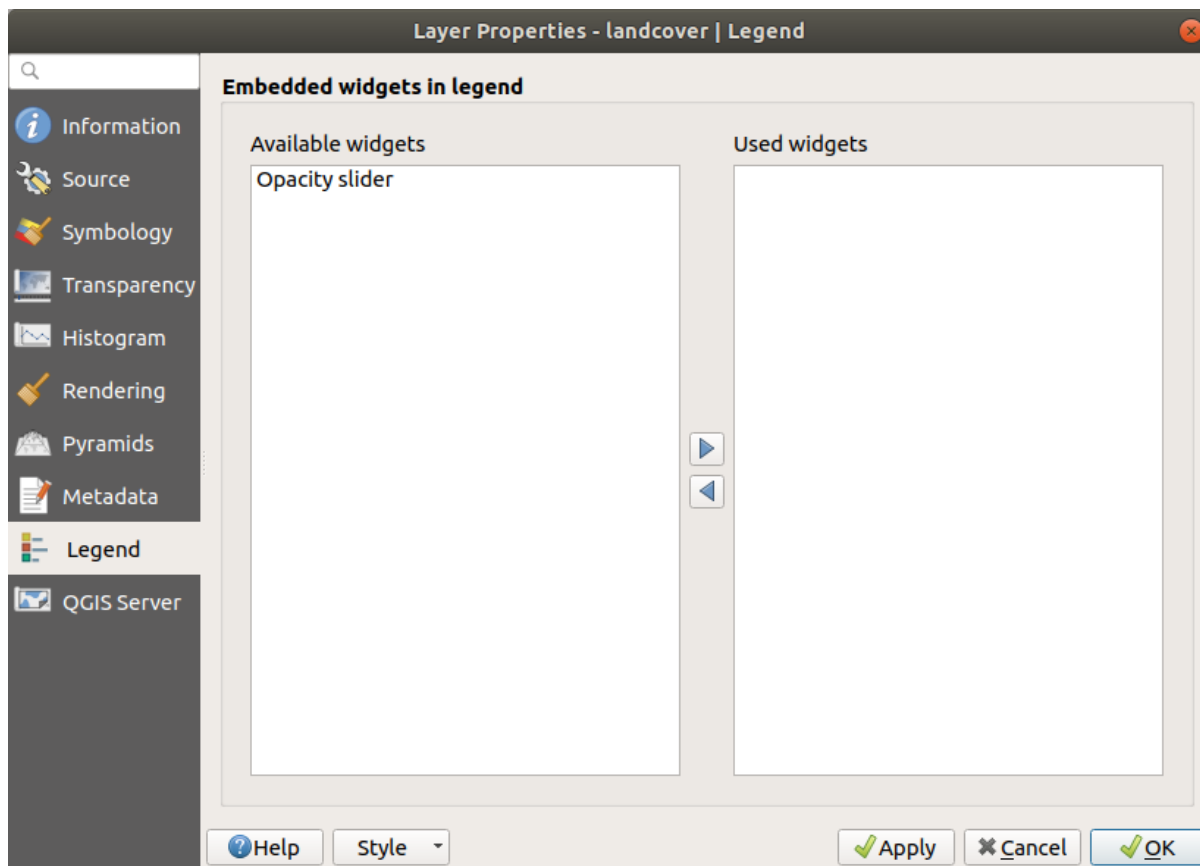


图 15.15: Raster Legend

15.1.10 QGIS Server Properties

From the  *QGIS Server* tab, information can be provided for *Description*, *Attribution*, *MetadataUrl* and *LegendUrl*.

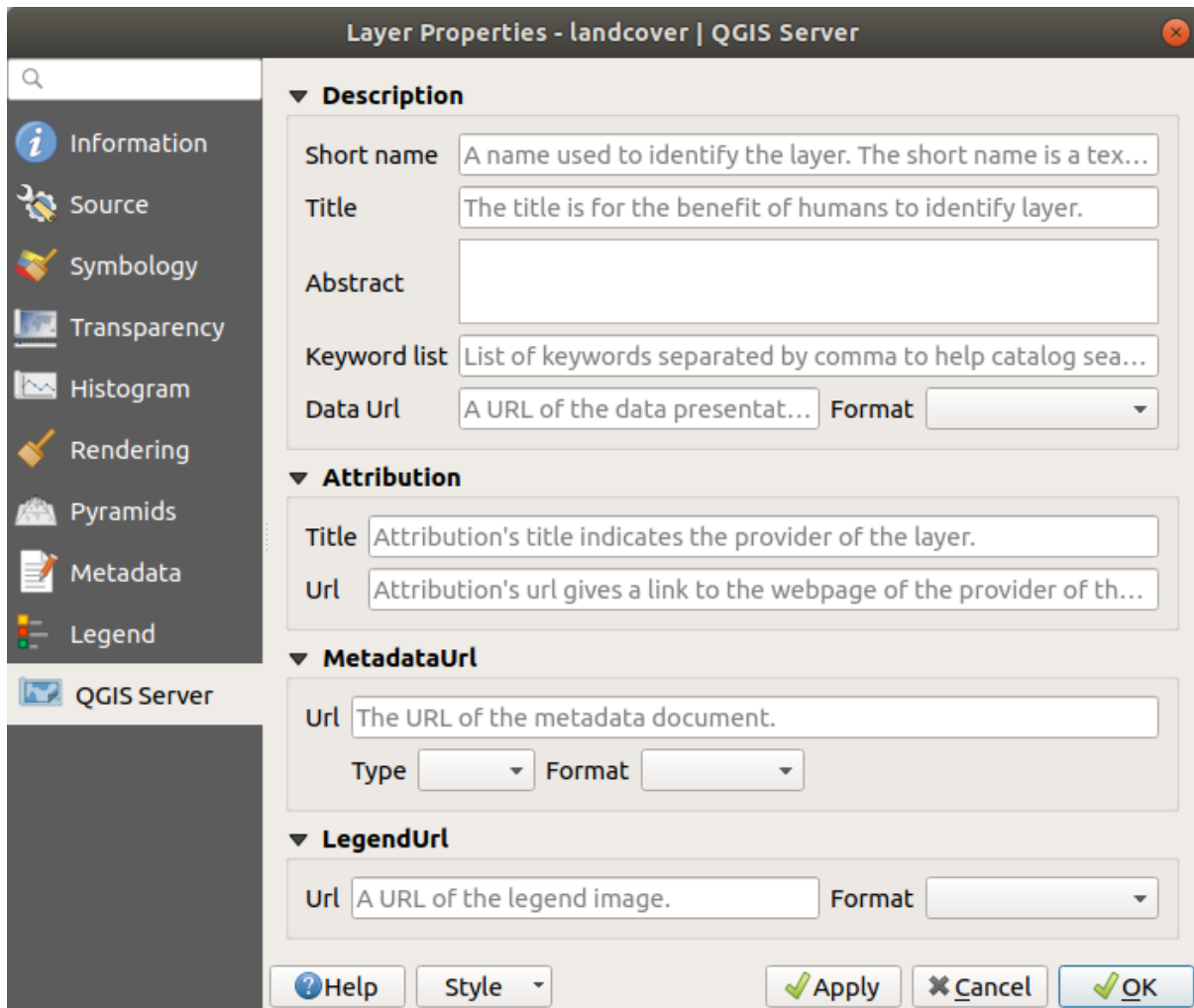


图 15.16: QGIS Server in Raster Properties

15.2 Raster Analysis

15.2.1 Raster Calculator

The *Raster Calculator* in the *Raster* menu allows you to perform calculations on the basis of existing raster pixel values (see 图 15.17). The results are written to a new raster layer in a GDAL-supported format.

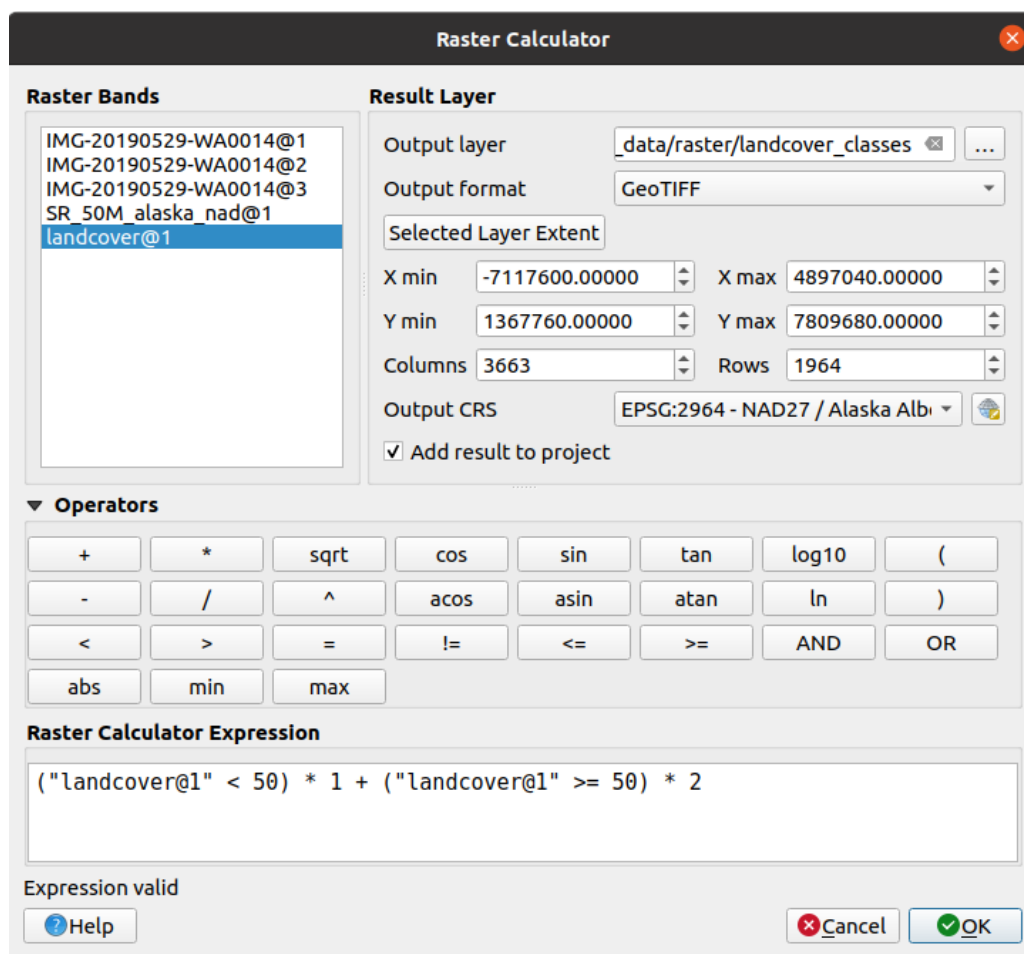


图 15.17: Raster Calculator

The **Raster bands** list contains all loaded raster layers that can be used. To add a raster to the raster calculator expression field, double click its name in the Fields list. You can then use the operators to construct calculation expressions, or you can just type them into the box.

In the **Result layer** section, you will need to define an output layer. You can then define the extent of the calculation area based on an input raster layer, or based on X,Y coordinates and on columns and rows, to set the resolution of the output layer. If the input layer has a different resolution, the values will be resampled with the nearest neighbor algorithm.

The **Operators** section contains all available operators. To add an operator to the raster calculator expression box, click the appropriate button. Mathematical calculations (+, -, *, ...) and trigonometric functions (sin, cos, tan, ...) are available. Conditional expressions (=, !=, <, >=, ...) return either 0 for false or 1 for true, and therefore can be used with other operators and functions.

With the *Add result to project* checkbox, the result layer will automatically be added to the legend area and can be visualized.

提示: See also the *Raster calculator* algorithm.

Examples

Convert elevation values from meters to feet

Creating an elevation raster in feet from a raster in meters, you need to use the conversion factor for meters to feet: 3.28. The expression is:

```
"elevation@1" * 3.28
```

Using a mask

If you want to mask out parts of a raster -- say, for instance, because you are only interested in elevations above 0 meters -- you can use the following expression to create a mask and apply the result to a raster in one step.

```
("elevation@1" >= 0) * "elevation@1"
```

In other words, for every cell greater than or equal to 0 the conditional expression evaluates to 1, which keeps the original value by multiplying it by 1. Otherwise the conditional expression evaluates to 0, which sets the raster value to 0. This creates the mask on the fly.

If you want to classify a raster -- say, for instance into two elevation classes, you can use the following expression to create a raster with two values 1 and 2 in one step.

```
("elevation@1" < 50) * 1 + ("elevation@1" >= 50) * 2
```


In other words, for every cell less than 50 set its value to 1. For every cell greater than or equal 50 set its value to 2.

15.2.2 Raster Alignment

This tool is able to take several rasters as input and to align them perfectly, that means:

- reproject to the same CRS,
- resample to the same cell size and offset in the grid,
- clip to a region of interest,
- rescale values when required.

All rasters will be saved in another files.

First, open the tools from *Raster ► Align Raster...* and click on the  Add new raster button to choose one existing raster in QGIS. Select an output file to save the raster after the alignment, the resampling method and if the tools need to *Rescale values according to the cell size*. The resampling method can be (see [图 15.18](#)):

- **Nearest Neighbor**
- **Bilinear (2x2 kernel)**
- **Cubic (4x4 kernel)**: Cubic Convolution Approximation
- **Cubic B-Spline (4x4 kernel)**: Cubic B-Spline Approximation
- **Lanczos (6x6 kernel)**: Lanczos windowed sinc interpolation
- **Average**: computes the average of all non-NODATA contributing pixels
- **Mode**: selects the value which appears most often of all the sampled points
- **Maximum, Minimum, Mediane, First Quartile (Q1) or Third Quartile (Q3)** of all non-NODATA contributing pixels

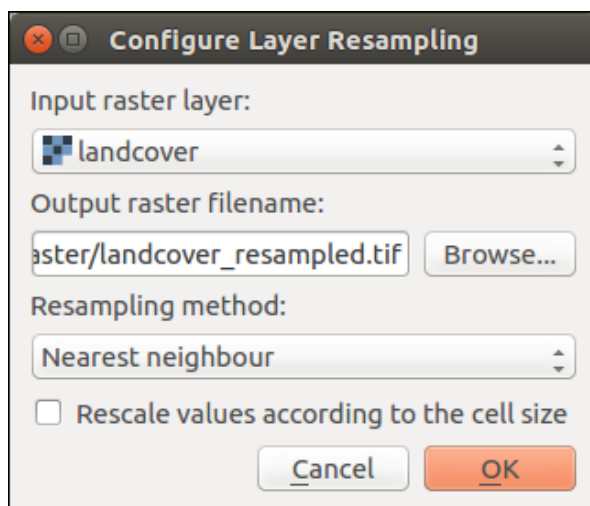


图 15.18: Select Raster Resampling Options

In the main *Align raster* dialog, you can still  Edit file settings or  Remove an existing file from the list of raster layers. You can also choose one or more other options (see [图 15.19](#)):

- Select the *Reference Layer*,
- Transform into a new *CRS*,
- Setup a different *Cell size*,
- Setup a different *Grid Offset*,
- *Clip to Extent*: it can be user-defined, bound to a layer or to the map canvas
- *Output Size*,
- *Add aligned raster to the map canvas*.

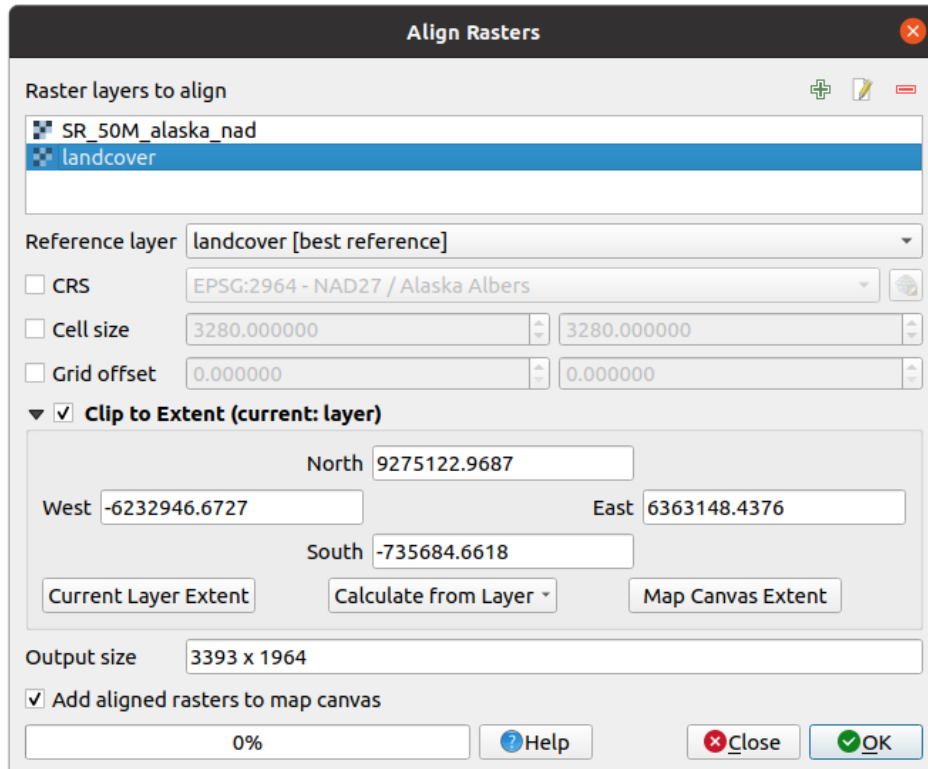



图 15.19: Raster Alignment

15.3 Georeferencer

The  Georeferencer is a tool for generating world files for rasters. It allows you to reference rasters to geographic or projected coordinate systems by creating a new GeoTiff or by adding a world file to the existing image. The basic approach to georeferencing a raster is to locate points on the raster for which you can accurately determine coordinates.

Features




















Icon	目标	Icon	目标
	Open raster		Start georeferencing
	Generate GDAL Script		Load GCP Points
	Save GCP Points As		Transformation settings
	Add Point		Delete Point
	Move GCP Point		Pan
	Zoom In		Zoom Out
	Zoom To Layer		Zoom Last
	Zoom Next		Link Georeferencer to QGIS
	Link QGIS to Georeferencer		Full histogram stretch
	Local histogram stretch		

Table Georeferencer: Georeferencer Tools

15.3.1 Usual procedure

As X and Y coordinates (DMS (dd mm ss.ss), DD (dd.dd) or projected coordinates (mmmm.mm)), which correspond with the selected point on the image, two alternative procedures can be used:

- The raster itself sometimes provides crosses with coordinates "written" on the image. In this case, you can enter the coordinates manually.
- Using already georeferenced layers. This can be either vector or raster data that contain the same objects/features that you have on the image that you want to georeference and with the projection that you want for your image. In this case, you can enter the coordinates by clicking on the reference dataset loaded in the QGIS map canvas.

The usual procedure for georeferencing an image involves selecting multiple points on the raster, specifying their coordinates, and choosing a relevant transformation type. Based on the input parameters and data, the Georeferencer will compute the world file parameters. The more coordinates you provide, the better the result will be.

The first step is to start QGIS and click on *Raster* ►  *Georeferencer*, which appears in the QGIS menu bar. The Georeferencer dialog appears as shown in  15.20.

For this example, we are using a topo sheet of South Dakota from SDGS. It can later be visualized together with the data from the GRASS *spearfish60* location. You can download the topo sheet here: https://grass.osgeo.org/sampledata/spearfish_toposheet.tar.gz.

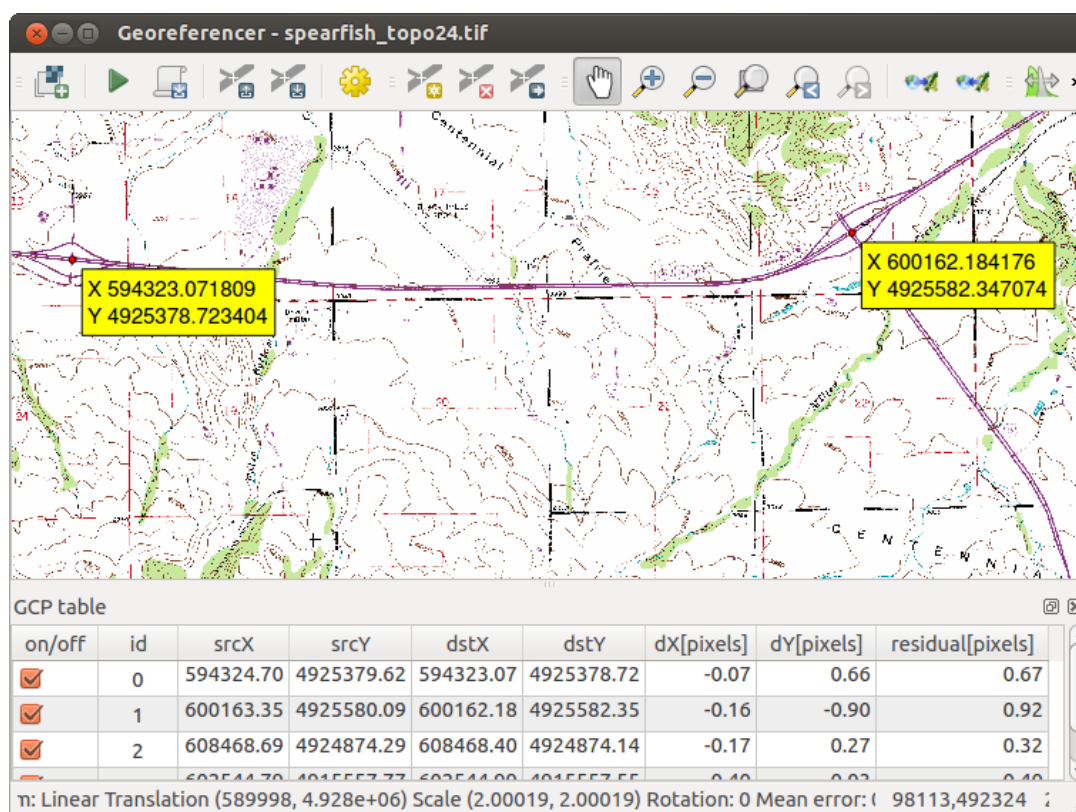






图 15.20: Georeferencer Dialog

Entering ground control points (GCPs)

1. To start georeferencing an unreferenced raster, we must load it using the  button. The raster will show up in the main working area of the dialog. Once the raster is loaded, we can start to enter reference points.
2. Using the  Add Point button, add points to the main working area and enter their coordinates (see Figure [图 15.21](#)). For this procedure you have three options:
 - Click on a point in the raster image and enter the X and Y coordinates manually.
 - Click on a point in the raster image and choose the  From map canvas button to add the X and Y coordinates with the help of a georeferenced map already loaded in the QGIS map canvas.
 - With the  button, you can move the GCPs in both windows, if they are at the wrong place.
3. Continue entering points. You should have at least four points, and the more coordinates you can provide, the better the result will be. There are additional tools for zooming and panning the working area in order to locate a relevant set of GCP points.

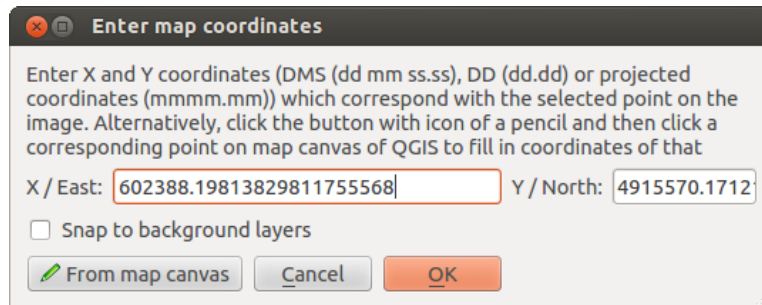




图 15.21: Add points to the raster image

The points that are added to the map will be stored in a separate text file (`[filename].points`) usually together with the raster image. This allows us to reopen the Georeferencer at a later date and add new points or delete existing ones to optimize the result. The points file contains values of the form: `mapX, mapY, pixelX, pixelY`. You can use the  Load GCP points and  Save GCP points as buttons to manage the files.

Defining the transformation settings

After you have added your GCPs to the raster image, you need to define the transformation settings for the georeferencing process.

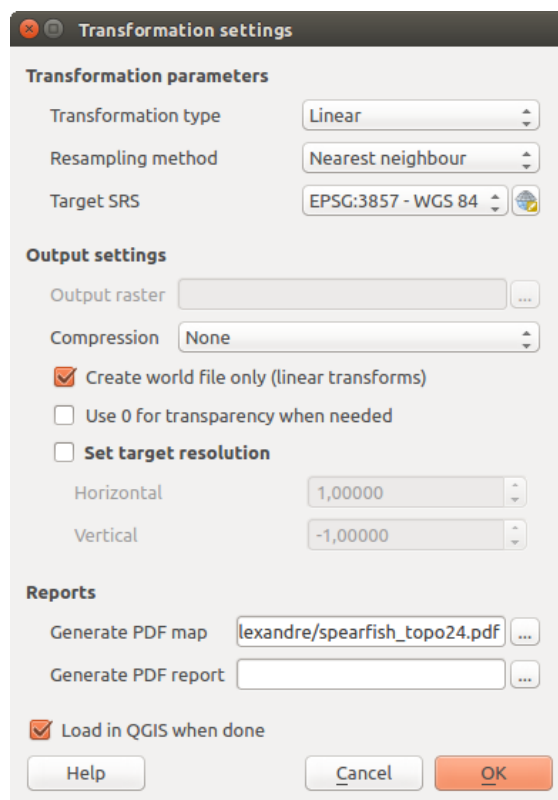


图 15.22: Defining the georeferencer transformation settings

Available Transformation algorithms

A number of transformation algorithms are available, dependent on the type and quality of input data, the nature and amount of geometric distortion that you are willing to introduce to the final result, and the number of ground control points (GCPs).

Currently, the following *Transformation types* are available:

- The **Linear** algorithm is used to create a world file and is different from the other algorithms, as it does not actually transform the raster pixels. It allows positioning (translating) the image and uniform scaling, but no rotation or other transformations. It is the most suitable if your image is a good quality raster map, in a known CRS, but is just missing georeferencing information. At least 2 GCPs are needed.
- The **Helmert** transformation also allows rotation. It is particularly useful if your raster is a good quality local map or orthorectified aerial image, but not aligned with the grid bearing in your CRS. At least 2 GCPs are needed.
- The **Polynomial 1** algorithm allows a more general affine transformation, in particular also a uniform shear. Straight lines remain straight (i.e., collinear points stay collinear) and parallel lines remain parallel. This is particularly useful for georeferencing data cartograms, which may have been plotted (or data collected) with different ground pixel sizes in different directions. At least 3 GCP's are required.
- The **Polynomial** algorithms 2-3 use more general 2nd or 3rd degree polynomials instead of just affine transformation. This allows them to account for curvature or other systematic warping of the image, for instance photographed maps with curving edges. At least 6 (respectively 10) GCP's are required. Angles and local scale are not preserved or treated uniformly across the image. In particular, straight lines may become curved, and there may be significant distortion introduced at the edges or far from any GCPs arising from extrapolating the data-fitted polynomials too far.
- The **Projective** algorithm generalizes Polynomial 1 in a different way, allowing transformations representing a central projection between 2 non-parallel planes, the image and the map canvas. Straight lines stay straight, but parallelism is not preserved and scale across the image varies consistently with the change in perspective.

This transformation type is most useful for georeferencing angled photographs (rather than flat scans) of good quality maps, or oblique aerial images. A minimum of 4 GCPs is required.

- Finally, the **Thin Plate Spline** (TPS) algorithm "rubber sheets" the raster using multiple local polynomials to match the GCPs specified, with overall surface curvature minimized. Areas away from GCPs will be moved around in the output to accommodate the GCP matching, but will otherwise be minimally locally deformed. TPS is most useful for georeferencing damaged, deformed, or otherwise slightly inaccurate maps, or poorly orthorectified aerials. It is also useful for approximately georeferencing and implicitly reprojecting maps with unknown projection type or parameters, but where a regular grid or dense set of ad-hoc GCPs can be matched with a reference map layer. It technically requires a minimum of 10 GCPs, but usually more to be successful.

In all of the algorithms except TPS, if more than the minimum GCPs are specified, parameters will be fitted so that the overall residual error is minimized. This is helpful to minimize the impact of registration errors, i.e. slight imprecisions in pointer clicks or typed coordinates, or other small local image deformations. Absent other GCPs to compensate, such errors or deformations could translate into significant distortions, especially near the edges of the georeferenced image. However, if more than the minimum GCPs are specified, they will match only approximately in the output. In contrast, TPS will precisely match all specified GCPs, but may introduce significant deformations between nearby GCPs with registration errors.

Define the Resampling method

The type of resampling you choose will likely depend on your input data and the ultimate objective of the exercise. If you don't want to change statistics of the raster (other than as implied by nonuniform geometric scaling if using other than the Linear, Helmert, or Polynomial 1 transformations), you might want to choose 'Nearest neighbour'. In contrast, 'cubic resampling', for instance, will usually generate a visually smoother result.

It is possible to choose between five different resampling methods:

1. Nearest neighbour
2. Linear
3. 立方体
4. Cubic Spline
5. Lanczos

Define the transformation settings

There are several options that need to be defined for the georeferenced output raster.

- The *Create world file* checkbox is only available if you decide to use the linear transformation type, because this means that the raster image actually won't be transformed. In this case, the *Output raster* field is not activated, because only a new world file will be created.
- For all other transformation types, you have to define an *Output raster*. As default, a new file ([file-name]_modified) will be created in the same folder together with the original raster image.
- As a next step, you have to define the *Target SRS* (Spatial Reference System) for the georeferenced raster (see *Working with Projections*).
- If you like, you can **generate a pdf map** and also a **pdf report**. The report includes information about the used transformation parameters, an image of the residuals and a list with all GCPs and their RMS errors.
- Furthermore, you can activate the *Set Target Resolution* checkbox and define the pixel resolution of the output raster. Default horizontal and vertical resolution is 1.
- The *Use 0 for transparency when needed* can be activated, if pixels with the value 0 shall be visualized transparent. In our example toposheet, all white areas would be transparent.

- Finally, *Load in QGIS when done* loads the output raster automatically into the QGIS map canvas when the transformation is done.


Show and adapt raster properties

Clicking on the *Raster properties* option in the *Settings* menu opens the *Layer properties* dialog of the raster file that you want to georeference.

Configure the georeferencer

- You can define whether you want to show GCP coordinates and/or IDs.
- As residual units, pixels and map units can be chosen.
- For the PDF report, a left and right margin can be defined and you can also set the paper size for the PDF map.
- Finally, you can activate to *Show Georeferencer window docked*.

Running the transformation

After all GCPs have been collected and all transformation settings are defined, just press the  Start georeferencing button to create the new georeferenced raster.

16.1 What's a mesh?

A mesh is an unstructured grid usually with temporal and other components. The spatial component contains a collection of vertices, edges and faces in 2D or 3D space:

- **vertices** - XY(Z) points (in the layer's coordinate reference system)
- **edges** - connect pairs of vertices
- **faces** - a face is a set of edges forming a closed shape - typically a triangle or a quadrilateral (quad), rarely polygons with more vertices

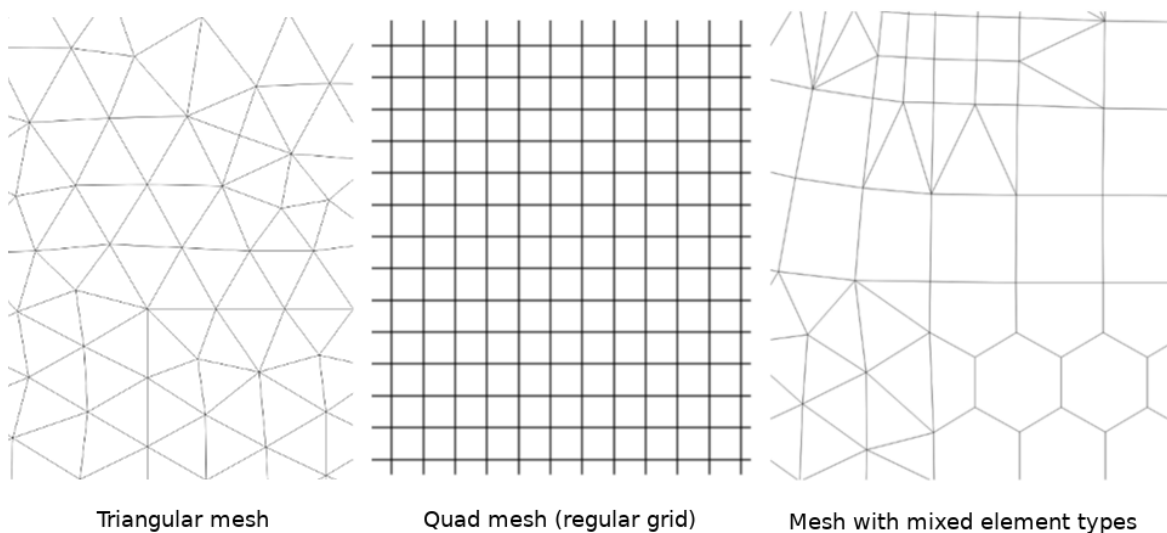


图 16.1: Different mesh types

QGIS can currently render mesh data using triangles or regular quads.

Mesh provides information about the spatial structure. In addition, the mesh can have datasets (groups) that assign a value to every vertex. For example, having a triangular mesh with numbered vertices as shown in the image below:

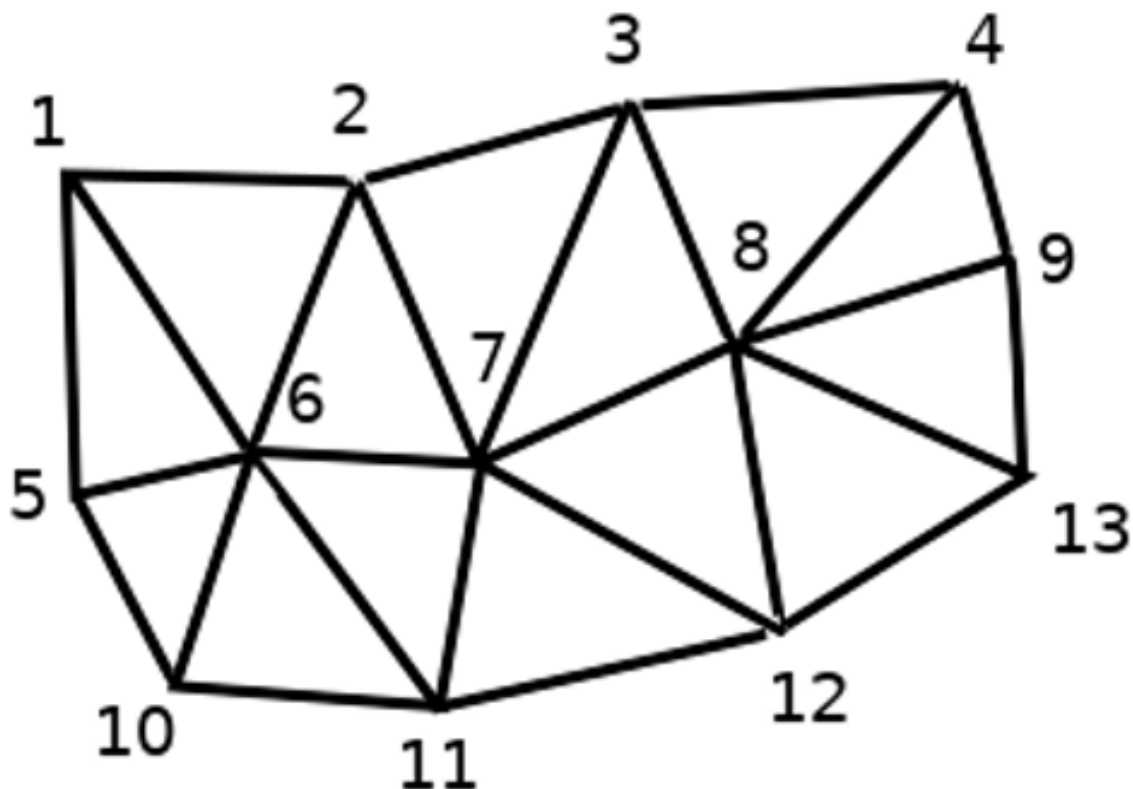


图 16.2: Triangular grid with numbered vertices

Each vertex can store different datasets (typically multiple quantities), and those datasets can also have a temporal dimension. Thus, a single file may contain multiple datasets.

The following table gives an idea about the information that can be stored in mesh datasets. Table columns represent indices of mesh vertices, each row represents one dataset. Datasets can have different datatypes. In this case, it stores wind velocity at 10m at a particular moments in time (t1, t2, t3).

In a similar way, the mesh dataset can also store vector values for each vertex. For example, wind direction vector at the given time stamps:

10 metre wind	1	2	3	...
10 metre speed at time=t1	17251	24918	32858	...
10 metre speed at time=t2	19168	23001	36418	...
10 metre speed at time=t3	21085	30668	17251	...
...
10m wind direction time=t1	[20,2]	[20,3]	[20,4.5]	...
10m wind direction time=t2	[21,3]	[21,4]	[21,5.5]	...
10m wind direction time=t3	[22,4]	[22,5]	[22,6.5]	...
...

We can visualize the data by assigning colors to values (similarly to how it is done with *Singleband pseudocolor* raster rendering) and interpolating data between vertices according to the mesh topology. It is common that some quantities are 2D vectors rather than being simple scalar values (e.g. wind direction). For such quantities it is desirable to display arrows indicating the directions.

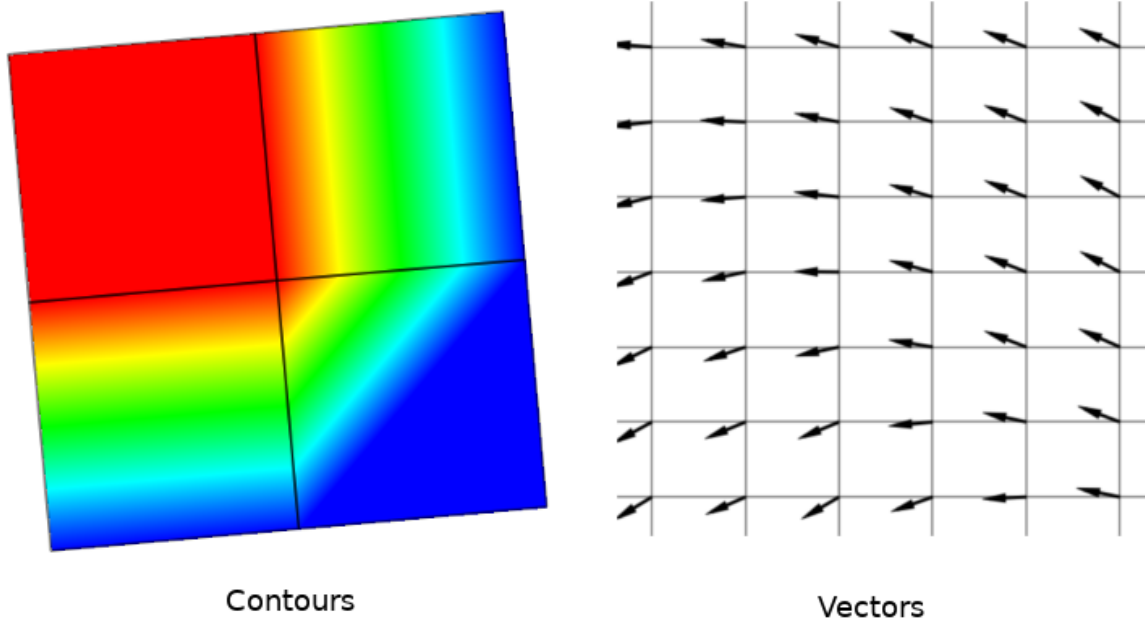



图 16.3: Possible visualisation of mesh data

16.2 Supported formats

QGIS accesses mesh data using the [MDAL drivers](#). Hence, the natively supported formats are:

- NetCDF: Generic format for scientific data
- GRIB: Format commonly used in meteorology
- XMDF: As an example, hydraulic outputs from TUFLOW modelling package
- DAT: Outputs of various hydrodynamic modelling packages (e.g. BASEMENT, HYDRO_AS-2D, TUFLOW)
- 3Di: 3Di modelling package format based on Climate and Forecast Conventions (<http://cfconventions.org/>)
- Some examples of mesh datasets can be found at <https://apps.ecmwf.int/datasets/data/interim-full-daily/levtype=sfc/>

To load a mesh dataset into QGIS, use the  *Mesh* tab in the *Data Source Manager* dialog. Read *Loading a mesh layer* for more details.

16.3 Mesh Dataset Properties

16.3.1 Information Properties

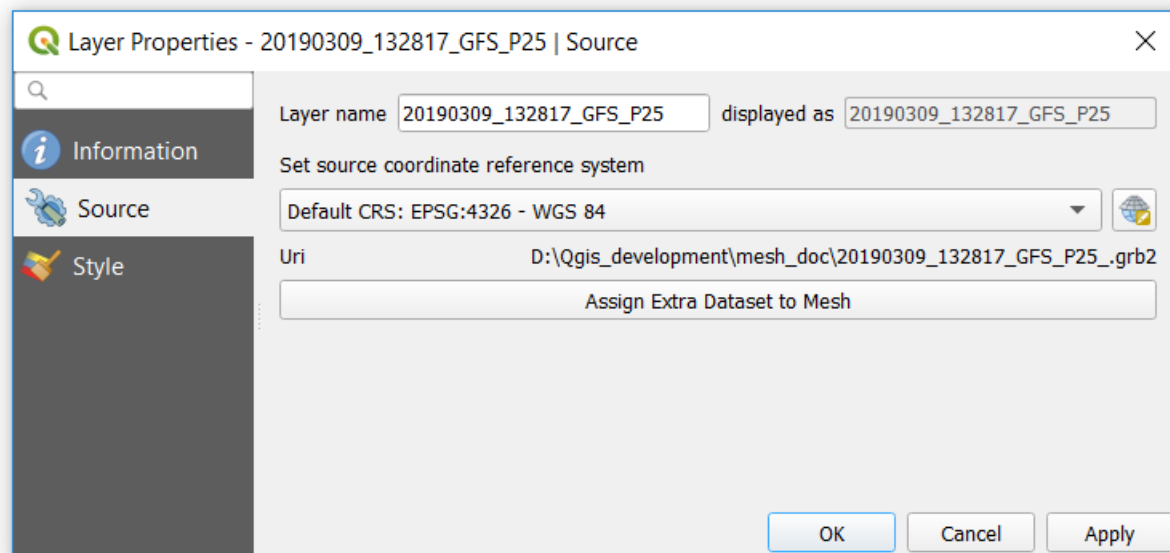



图 16.4: Mesh Layer Properties

The *Information* tab is read-only and represents an interesting place to quickly grab summarized information and metadata on the current layer. Provided information are (based on the provider of the layer) uri, vertex count, face count and dataset groups count.


16.3.2 Source Properties

The *Source* tab displays basic information about the selected mesh, including:

- the Layer name to display in the *Layers* panel
- setting the Coordinate Reference System: Displays the layer's *Coordinate Reference System (CRS)*. You can change the layer's CRS by selecting a recently used one in the drop-down list or clicking on  *Select CRS* button (see *Coordinate Reference System Selector*). Use this process only if the CRS applied to the layer is wrong or if none was applied.

Use the *Assign Extra Dataset to Mesh* button to add more groups to the current mesh layer.

16.3.3 Symbology Properties

Click the  *Symbology* button to activate the dialog as shown in the following image:

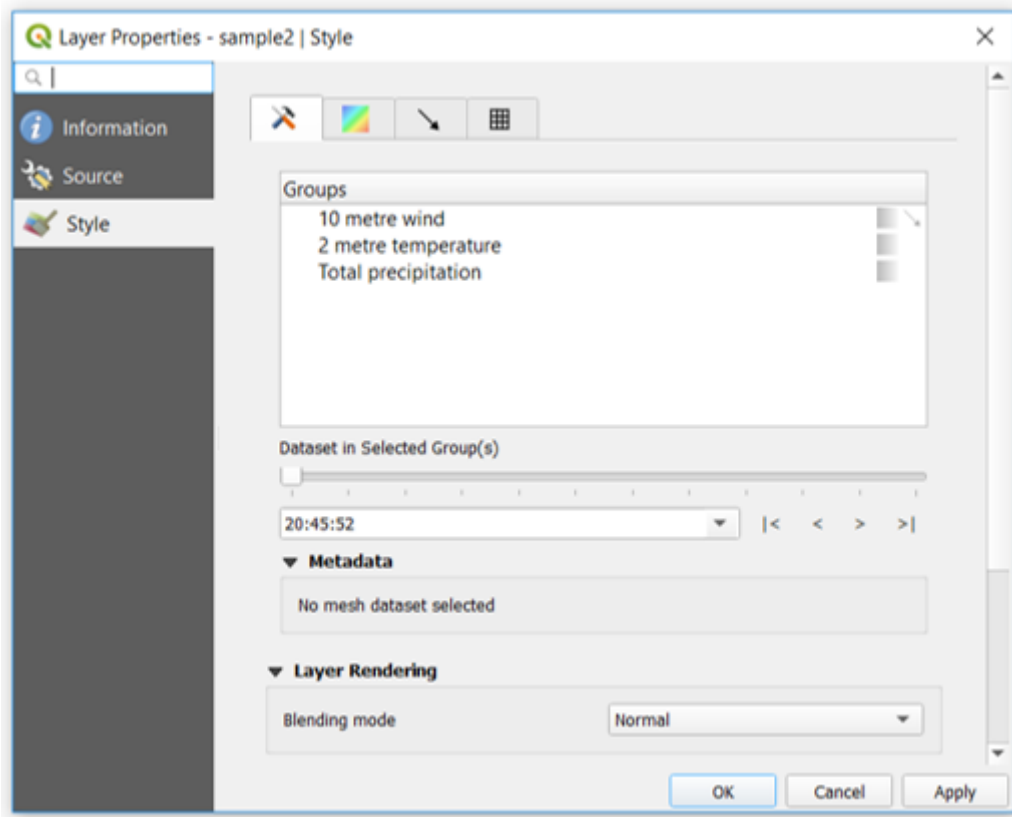



图 16.5: Mesh Layer Symbology



Symbology properties are divided in several tabs:

- *General*
- *Contours Symbology*
- *Vectors Symbology*
- *Rendering*

General

The tab  presents the following items:

- groups available in the mesh dataset
- dataset in the selected group(s), for example, if the layer has a temporal dimension
- metadata if available
- *blending mode* available for the selected dataset.

The slider , the combo box  and the |<, <, >, >| buttons allow to explore another dimension of the data, if available. As the slider moves, the metadata is presented accordingly. See the figure *Mesh groups* below as an example. The map canvas will display the selected dataset group as well.

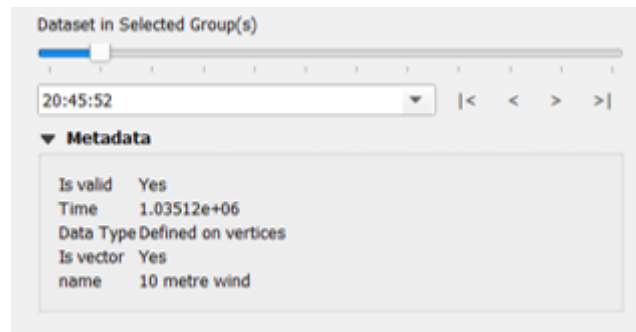



图 16.6: Dataset in Selected Group(s)

You can apply symbology to each group using the tabs.

Contours Symbology

Under *Groups*, click on  to show contours with default visualization parameters.

In the tab  you can see and change the current visualization options of contours for the selected group, as shown in 图 16.7 below:

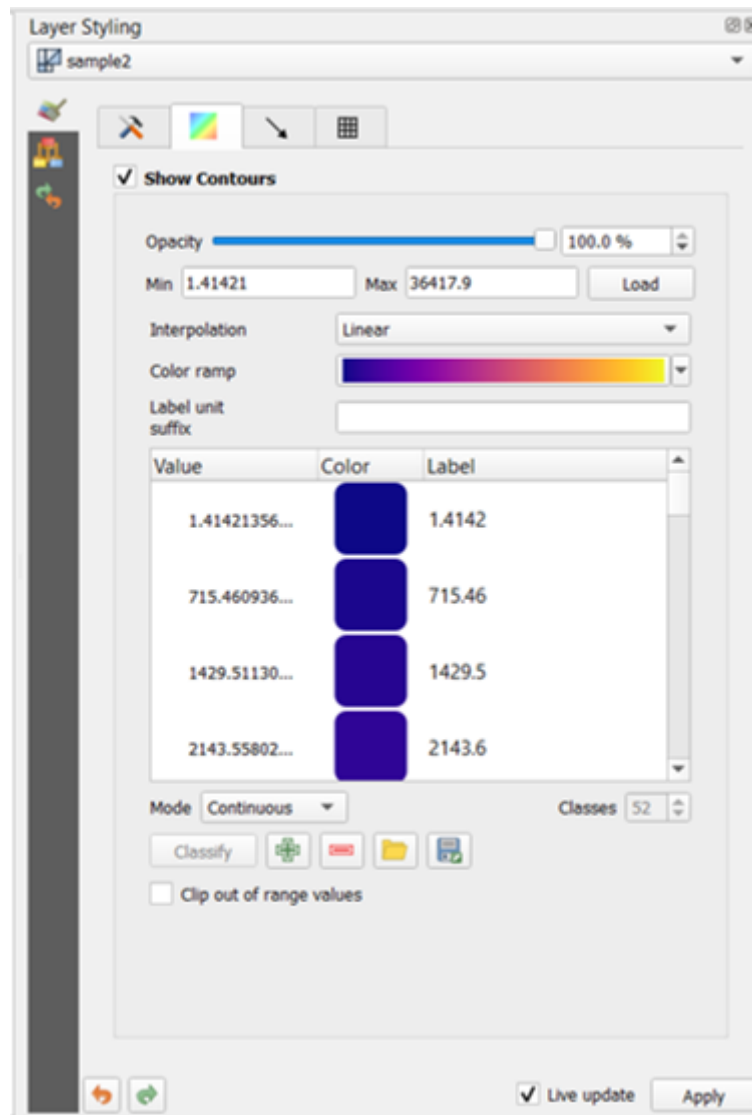


图 16.7: Styling Contours in a Mesh Layer

Use the slide bar or combo box to set the opacity of the current group.



Use *Load* to adjust the min and max values of the current group.

The *Interpolation* list contains three options to render contours: *Linear*, *Discrete* and *Exact*.




The *Color ramp* widget opens the *color ramp drop-down shortcut*.

The *Label unit suffix* is a label added after the value in the legend.

By selecting *Continuous* in the classification *Mode*, QGIS creates classes automatically considering the *Min* and *Max* values. With ‘Equal interval’, you only need to select the number of classes using the combo box *Classes* and press the button *Classify*.

The button  *Add values manually* adds a value to the individual color table. The button  *Remove selected row* deletes a value from the individual color table. Double clicking on the value column lets you insert a specific value. Double clicking on the color column opens the dialog *Change color*, where you can select a color to apply on that value.

Vectors Symbology

In the tab , click on  to display vectors if available. The map canvas will display the vectors in the selected group with default parameters. Click on the tab  to change the visualization parameters for vectors as shown in the image below:

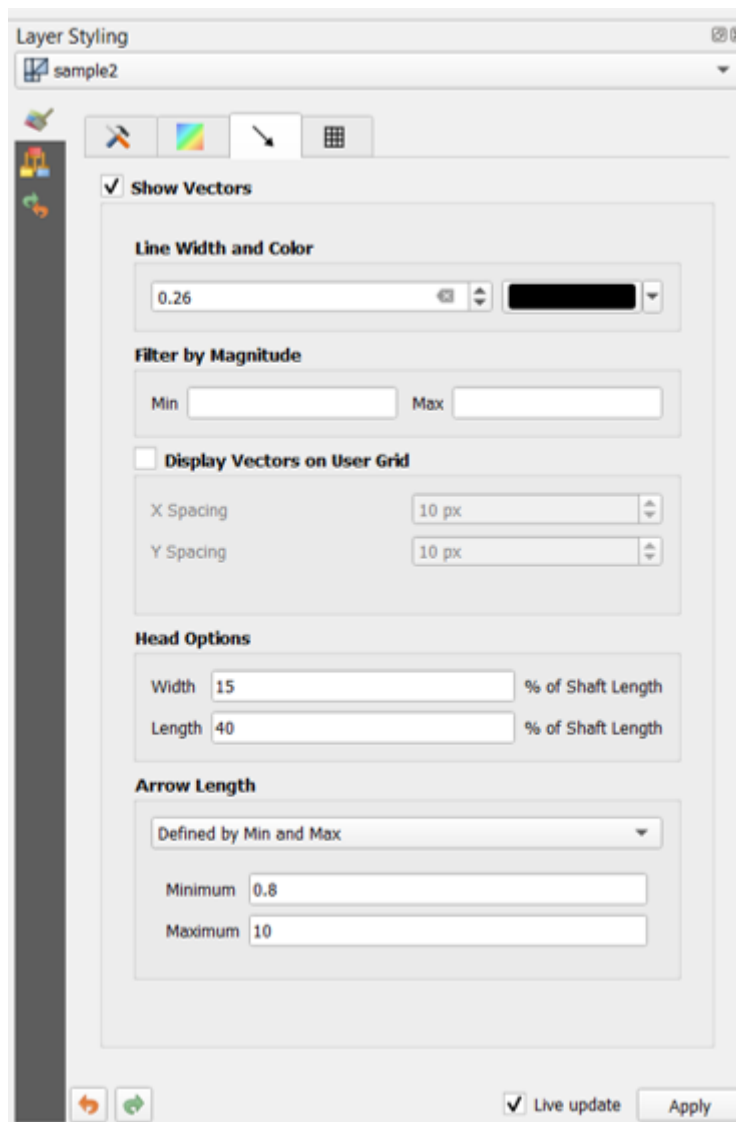



图 16.8: Styling Vectors in a Mesh Layer

The line width can be set using the combo box or typing the value. The color widget opens the dialog *Change color*, where you can select a color to apply to vectors.

Enter values for *Min* and *Max* to filter vectors according to their magnitude.

Check on the box  *Display Vectors on User Grid* and specify the *X spacing* and the *Y spacing*, QGIS will render the vector considering the given spacing.

With the Head Options *Head Options*, QGIS allows the shape of the arrow head to be set by specifying width and length (in percentage).

Vector's *Arrow length* can be rendered in QGIS in three different ways:

- Defined by Min and Max: You specify the minimum and maximum length for the vectors, QGIS will adjust their visualization accordingly
- Scale to magnitude: You specify the (multiplying) factor to use
- Fixed: all the vectors are shown with the same length

Rendering

In the tab , QGIS offers two possibilities to display the grid, as shown in  16.9:

- Native Mesh Rendering that shows quadrants
- Triangular Mesh Rendering that display triangles

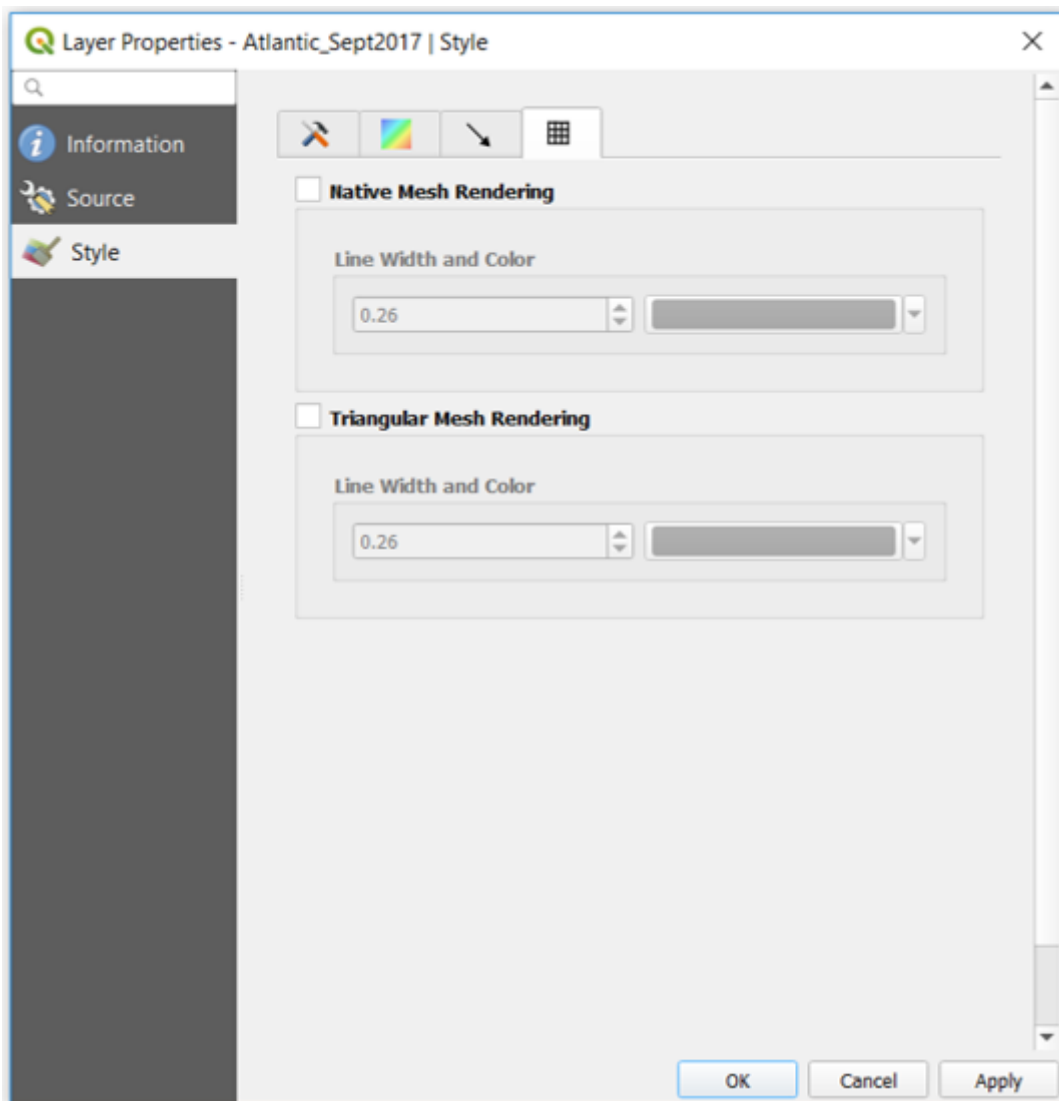


图 16.9: Mesh Rendering

The line width and color can be changed in this dialog, and both the grid renderings can be turned off.

17.1 What are Vector Tiles?

Vector tiles are packets of geographic data, packaged into pre-defined roughly-square shaped "tiles" for transfer over the web. They combine pre-rendered raster map tiles and vector map tiles. The vector tile server returns vector map data, which has been clipped to the boundaries of each tile, instead of a pre-rendered map image. The clipped tiles represent the zoom-levels of the vector tile service, derived from a pyramid approach. Using this structure, the data-transfer is reduced in comparison to un-tiled vector maps. Only data within the current map view, and at the current zoom level need to be transferred. Also, compared to a tiled raster map, data transfer is also greatly reduced, as vector data is typically much smaller than a rendered bitmap. Vector tiles do not have any styling information assigned so QGIS needs to apply a cartographic style in order to display the data.

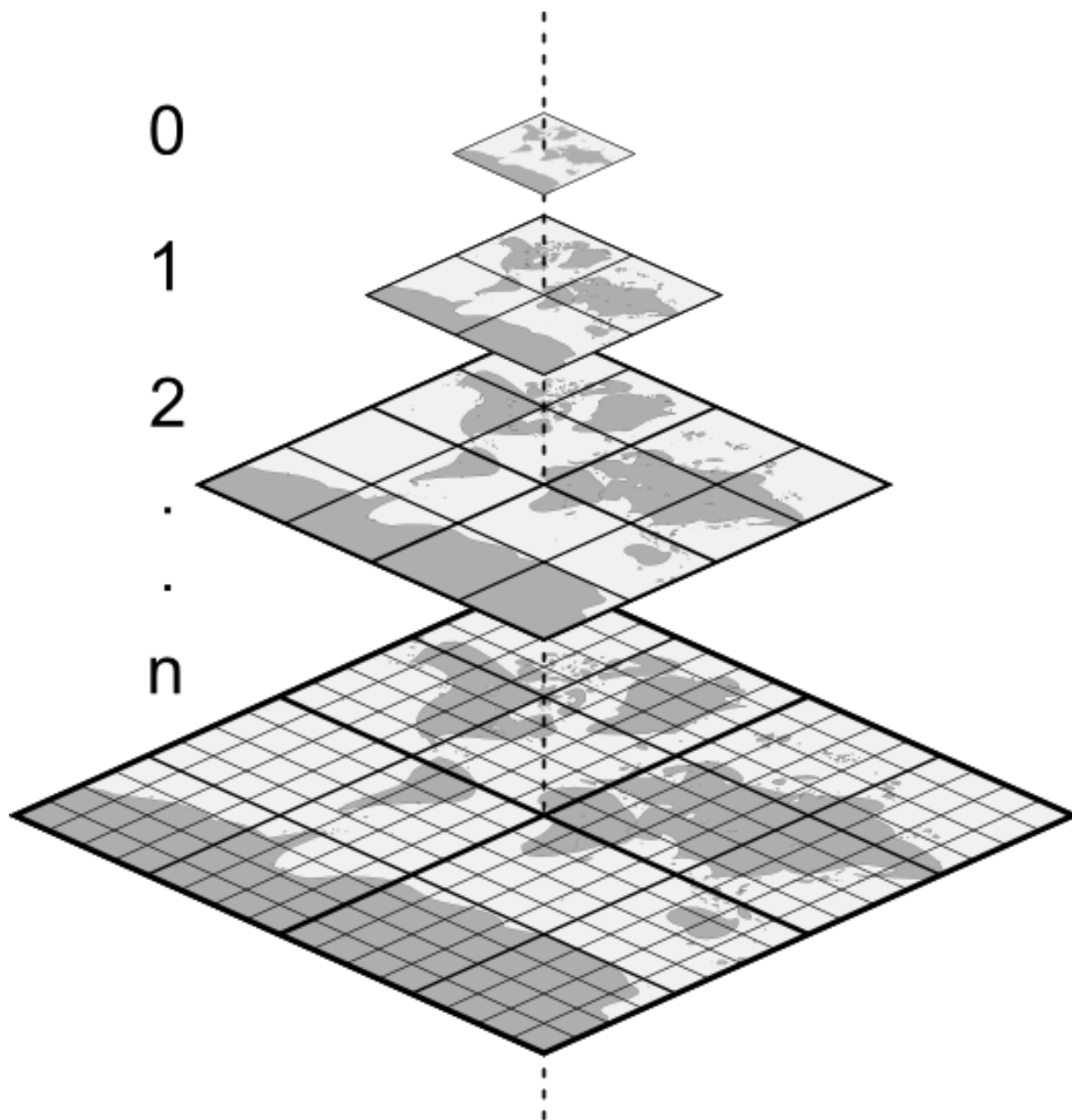


图 17.1: Pyramid structure of vector tiles with zoom-levels

17.2 Supported Formats

There is support for vector tiles through:

- remote sources (HTTP/S) - with XYZ template - `type=xyz&url=http://example.com/{z}/{x}/{y}.pbf`
- local files - with XYZ template - e.g. `type=xyz&url=file:///path/to/tiles/{z}/{x}/{y}.pbf`
- local MBTiles database - e.g. `type=mbtiles&url=file:///path/to/file.mbtiles`


Laying out the maps

With Print Layouts and Reports you can create maps and atlases, and print them or save them as image, PDF or SVG files.





18.1 Overview of the Print Layout


The print layout provides growing layout and printing capabilities. It allows you to add elements such as the QGIS map canvas, text labels, images, legends, scale bars, basic shapes, arrows, attribute tables and HTML frames. You can size, group, align, position and rotate each element and adjust their properties to create your layout. The layout can be printed or exported to image formats, PostScript, PDF or to SVG. You can save the layout as a template and load it again in another session. Finally, generating several maps based on a template can be done through the atlas generator.

18.1.1 Sample Session for beginners

Before you start to work with the print layout, you need to load some raster or vector layers in the QGIS map canvas and adapt their properties to suit your own convenience. After everything is rendered and symbolized to your liking, click the  New Print Layout icon in the toolbar or choose *File* ► *New Print Layout*. You will be prompted to choose a title for the new layout.

To demonstrate how to create a map please follow the next instructions.

1. On the left side, select the  Add map toolbar button and draw a rectangle on the canvas holding down the left mouse button. Inside the drawn rectangle the QGIS map view to the canvas.
2. Select the  Add scalebar toolbar button and click with the left mouse button on the print layout canvas. A scalebar will be added to the canvas.
3. Select the  Add legend toolbar button and draw a rectangle on the canvas holding down the left mouse button. Inside the drawn rectangle the legend will be drawn.
4. Select the  Select/Move item icon to select the map on the canvas and move it a bit.

5. While the map item is still selected you can also change the size of the map item. Click while holding down the left mouse button, in a white little rectangle in one of the corners of the map item and drag it to a new location to change its size.
6. Click the *Item Properties* panel on the left down side and find the setting for the orientation. Change the value of the setting *Map orientation* to '15.00°'. You should see the orientation of the map item change.
7. Now, you can print or export your print layout to image formats, PDF or to SVG with the export tools in *Layout* menu.
8. Finally, you can save your print layout within the project file with the  Save Project button.



You can add multiple elements to the print layout. It is also possible to have more than one map view or legend or scale bar in the print layout canvas, on one or several pages. Each element has its own properties and, in the case of the map, its own extent. If you want to remove any elements from the layout canvas you can do that with the *Delete* or the *Backspace* key.

18.1.2 The Layout Manager

The *Layout Manager* is the main window to manage print layouts in the project. It gives you an overview of existing print layouts and reports in the project and offers tools to:

- search for a layout;
- add new print layout or new report from scratch, template or duplicating an existing one;
- rename or delete any of them;
- open them in the project.

To open the layout manager dialog:

- from the main QGIS dialog, select *Project ► Layout Manager...* menu or click on the  Layout Manager button in the *Project Toolbar*;
- from a print layout or report dialog, select *Layout ► Layout Manager...* menu or click on the  Layout Manager button in the *Layout Toolbar*.

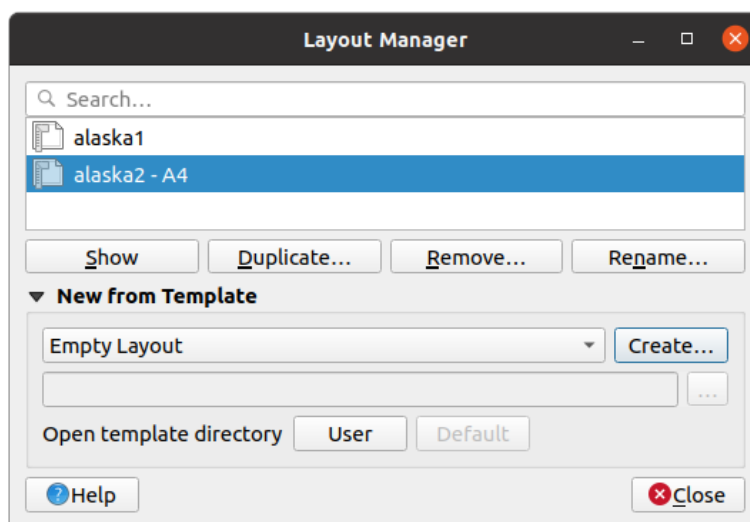


图 18.1: The Print Layout Manager

The layout manager lists in its upper part all the available print layouts or reports in the project with tools to:

- show the selection: you can select multiple reports and/or print layout(s) and open them in one-click. Double-click a name also opens it;

- duplicate the selected print layout or report (available only if one item is selected): it creates a new dialog using the selected one as template. You'll be prompted to choose a new title for the new layout;
- rename the report or layout (available only if one item is selected): you'll be prompted to choose a new title for the layout;
- remove the layout: the selected print layout(s) will be deleted from the project.

In the lower part, it's possible to create new print layouts or reports from scratch or a template. By default, QGIS will look for templates in the user profile and the application template directories (accessible with the two buttons at the bottom of the frame) but also in any folder declared as *Path(s) to search for extra print templates* in *Settings ► Options ► Layouts*. Found templates are listed in the combobox. Select an item and press the *Create* button to generate a new report or print layout.

You can also use layout templates from a custom folder; in that case, select *specific* in the templates drop-down list, browse to the template and press *Create*.

小技巧: Creating template-based print layouts from Browser panel

Drag-and-drop a print layout template `.qpt` file from any file browser onto the map canvas or double-click it in the *Browser panel* generates a new print layout from the template.

18.1.3 Menus, tools and panels of the print layout

Opening the print layout provides you with a blank canvas that represents the paper surface when using the print option. Initially you find buttons on the left beside the canvas to add print layout items: the current QGIS map canvas, text labels, images, legends, scale bars, basic shapes, arrows, attribute tables and HTML frames. In this toolbar you also find buttons to navigate, zoom in on an area and pan the view on the layout as well as buttons to select any layout item and to move the contents of the map item.

图 18.2 shows the initial view of the print layout before any elements are added.

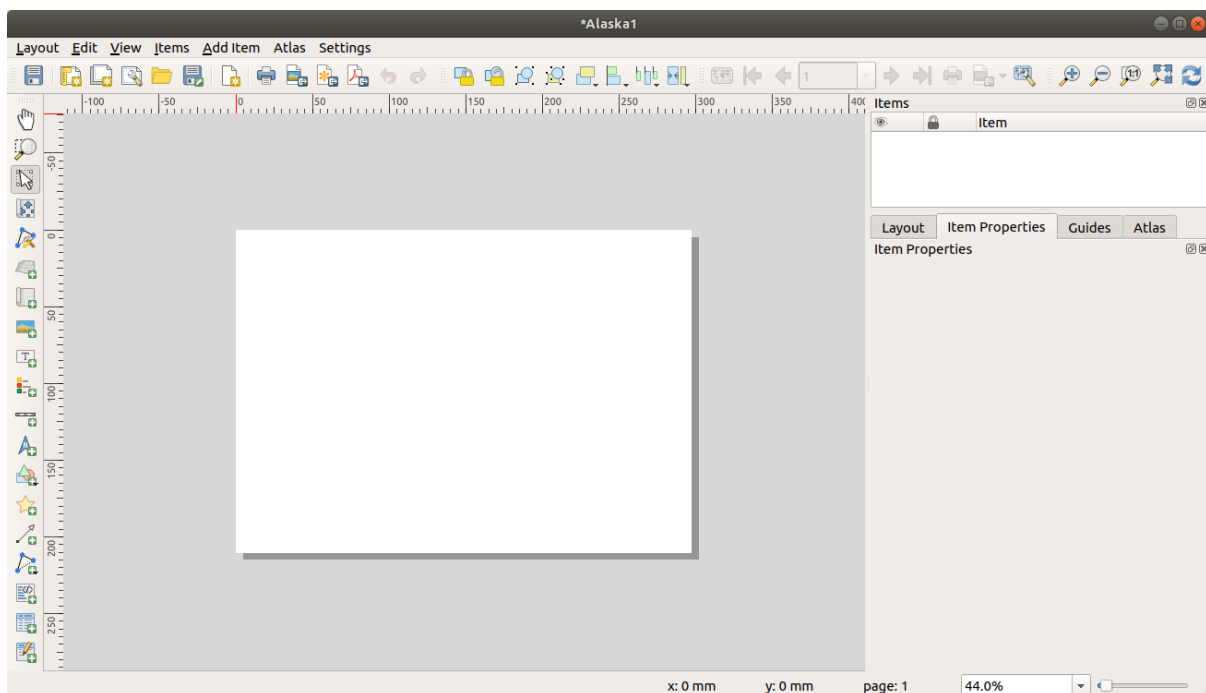



图 18.2: Print Layout

On the right beside the canvas you find two set of panels. The upper one holds the panels *Items* and *Undo History* and the lower holds the panels *Layout*, *Item properties* and *Atlas generation*.

- The *Items* panel provides a list of all the print layout items added to the canvas and ways to globally interact with them (see *The Items Panel* for more information).
- The *Undo History* panel displays a history of all changes applied to the layout. With a mouse click, it is possible to undo and redo layout steps back and forth to a certain status.
- The *Layout* panel allows you to set general parameters to apply to the layout when exporting or working within (see *The Layout Panel* for more details);
- The *Item Properties* panel displays the properties for the selected item. Click the  *Select/Move item* icon to select an item (e.g., legend, scale bar or label) on the canvas. Then click the *Item Properties* panel and customize the settings for the selected item (see *布局项* for detailed information on each item settings).
- The *Atlas* panel allows you to enable the generation of an atlas for the current layout and gives access to its parameters (see *Generate an Atlas* for detailed information on atlas generation usage).

In the bottom part of the print layout window, you can find a status bar with mouse position, current page number, a combo box to set the zoom level, the number of selected items if applicable and, in the case of atlas generation, the number of features.





In the upper part of the print layout window, you can find menus and other toolbars. All print layout tools are available in menus and as icons in a toolbar.



The toolbars and the panels can be switched off and on using the right mouse button over any toolbar or through *View ► Toolbars ►* or *View ► Panels ►*.




Menus and Tools

Layout menu













The *Layout* provides action to manage the layout:

- Save the project file directly from the print layout window.
- Create a new and blank print layout with  *New Layout...*
-  *Duplicate Layout...* : Create a new print layout by duplicating the current one.
- Remove the current layout with  *Delete Layout...*
- Open the  *Layout Manager...*
- *Layouts ►* : Open an existing print layout.

Once the layout is designed, with  *Save as Template* and  *Add Items from Template* icons, you can save the current state of a print layout session as a `.qpt` template file and load its items again in another session/print layout.

In the *Layout* menu, there are also powerful ways to share geographical information produced with QGIS that can be included in reports or published. These tools are  *Export as Image...*,  *Export as PDF...*,  *Export as SVG...* and  *Print...*

Below is a list of all the available tools in this menu with some convenient information.

Tool	Shortcut	Toolbar	Reference
 Save Project	Ctrl+S	Layout	QGIS 工程简介
 New Layout	Ctrl+N	Layout	The Layout Manager
 Duplicate Layout		Layout	The Layout Manager
 Delete Layout			
 Layout Manager...		Layout	The Layout Manager
Layouts ►			
Layout Properties...			The Layout Panel
Rename Layout...			
 Add Pages...		Layout	Working with the page properties
 Add Items from Template		Layout	Creating a layout item
 Save as Template...		Layout	The Layout Manager
 Export as Image...		Layout	Export as Image
 Export as SVG...		Layout	Export as SVG
 Export as PDF...		Layout	Export as PDF
Page Setup...	Ctrl+Shift+P		
 Print...	Ctrl+P	Layout	Creating an Output
Close	Ctrl+Q		















Edit menu

The *Edit* menu offers tools to manipulate print layout items. It includes common actions like selection tools, Copy/Cut/Paste and undo/redo (see [The Undo History Panel: Revert and Restore actions](#)) functionality for the items in the layout.

When using the Paste action, the elements will be pasted according to the current mouse position. Using the *Edit ► Paste in Place* action or pressing Ctrl+Shift+V will paste the items into the current page, at the same position they were in their initial page. It ensures to copy/paste items at the same place, from page to page.


Below is a list of all the available tools in this menu with some convenient information.

表 18.1: Available Tools

Tool	Shortcut	Toolbar	Reference
 Undo (last change)	Ctrl+Z	Layout	<i>The Undo History Panel: Revert and Restore actions</i>
 Redo (last reverted change)	Ctrl+Y	Layout	<i>The Undo History Panel: Revert and Restore actions</i>
 Delete	Del		
 Cut	Ctrl+X		
 Copy	Ctrl+C		
 Paste	Ctrl+V		
Paste in place	Ctrl+Shift+V		
 Select All	Ctrl+A		
 Deselect all	Ctrl+Shift+A		
 Invert Selection			
Select Next Item Below	Ctrl+Alt+[
Select Next Item above	Ctrl+Alt+]		
 Pan Layout	P	Toolbox	
 Zoom	Z	Toolbox	
 Select/Move Item	V	Toolbox	<i>Interacting with layout items</i>
 Move Content	C	Toolbox	<i>The Map Item</i>
 Edit Nodes Item		Toolbox	基于结点的形状项

View menu






The *View* menu gives access to navigation tools and helps to configure general behavior of the print layout. Beside the common zoom tools, you have means to:

-  Refresh view (if you find the view in an inconsistent state);
- enable a *grid* you could snap items to when moving or creating them. Grids setting is done in *Settings ► Layout Options...* or in the *Layout Panel*;
- enable *guides* you could snap items to when moving or creating them. Guides are red lines that you can create by clicking in the ruler (above or at the left side of the layout) and drag and drop to the desired location;
- *Smart Guides*: uses other layout items as guides to dynamically snap to as you move or reshape an item;
- *Clear Guides* to remove all current guides;
- *Show Bounding box* around the items to better identify your selection;
- *Show Rules* around the layout;
- *Show Pages* or set up pages to transparent. Often layout is used to create non-print layouts, e.g. for inclusion in presentations or other documents, and it's desirable to export the composition using a totally transparent background. It's sometimes referred to as "infinite canvas" in other editing packages.

In the print layout, you can change the zoom level using the mouse wheel or the slider and combo box in the status bar. If you need to switch to pan mode while working in the layout area, you can hold the Spacebar or the mouse wheel. With Ctrl+Spacebar, you can temporarily switch to Zoom In mode, and with Ctrl+Alt+Spacebar, to Zoom Out mode.









Panels and toolbars can be enabled from the *View* ► menu. To maximise the space available to interact with a composition you can check the *View* ► *Toggle Panel Visibility* option or press **Ctrl+Tab**; all panels are hidden and only previously visible panels are restored when unchecked.

It's also possible to switch to a full screen mode to have more space to interact with by pressing **F11** or using *View* ► *Toggle Full Screen*.

Tool	Shortcut	Toolbar	Reference
 <i>Refresh</i>	F5	<i>Navigation</i>	
<i>Preview</i> ►			
 <i>Zoom In</i>	Ctrl++	<i>Navigation</i>	
 <i>Zoom Out</i>	Ctrl+-	<i>Navigation</i>	
 <i>Zoom to 100%</i>	Ctrl+1	<i>Navigation</i>	
 <i>Zoom Full</i>	Ctrl+0	<i>Navigation</i>	
<i>Zoom to Width</i>			
 <i>Show Grid</i>	Ctrl+'		<i>Guides and Grid</i>
<input type="checkbox"/> <i>Snap to Grid</i>	Ctrl+Shift+'		<i>Guides and Grid</i>
<input checked="" type="checkbox"/> <i>Show Guides</i>	Ctrl+;		<i>Guides and Grid</i>
<input checked="" type="checkbox"/> <i>Snap to Guides</i>	Ctrl+Shift+;		<i>Guides and Grid</i>
<input checked="" type="checkbox"/> <i>Smart Guides</i>	Ctrl+Alt+;		
<i>Manage Guides...</i>			<i>The Guides Panel</i>
<i>Clear Guides</i>			<i>The Guides Panel</i>
<input checked="" type="checkbox"/> <i>Show Rulers</i>	Ctrl+R		
<input checked="" type="checkbox"/> <i>Show Bounding Boxes</i>	Ctrl+Shift+B		
<input checked="" type="checkbox"/> <i>Show Pages</i>			
<i>Toolbars</i> ►			面板和工具栏
<i>Panels</i> ►			面板和工具栏
<input type="checkbox"/> <i>Toggle Full Screen</i>	F11		视图
<input type="checkbox"/> <i>Toggle Panel Visibility</i>	Ctrl+Tab		视图

Items menu

The *Items* helps you configure items' position in the layout and the relations between them (see *Interacting with layout items*).











Tool	Shortcut	Toolbar	Reference
 <i>Group</i>	Ctrl+G	<i>Actions</i>	<i>Grouping items</i>
 <i>Ungroup</i>	Ctrl+Shift+G	<i>Actions</i>	<i>Grouping items</i>
 <i>Raise</i>	Ctrl+]	<i>Actions</i>	<i>Alignment</i>
 <i>Lower</i>	Ctrl+[<i>Actions</i>	<i>Alignment</i>
 <i>Bring to Front</i>	Ctrl+Shift+]	<i>Actions</i>	<i>Alignment</i>
 <i>Send to Back</i>	Ctrl+Shift+[<i>Actions</i>	<i>Alignment</i>
 <i>Lock Selected Items</i>	Ctrl+L	<i>Actions</i>	<i>Locking items</i>
 <i>Unlock All</i>	Ctrl+Shift+L	<i>Actions</i>	<i>Locking items</i>
<i>Align Items</i> ►		<i>Actions</i>	<i>Alignment</i>
<i>Distribute Items</i> ►		<i>Actions</i>	<i>Moving and resizing items</i>
<i>Resize</i> ►		<i>Actions</i>	<i>Moving and resizing items</i>

Add Item menu

These are tools to create layout items. Each of them is deeply described in 布局项 chapter.

Tool	Toolbar	Reference
 <i>Add Map</i>	<i>Toolbox</i>	<i>The Map Item</i>
 <i>Add Picture</i>	<i>Toolbox</i>	<i>The Picture Item</i>
 <i>Add Label</i>	<i>Toolbox</i>	<i>The Label Item</i>
 <i>Add Legend</i>	<i>Toolbox</i>	<i>The Legend Item</i>
 <i>Add Scale Bar</i>	<i>Toolbox</i>	<i>The Scale Bar Item</i>
 <i>Add North Arrow</i>	<i>Toolbox</i>	<i>The North Arrow Item</i>
 <i>Add Shape</i> ►	<i>Toolbox</i>	常规图形项
 ► <i>Add Rectangle</i>	<i>Toolbox</i>	常规图形项
 ► <i>Add Ellipse</i>	<i>Toolbox</i>	常规图形项
 ► <i>Add Triangle</i>	<i>Toolbox</i>	常规图形项
 <i>Add Marker</i>	<i>Toolbox</i>	
 <i>Add Arrow</i>	<i>Toolbox</i>	<i>The Arrow Item</i>
 <i>Add Node Item</i> ►	<i>Toolbox</i>	基于结点的形状项
 ► <i>Add Polygon</i>	<i>Toolbox</i>	基于结点的形状项
 ► <i>Add Polyline</i>	<i>Toolbox</i>	基于结点的形状项
 <i>Add HTML</i>	<i>Toolbox</i>	<i>The HTML Frame Item</i>
 <i>Add Attribute Table</i>	<i>Toolbox</i>	<i>The attribute table item</i>
 <i>Add Fixed Table</i>	<i>Toolbox</i>	<i>The fixed table item</i>
 <i>Add 3D Map</i>	<i>Toolbox</i>	<i>The 3D Map Item</i>

Atlas menu

Tool	Shortcut	Toolbar	Reference
 <i>Preview Atlas</i>	Ctrl+Alt+/ 	<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>First Feature</i>	Ctrl+<	<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>Previous Feature</i>	Ctrl+,	<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>Next Feature</i>	Ctrl+.	<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>Last feature</i>	Ctrl+>	<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>Print Atlas...</i>		<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>Export Atlas as Images...</i>		<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>Export Atlas as SVG...</i>		<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>Export Atlas as PDF...</i>		<i>Atlas</i>	<i>Preview and generate an atlas</i>
 <i>Atlas Settings</i>		<i>Atlas</i>	<i>Generate an Atlas</i>



Settings Menu

The *Settings ► Layout Options...* menu is a shortcut to *Settings ► Options ► Layouts* menu of QGIS main canvas. Here, you can set some options that will be used as default on any new print layout:

- *Layout defaults* let you specify the default font to use;
- With *Grid appearance*, you can set the grid style and its color. There are three types of grid: **Dots**, **Solid** lines and **Crosses**;
- *Grid and guide defaults* defines spacing, offset and tolerance of the grid (see *Guides and Grid* for more details);
- *Layout Paths*: to manage list of custom paths to search print templates.

Contextual menus

Depending on where you right-click in the print layout dialog, you open a contextual menu with various features:

- Right-click on the menu bar or any toolbar and you get the list of layout panels and toolbars you can enable or disable in one-click.
- Right-click over a ruler and you can  *Show Guides*,  *Snap to Guides*, *Manage Guides...* opening the *Guides panel* or *Clear Guides*. It's also possible to hide the rulers.
- Right-click in the print layout canvas and:
 - You'll be able to *Undo* and *Redo* recent changes, or *Paste* any copied item (only available if no item is selected).
 - If you click over a page, you can additionally access the current *Page Properties* panel or *Remove Page*.
 - If you click on a selected item then you can cut or copy it as well as open the *Item Properties* panel.
 - If more than one item are selected, then you can either group them and/or ungroup if at least one group is already in the selection.
- Right-click inside a text box or spinbox widget of any layout panel provides edit options to manipulate its content.

The Layout Panel

In the *Layout* panel, you can define the global settings of your print layout.

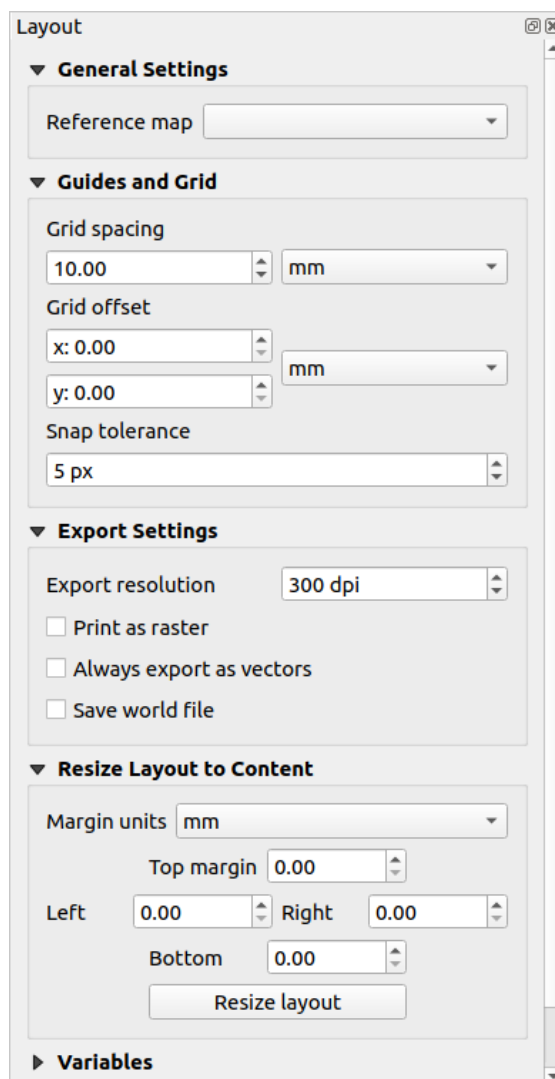


图 18.3: Layout Settings in the Print Layout

General settings

In a print layout, you can use more than one map item. The *Reference map* represents the map item to use as the layout's master map. It's assigned as long as there's a map item in the layout. The layout will use this map in any of their properties and variables calculating units or scale. This includes exporting the print layout to georeferenced formats.

Moreover, new layout items such as scale bar, legend or north arrow have by default their settings (orientation, displayed layers, scale, ...) bound to the map item they are drawn over, and fall back to the reference map if no overlapping map.

Guides and Grid

You can put some reference marks on your paper sheet to help you accurately place some items. These marks can be:

- simple horizontal or vertical lines (called **Guides**) put at the position you want (see *The Guides Panel* for guides creation).
- or regular **Grid**: a network of horizontal and vertical lines superimposed over the layout.

Settings like *Grid spacing* or *Grid offset* can be adjusted in this group as well as the *Snap tolerance* to use for items. The tolerance is the maximum distance below which the mouse cursor is snapped to a grid or a guide, while moving, resizing or creating an item.

Whether grid or guides should be shown is set in *View* menu. There, you can also decide if they might be used to snap layout items. When both a grid line and a guide line are within tolerance of a point, guides will always take precedence - since they have been manually set (hence, assumption that they have been explicitly placed at highly desirable snapping locations, and should be selected over the general grid).

注解: In the *Settings* ► *Layout Options* menu, you can also set the grid and guides parameters exposed above. However, these options will only apply as defaults to new print layouts.

Export settings

You can define a resolution to use for all exported maps in *Export resolution*. This setting can then be overridden each time you export a map.

Because of some advanced rendering options (*blending mode, effects...*), a layout item may need rasterization in order to be exported correctly. QGIS will individually rasterize it without forcing every other item to also be rasterized. This allows printing or saving as PostScript or PDF to keep items as much as possible as vectors, e.g. a map item with layer opacity won't force labels, scale bars, etc to be rasterized too. You can however:

- force all the items to be rasterized checking the *Print as raster* box;
- or use the opposite option, i.e. *Always export as vectors*, to force the export to keep items as vectors when exported to a compatible format. Note that in some cases, this could cause the output to look different to layout.

Where the format makes it possible (e.g., .TIF, .PDF) exporting a print layout results by default in a georeferenced file (based on the *Reference map* item in the *General settings* group). For other formats, georeferenced output requires you to generate a world file by checking *Save world file*. The world file is created beside the exported map(s), has the name of the page output with the reference map item and contains information to georeference it easily.



Resize layout to content

Using the *Resize page* tool in this group, you create a unique page composition whose extent covers the current contents of the print layout (with some optional *margins* around the cropped bounds).

Note that this behavior is different from the *crop to content* option in that all the items are placed on a real and unique page in replacement of all the existing pages.

Variables

The *Variables* lists all the variables available at the layout's level (which includes all global and project's variables).

It also allows the user to manage layout-level variables. Click the  button to add a new custom layout-level variable. Likewise, select a custom layout-level variable from the list and click the  button to remove it.

More information on variables usage in the *General Tools* section.

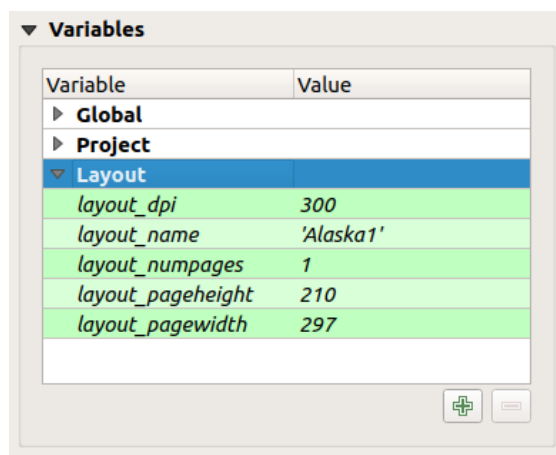



图 18.4: Variables Editor in the Print Layout

Working with the page properties

A layout can be composed of several pages. For instance, a first page can show a map canvas, and a second page can show the attribute table associated with a layer, while a third one shows an HTML frame linking to your organization website. Or you can add many types of items on each page.

Adding a new page

Futhermore, a layout can be made using different size and/or orientation of pages. To add a page, select the  *Add Pages...* tool from the *Layout* menu or *Layout Toolbar*. The *Insert Pages* dialog opens and you are asked to fill:

- the number of pages to insert;
- the position of the page(s): before or after a given page or at the end of the print layout;
- The *Page size*: it could be of a preset format page (A4, B0, Legal, Letter, ANSI A, Arch A and their derivatives as well as a resolution type, such as 1920x1080 or 1024x768) with associated *Orientation* (Portrait or Landscape).

The page size can also be of a custom format; In that case, you'd need to enter its *Width* and *Height* (with locked size ratio if needed) and select the unit to use among mm, cm, px, pt, in, ft... Conversion of entered values is automatically applied when switching from one unit to another.

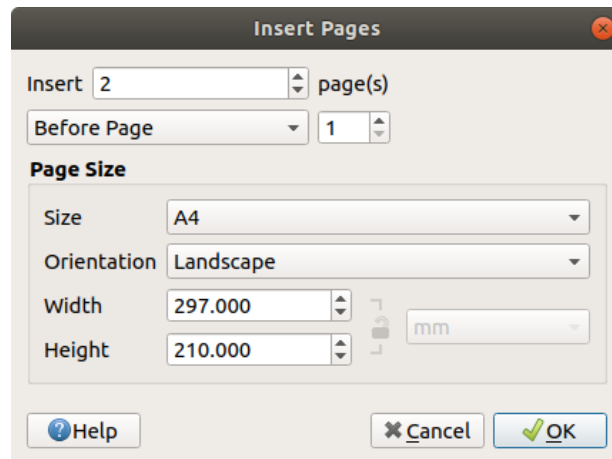


图 18.5: Creating a new page in the Print Layout

Updating page properties

Any page can be later customized through the *Page Item Properties* panel. Right-click on a page and select *Page Properties...*. The *Item Properties* panel opens with settings such as:

- the *Page size* frame described above. You can modify each property using the data defined override options (see *Explore Data-defined override buttons with atlas* for a use case);
- the *Exclude page from exports* to control whether the current page with its content should be included in the *layout output*;
- the *Background* of the current page using the *color* or *symbol* you want.

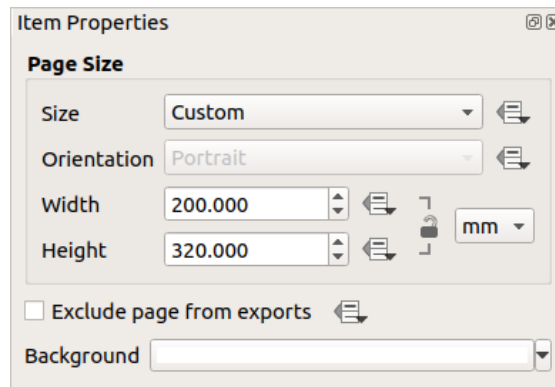


图 18.6: Page properties dialog

The Guides Panel

Guides are vertical or horizontal line references you can place on a layout page to assist you on items placement, when creating, moving or resizing them. To be active, guides require the *View ► Show Guides* and *View ► Snap to Guides* options to be checked. To create a guide, there are two different methods:

- if the *View ► Show Rulers* option is set, drag out a ruler and release the mouse button within the page area, at the desired position.
- for more precision, use the *Guides* panel from the *View ► Toolbox ►* or by selecting *Manage guides for page...* from the page's contextual menu.

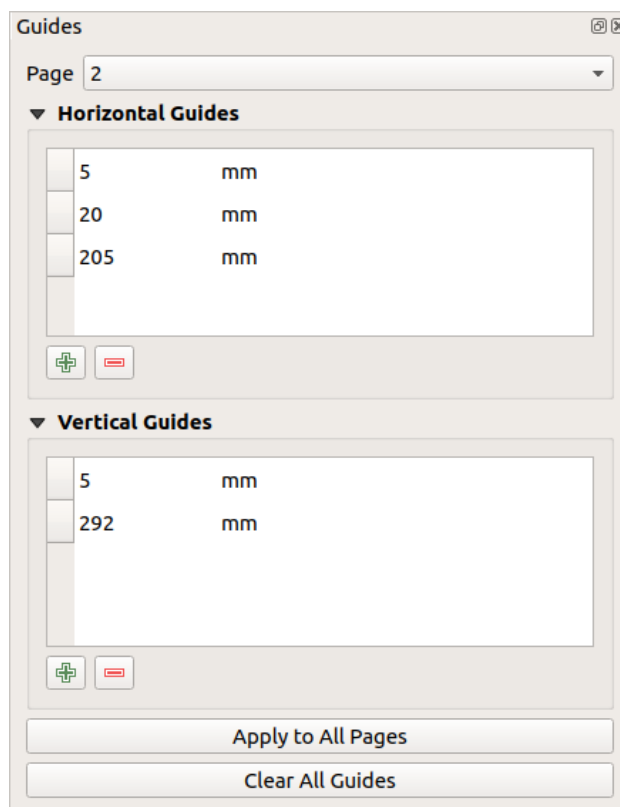




图 18.7: The Guides panel

The *Guides* panel allows creation of snap lines at specific locations:

1. Select the *Page* you'd like to add the guides to
2. Click the  *Add new guide* button and enter the coordinates of the horizontal or vertical line. The origin is at the top left corner. Different units are available for this.
The panel also allows adjusting the position of existing guides to exact coordinates: double-click and replace the value.
3. The *Guides* panel lists only the items for the current page. It allows creation or removal of guides only in the current page. However, you can use the *Apply to All Pages* button to replicate the guide configuration of the current page to the other pages in the layout.
4. To delete a guide, select it and press the  *Remove selected guide* button. Use *Clear All Guides* to remove all the guides in the current page.



小技巧: Snapping to existing layout items


Other than guides and grids, you can use existing items as snapping references when moving, resizing or creating new items; these are called **smart guides** and require *View ► Smart Guides* option to be checked. Anytime the mouse pointer is close to an item's bound, a snapping cross appears.

The Items Panel

The *Items* panel offers some options to manage selection and visibility of items. All the items added to the print layout canvas (including *items group*) are shown in a list and selecting an item makes the corresponding row selected in the list as well as selecting a row does select the corresponding item in the print layout canvas. This is thus a handy way to select an item placed behind another one. Note that a selected row is shown as bold.

For any selected item, you can :

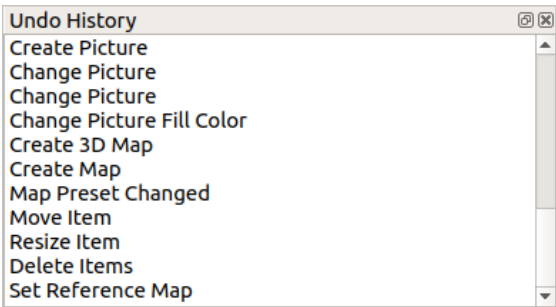
-  set it visible or not;
-  lock or unlock its position;
- sort its Z position. You can move up and down each item in the list with a click and drag. The upper item in the list will be brought to the foreground in the print layout canvas. By default, a newly created item is placed in the foreground.
- change the item ID by double-clicking the text;
- right-click an item and copy or delete it or open its *properties panel*.

Once you have found the correct position for an item, you can lock it by ticking the box in  column. Locked items are **not** selectable on the canvas. Locked items can be unlocked by selecting the item in the *Items* panel and unchecking the tickbox or you can use the icons on the toolbar.

The Undo History Panel: Revert and Restore actions

During the layout process, it is possible to revert and restore changes. This can be done with the revert and restore tools available in the *Edit* menu, the *Layout* toolbar or the contextual menu any time you right-click in the print layout area:

-  Revert last change
-  Restore last change

This can also be done by mouse click within the *Undo history* panel (see ). The History panel lists the last actions done within the print layout. Just select the point you want to revert to and once you do new action all the actions done after the selected one will be removed.

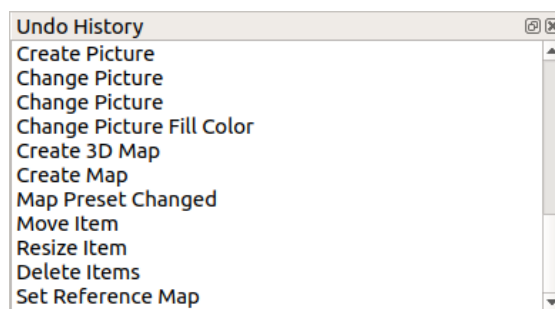


图 18.8: Undo History in the Print Layout

18.2 布局项

18.2.1 Layout Items Common Options

QGIS provides a large set of items to layout a map. They can be of map, legend, scale bar, picture, table, north arrow, image type... They however share some common options and behavior that are exposed below.

Creating a layout item

Items can be created using different tools, either from scratch or based on existing items.

To create a layout item from scratch:

1. Select the corresponding tool either from the *Add Item* menu or the *Toolbox* bar.
2. Then:
 - Click on the page and fill the size and placement information requested in the *New Item Properties* dialog that pops up (for details, see *Position and Size*);

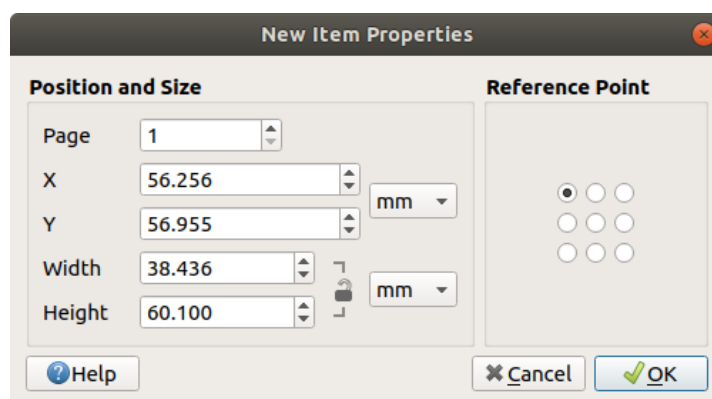



图 18.9: New Item properties dialog

- Or click-and-drag to define the initial size and placement of the item. You can rely on *grids and guides* snapping for a better position.

注解: Because they can have particular shapes, drawing node or arrow items does not work with one-click or click-and-drag methods; you need to click and place each node of the item. See [基于结点的形状项](#) for more details.

You can also:

1. Select an existing item with the  Select/Move item button from the *Toolbox* toolbar
2. Use the contextual menu or the *Edit* menu tools to copy/cut the item and paste it at the mouse position as a new item.


You can also use the *Paste in Place* (Ctrl+Shift+V) command to duplicate an item from one page to another and place it in the new page at the same coordinates as the original.


Moreover, you can create items using a print layout template (for details, see *The Layout Manager*) through the *Layout* ► *Add Items from Template...* command.

小技巧: Add layout items using the file browser

From your file browser or using the *Browser* panel, drag-and-drop a print layout template (.qpt file) onto a print layout dialog and QGIS automatically adds all items from that template to the layout.

Interacting with layout items

Each item inside the print layout can be moved and resized to create a perfect layout. For both operations the first step is to activate the  Select/Move item tool and click on the item.

You can select multiple items with the  Select/Move item button: click and drag over the items or hold the *Shift* button and click on each of the items you want. To deselect an item, click on it holding the *Shift* button.






Each time there's a selection, count of selected items is displayed on the status bar. Inside the *Edit* menu, you can find actions to select all the items, clear all selections, invert the current selection and more...

Moving and resizing items

Unless *View ► Show Bounding Boxes* option is unchecked, a selected item will show squares on its boundaries ; moving one of them with the mouse will resize the item in the corresponding direction. While resizing, holding *Shift* will maintain the aspect ratio. Holding *Alt* will resize from the item center.

To move a layout item, select it with the mouse and move while holding the left button. If you need to constrain the movements to the horizontal or vertical axis, just hold the *Shift* button on the keyboard while moving the mouse. You can also move a selected item using the *Arrow* keys on the keyboard; if the movement is too slow, you can speed it up by holding *Shift*. If you need better precision, use the *Position and size* properties, or grid/guides snapping as explained above for item's creation.

Resizing or moving several items at once is made the same way as for a single item. QGIS however provides some advanced tools to automatically resize a selection of items following different rules:


- each item height matches the  tallest or the  shortest selected item;
- each item width matches the  widest or the  narrowest selected item;
- resizes items to  squares: each item is enlarged to form a square.

Likewise, automated tools are available to organize multiple items position by distributing equidistantly:


- edges (left, right, top or bottom) of items;
- centers of items either horizontally or vertically.

Grouping items

Grouping items allows you to manipulate a set of items like a single one: you can easily resize, move, delete, copy the items as a whole.


To create a group of items, select more than one and press the  *Group* button on the *View* menu or the *Actions* toolbar or from the right-click menu. A row named *Group* is added to the *Items* panel and can be locked or hidden like any other *Items panel's object*. Grouped items are **not individually** selectable on the canvas; use the *Items* panel for direct selection and access the item's properties panel.

Locking items

Once you have found the correct position for an item, you can lock it by using the  *Lock selected items* button in the *Items* menu or the *Actions* toolbar or ticking the box next to the item in the *Items* panel. Locked items are **not** selectable on the canvas.

Locked items can be unlocked by selecting the item in the *Items* panel and unchecking the tickbox or you can use the icons on the toolbar.

Alignment

Raising or lowering the visual hierarchy for elements are inside the  *Raise selected items* pull-down menu. Choose an element on the print layout canvas and select the matching functionality to raise or lower the selected element compared to the other elements. This order is shown in the *Items* panel. You can also raise or lower objects in the *Items* panel by clicking and dragging an object's label in this list.

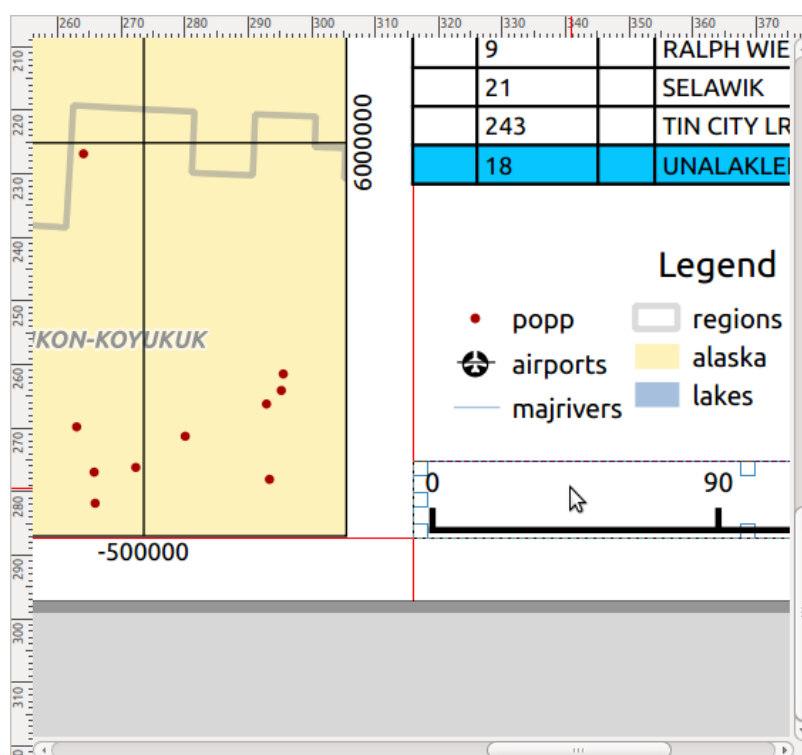









图 18.10: Alignment helper lines in the print layout

There are several alignment options available within the  *Align selected items* pull-down menu (see 图 18.10). To use an alignment function, you first select the elements and then click on one of the alignment icons:

-  *Align Left* or  *Align Right*;
-  *Align Top* or  *Align Bottom*;
-  *Align Center horizontally* or  *Align Center Vertical*.

All selected elements will then be aligned to their common bounding box. When moving items on the layout canvas, alignment helper lines appear when borders, centers or corners are aligned.

Items Common Properties

Layout items have a set of common properties you will find at the bottom of the *Item Properties* panel: Position and size, Rotation, Frame, Background, Item ID, Variables and Rendering (See [图 18.11](#)).

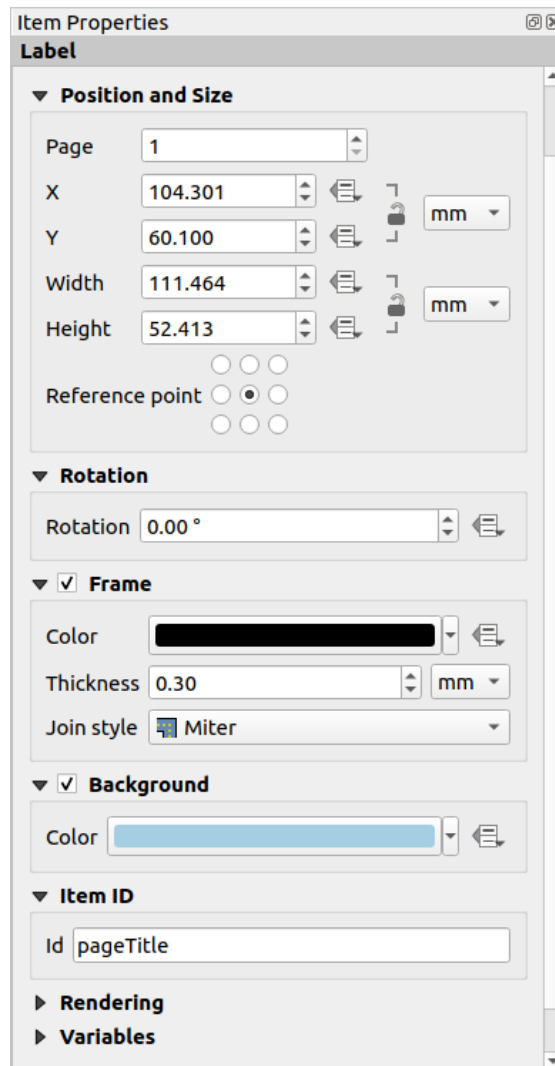




图 18.11: Common Item Properties groups

注解: The  Data defined override icon next to most of the options means that you can associate that property with a layer, features attributes, geometry or with any other layout item's property, using *expressions* or *variables*. For more information see [Data defined override setup](#).

- The *Position and size* group lets you define the size and position of the frame which contains the item (see [Position and Size](#) for more information).
- The *Rotation* sets the rotation of the element (in degrees).
- The  *Frame* shows or hides the frame around the item. Use the *Color*, *Thickness* and *Join style* widgets to adjust those properties.
- Use the *Background color* menu for setting a background color. Click on the [Color...] button to display a dialog where you can pick a color or choose from a custom setting. Transparency can be adjusted through altering the alpha field settings.

- Use the *Item ID* to create a relationship to other print layout items. This is used with QGIS server and other potential web clients. You can set an ID on an item (for example, a map or a label), and then the web client can send data to set a property (e.g., label text) for that specific item. The `GetProjectSettings` command will list the items and IDs which are available in a layout.
- *Rendering* mode helps you set whether and how the item can be displayed: you can, for instance, apply *blending mode*, adjust the opacity of the item or *Exclude item from exports*.

Position and Size

Extending the features of the *New Item Properties* dialog with data-defined capabilities, this group allows you to place the items accurately.

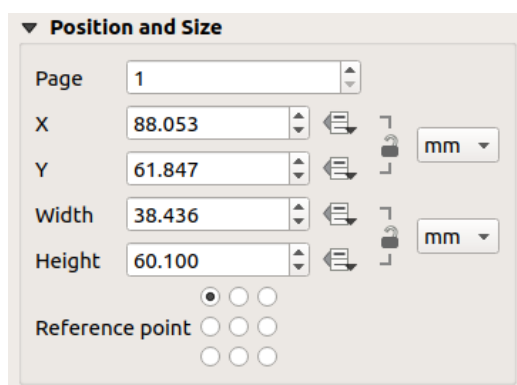




图 18.12: Position and size

- the actual number of the page to place the item on;
- the reference point of the item;
- the *X* and *Y* coordinates of the *Reference point* of the item on the chosen page. The ratio between these values can be locked by clicking on the  button. Changes made to a value using the widget or the  Select/Move item tool will be reflected in both of them;
- the *Width* and *Height* of the item bounding box. As for coordinates, the ratio between width and height can be locked.

Rendering mode

QGIS allows advanced rendering for layout items just like vector and raster layers.

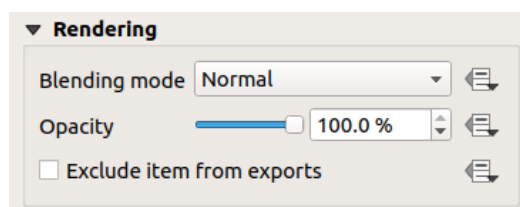





图 18.13: Rendering mode

- *Blending mode*: With this tool you can achieve effects which would otherwise only be achieved using graphic rendering software. The pixels of your overlaying and underlying items can be mixed according to the mode set (see *Blending Modes* for description of each effect).

- **Transparency** : You can make the underlying item in the layout visible with this tool. Use the slider to adapt the visibility of your item to your needs. You can also make a precise definition of the percentage of visibility in the menu beside the slider.
- **Exclude item from exports**: You can decide to make an item invisible in all exports. After activating this checkbox, the item will not be included in export to PDF, print etc..


Variables

The *Variables* lists all the variables available at the layout item's level (which includes all global, project and composition's variables). Map items also include Map settings variables that provide easy access to values like the map's scale, extent, and so on.

In *Variables*, it's also possible to manage item-level variables. Click the  button to add a new custom variable. Likewise, select any custom item-level variable from the list and click the  button to remove it.

More information on variables usage in the *Storing values in Variables* section.

18.2.2 The Map Item

The map item is the main frame that displays the map you've designed in the map canvas. Use the  *Add Map* tool following *items creation instructions* to add a new map item that you can later manipulate the same way as exposed in *Interacting with layout items*.

By default, a new map item shows the current status of the *map canvas* with its extent and visible layers. You can customize it thanks to the *Item Properties* panel. Other than the *items common properties*, this feature has the following functionalities:

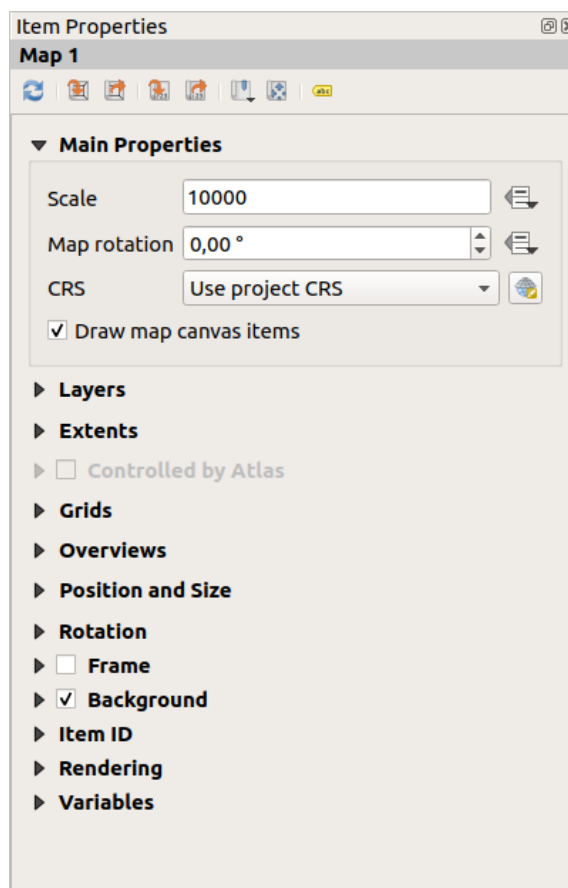











图 18.14: Map Item Properties Panel

The Toolbar


The Map *Item Properties* panel embeds a toolbar with the following functionalities:

-  Update map preview
-  Set map canvas to match main canvas extent
-  View current map extent in main canvas
-  Set map scale to match main canvas scale
-  Set main canvas to match current map scale
-  Bookmarks: set the map item extent to match an existing spatial bookmark
-  Interactively edit map extent: pan and zoom interactively within the map item
-  Labeling settings: control feature label behaviour (placement, visibility...) in the layout map item extent:
 - set a *Margin from map edges*, a data definable distance from the map item's limits inside which no label should be displayed
 -  *Allow truncated labels on edges of map*: controls whether labels which fall partially outside of the map item allowed extent should be rendered. If checked, these labels will be shown (when there's no way to place them fully within the visible area). If unchecked then partially visible labels will be skipped.
 - *Label blocking items*: allows other layout items (such as scalebars, north arrows, inset maps, etc) to be marked as a blockers for the map labels in the **active** map item. This prevents any map labels from being

placed under those items - causing the labeling engine to either try alternative placement for these labels or discard them altogether.

If a *Margin from map edges* is set, the map labels are not placed closer than the specified distance from the checked layout items.

- *Show unplaced labels*: can be used to determine whether labels are missing from the layout map (e.g. due to conflicts with other map labels or due to insufficient space to place the label) by highlighting them in a *predefined color*.

-  Clipping settings: allows to clip the map item to the atlas feature and to shape and polygon items:

- *Clip to atlas feature*: you can determine that the layout map item will be clipped automatically to the current *atlas feature*.

There are different clipping modes available:

- * *Clip During Render Only*: applies a painter based clip, so that portions of vector features which sit outside the atlas feature become invisible
- * *Clip Feature Before Render*: applies the clip before rendering features, so borders of features which fall partially outside the atlas feature will still be visible on the boundary of the atlas feature
- * *Render Intersecting Features Unchanged*: renders all features which intersect the current atlas feature, but without clipping their their geometry.

You can *Force labels inside atlas feature*. If you don't want to *Clip all layers* to the atlas feature you can use the *Clip selected layers* option.

- *Clip to item*: it is possible to change the shape of the map item by using a *shape* or *polygon* item from the print layout. When you enable this option the map will be automatically clipped to the selected shape in the combobox. Again, the above mentioned clipping modes are available and labels can be forced to display only inside the clipping shape.

Main properties

In the *Main properties* group (see [图 18.14](#)) of the map *Item Properties* panel, available options are:

- The *Update Preview* button to refresh the map item rendering if the view in map canvas has been modified. Note that most of the time, the map item refresh is automatically triggered by the changes;
- The *Scale* to manually set the map item scale;
- The *Map rotation* allows you to rotate the map item content clockwise in degrees. The rotation of the map canvas can be imitated here;
- The *CRS* allows you to display the map item content in any *CRS*. It defaults to `Use project CRS`;
- *Draw map canvas items* lets you show in the print layout *annotations* that are placed on the main map canvas.

Layers

By default, map item appearance is synced with the map canvas rendering meaning that toggling visibility of the layers or modifying their style in the *Layers Panel* is automatically applied to the map item. Because, like any other item, you may want to add multiple map items to a print layout, there's a need to break this synchronization in order to allow showing different areas, layer combinations, at different scales... The *Layers* properties group (see [图 18.15](#)) helps you do that.

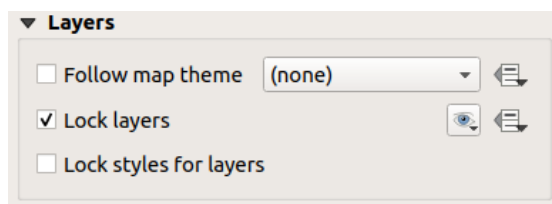


图 18.15: Map Layers group

If you want to keep the map item consistent with an existing *map theme*, check *Follow map theme* and select the desired theme in the drop-down list. Any changes applied to the theme in QGIS' main window (using the replace theme function) will automatically affect the map item. If a map theme is selected, the *Lock styles for layers* option is disabled because *Follow map theme* also updates the style (symbology, labels, diagrams) of the layers.

To lock the layers shown in a map item to the current map canvas visibility, check *Lock layers*. When this option is enabled, any changes on the layers' visibility in QGIS' main window will not affect the layout's map item. Nevertheless, style and labels of locked layers are still refreshed according to QGIS' main window. You can prevent this by using *Lock styles for layers*.

Instead of using the current map canvas, you can also lock the layers of the map item to those of an existing map theme: select a map theme from the Set layer list from a map theme drop-down button, and the *Lock layers* is activated. The set of visible layers in the map theme is from now on used for the map item until you select another map theme or uncheck the *Lock layers* option. You then may need to refresh the view using the Refresh view button of the *Navigation* toolbar or the *Update Preview* button seen above.

Note that, unlike the *Follow map theme* option, if the *Lock layers* option is enabled and set to a map theme, the layers in the map item will not be refreshed even if the map theme is updated (using the replace theme function) in QGIS' main window.

Locked layers in the map item can also be *data-defined*, using the icon beside the option. When used, this overrides the selection set in the drop-down list. You need to pass a list of layers separated by | character. The following example locks the map item to use only layers *layer 1* and *layer 2*:

```
concat ('layer 1', '|', 'layer 2')
```

Extents

The *Extents* group of the map item properties panel provides the following functionalities (see 图 18.16):

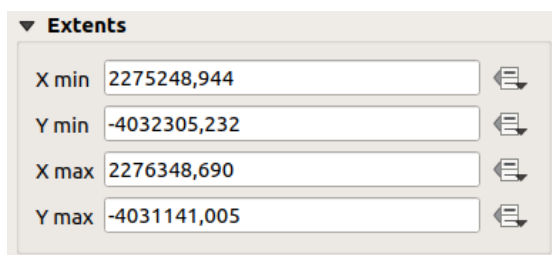




图 18.16: Map Extents group

The **Extents** area displays X and Y coordinates of the area shown in the map item. Each of these values can be manually replaced, modifying the map canvas area displayed and/or map item size. Clicking the *Set to Map Canvas Extent* button sets the extent of the layout map item to the extent of the main map canvas. The button *View Extent in Map Canvas* does exactly the opposite; it updates the extent of the main map canvas to the extent of the layout map item.

You can also alter a map item extent using the  Move item content tool: click-and-drag within the map item to modify its current view, keeping the same scale. With the  tool enabled, use the mouse wheel to zoom in or out, modifying the scale of the shown map. Combine the movement with `Ctrl` key pressed to have a smaller zoom.





Controlled by atlas

The *Controlled by atlas* group properties is available only if an *atlas* is active in the print layout. Check this option if you want the map item being ruled by the atlas; when iterating over the coverage layer, the map item extent is panned/zoomed to the atlas feature following:

- *Margin around features*: zooms to the feature at the best scale, keeping around each a margin representing a percentage of the map item width or height. The margin can be the same for all features or *set variable*, e.g., depending on map scale;
- *Predefined scale (best fit)*: zooms to the feature at the project *predefined scale* where the atlas feature best fits;
- *Fixed scale*: atlas features are panned from one to another, keeping the same scale of the map item. Ideal when working with features of same size (e.g., a grid) or willing to highlight size differences among atlas features.

Grids

With grids, you can add, over your map, information relative to its extent or coordinates, either in the map item projection or a different one. The *Grids* group provides the possibility to add several grids to a map item.

- With the  and  buttons you can add or remove a selected grid;
- With the  and  buttons you can move up and down a grid in the list, hence move it on top or bottom of another one, over the map item.

Double-click the added grid to rename it.

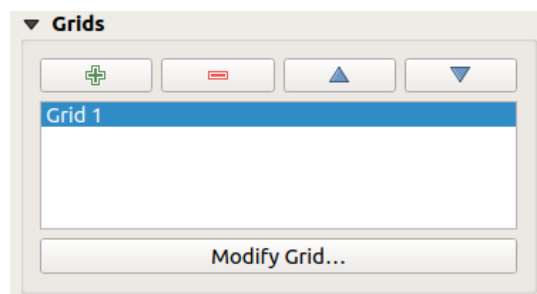


图 18.17: Map Grids Dialog

To modify a grid, select it and press the *Modify Grid...* button to open the *Map Grid Properties* panel and access its configuration options.

Grid Appearance

In the *Map Grid Properties* panel, check *Grid enabled* to show the grid on the map item.

As grid type, you can specify to use a:

- *Solid*: shows a line across the grid frame. The *Line style* can be customized using *color* and *symbol* selector widget;
- *Cross*: displays segment at the grid lines intersection for which you can set the *Line style* and the *Cross width*;
- *Markers*: only displays customizable markers symbol at grid lines intersection;
- or *Frame and annotations only*.

Other than the grid type, you can define:

- the *CRS* of the grid. If not changed, it will follow the Map CRS. The *Change* button lets you set it to a different CRS. Once set, it can be changed back to default by selecting any group heading (e.g **Geographic Coordinate System**) under *Predefined Coordinate Reference Systems* in the CRS selection dialog.
- the *Interval* type to use for the grid references. Available options are *Map Unit*, *Fit Segment Width*, *Millimeter* or *Centimeter*:
 - choosing *Fit Segment Width* will dynamically select the grid interval based on the map extent to a "pretty" interval. When selected, the *Minimum* and *Maximum* intervals can be set.
 - the other options allow you to set the distance between two consecutive grid references in the X and Y directions.
- the *Offset* from the map item edges, in the X and/or the Y direction
- and the *Blend mode* of the grid (see *Blending Modes*) when compatible.

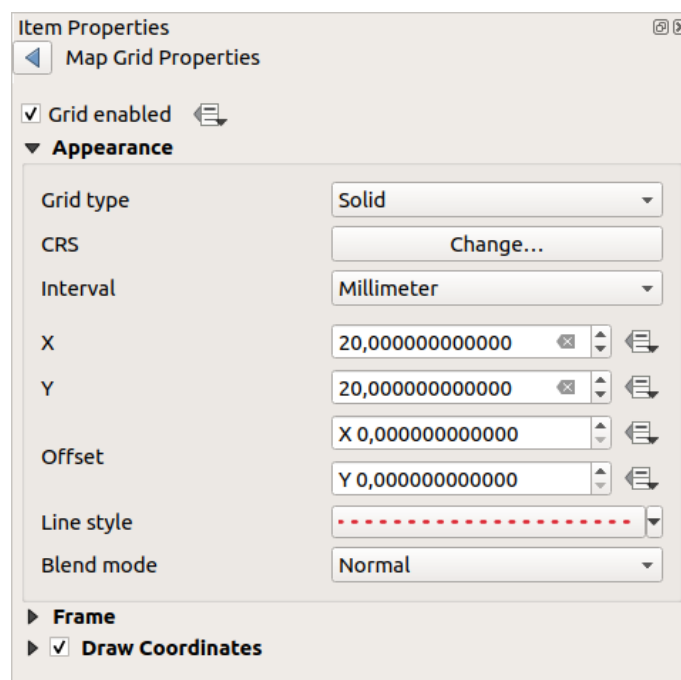


图 18.18: Grid Appearance Dialog

Grid Frame

There are different options to style the frame that holds the map. The following options are available: No Frame, Zebra, Zebra (nautical), Interior ticks, Exterior ticks, Interior and Exterior ticks, Line border and Line border (nautical).

When compatible, it's possible to set the *Frame size*, a *Frame margin*, the *Frame line thickness* with associated color and the *Frame fill colors*.

Using Latitude/Y only and Longitude/X only values in the divisions section you can prevent a mix of latitude/Y and longitude/X coordinates showing on each side when working with rotated maps or reprojected grids. Also you can choose to set visible or not each side of the grid frame.

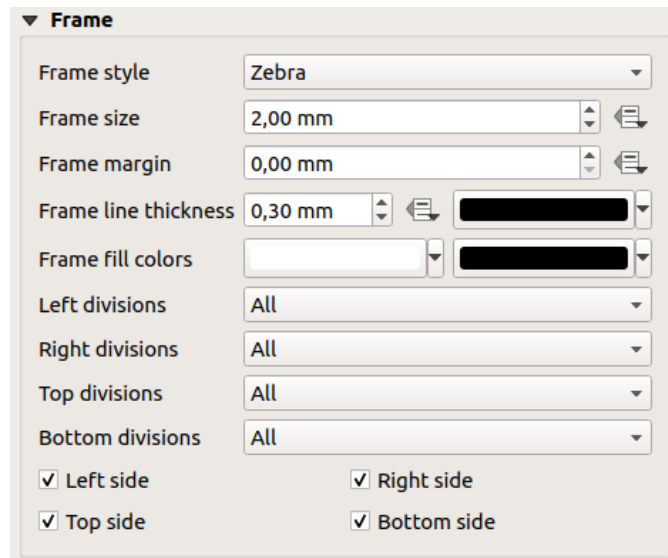


图 18.19: Grid Frame Dialog

Coordinates

The *Draw coordinates* checkbox allows you to add coordinates to the map frame. You can choose the annotation numeric format, the options range from decimal to degrees, minute and seconds, with or without suffix, aligned or not and a custom format using the expression dialog.

You can choose which annotation to show. The options are: show all, latitude only, longitude only, or disable(none). This is useful when the map is rotated. The annotation can be drawn inside or outside the map frame. The annotation direction can be defined as horizontal, vertical ascending or vertical descending.

Finally, you can define the annotation font, font color, distance from the map frame and the precision of the drawn coordinates.

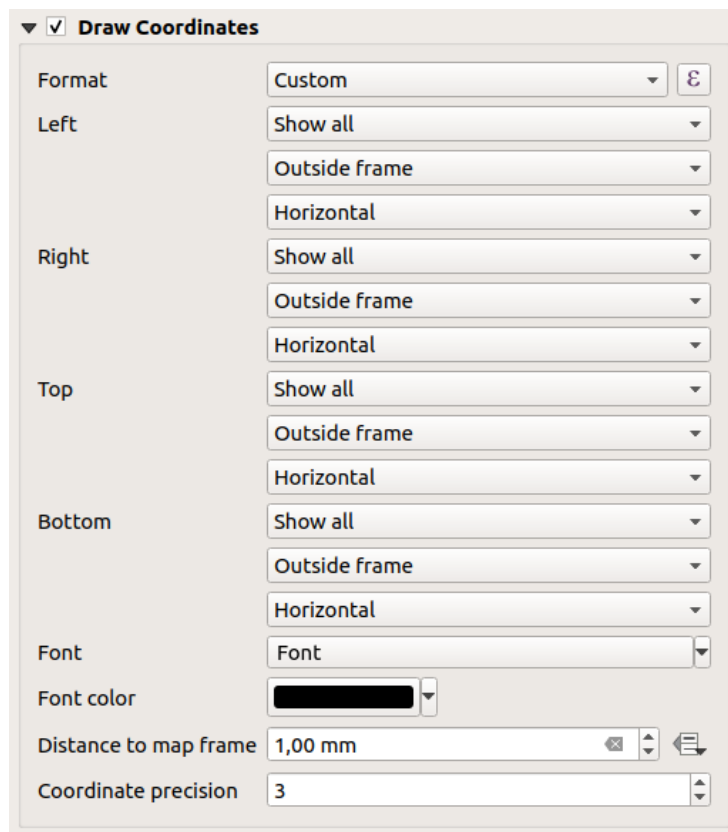


图 18.20: Grid Draw Coordinates dialog

Overviews

Sometimes you may have more than one map in the print layout and would like to locate the study area of one map item on another one. This could be for example to help map readers identify the area in relation with its larger geographic context shown in the second map.

The *Overviews* group of the map panel helps you create the link between two different maps extent and provides the following functionalities:

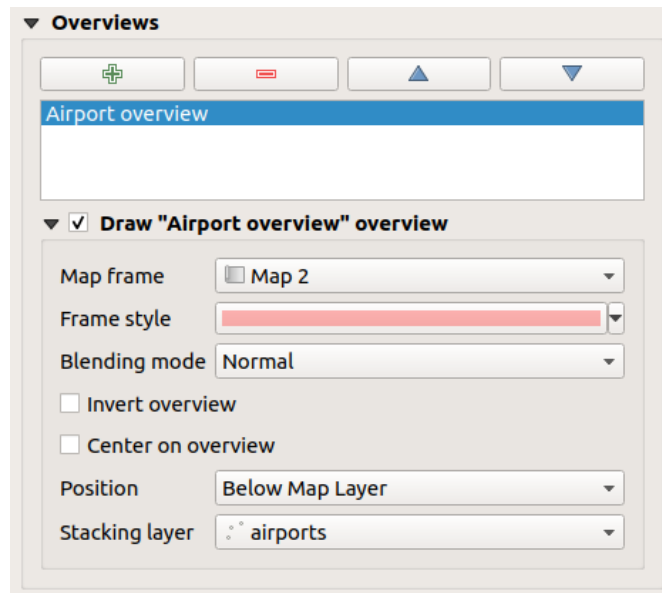


图 18.21: Map Overviews group

To create an overview, select the map item on which you want to show the other map item's extent and expand the *Overviews* option in the *Item Properties* panel. Then press the button to add an overview.


Initially this overview is named 'Overview 1' (see 图 18.21). You can:

- Rename it with a double-click
- With the and buttons, add or remove overviews
- With the and buttons, move an overview up and down in the list, placing it above or below other overviews in the map item (when they are at the same *stack position*).

Then select the overview item in the list and check the *Draw "<name_overview>" overview* to enable the drawing of the overview on the selected map frame. You can customize it with:

- The *Map frame* selects the map item whose extents will be shown on the present map item.
- The *Frame Style* uses the *symbol properties* to render the overview frame.
- The *Blending mode* allows you to set different transparency blend modes.
- The *Invert overview* creates a mask around the extents when activated: the referenced map extents are shown clearly, whereas the rest of the map item is blended with the frame fill color (if a fill color is used).
- The *Center on overview* pans the map item content so that the overview frame is displayed at the center of the map. You can only use one overview item to center, when you have several overviews.
- The *Position* controls exactly where in the map item's layer stack the overview will be placed, e.g. allowing an overview extent to be drawn below some feature layers such as roads whilst drawing it above other background layers. Available options are:
 - *Below map*
 - *Below map layer* and *Above map layer*: place the overview frame below and above the geometries of a layer, respectively. The layer is selected in the *Stacking layer* option.
 - *Below map labels*: given that labels are always rendered above all the feature geometries in a map item, places the overview frame above all the geometries and below any label.
 - *Above map labels*: places the overview frame above all the geometries and labels in the map item.

18.2.3 The 3D Map Item

The 3D Map item is used to display a *3D map view*. Use the  *Add 3D Map* button, and follow *items creation instructions* to add a new 3D Map item that you can later manipulate the same way as demonstrated in *Interacting with layout items*.

By default, a new 3D Map item is empty. You can set the properties of the 3D view and customize it in the *Item Properties* panel. In addition to the *common properties*, this feature has the following functionalities (图 18.22):

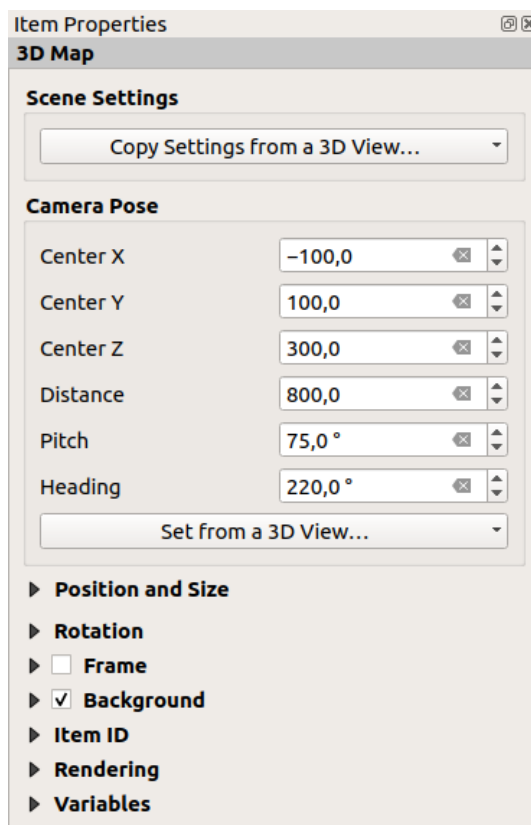


图 18.22: 3D Map Item Properties

Scene settings

Press *Copy Settings from a 3D View...* to choose the 3D map view to display.

The 3D map view is rendered with its current configuration (layers, terrain, lights, camera position and angle...).


Camera pose

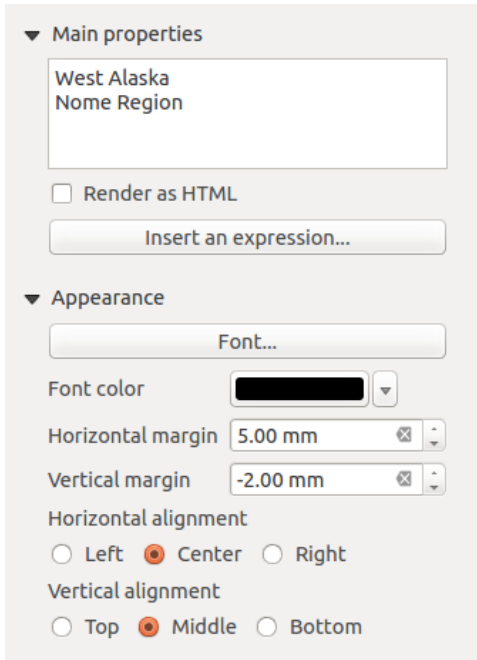
- *Center X* sets the X coordinate of the point the camera is pointing at
- *Center Y* sets the Y coordinate of the point the camera is pointing at
- *Center Z* sets the Z coordinate of the point the camera is pointing at
- *Distance* sets the distance from the camera center to the point the camera is pointing at
- *Pitch* sets the rotation of the camera around the X-axis (vertical rotation). Values from 0 to 360 (degrees). 0°: terrain seen straight from above; 90°: horizontal (from the side); 180°: straight from below; 270°: horizontal, upside down; 360°: straight from above.

- *Heading* sets the rotation of the camera around the Y-axis (horizontal rotation - 0 to 360 degrees). 0°/360°: north; 90°: west; 180°: south; 270°: east.

The *Set from a 3D View...* pull-down menu lets you populate the items with the parameters of a 3D View.

18.2.4 The Label Item

The *Label* item is a tool that helps decorate your map with texts that would help to understand it; it can be the title, author, data sources or any other information... You can add a label with the  *Add Label* tool following *items creation instructions* and manipulate it the same way as exposed in *Interacting with layout items*.

By default, the label item provides a default text that you can customize using its *Item Properties* panel. Other than the *items common properties*, this feature has the following functionalities (see  18.23):

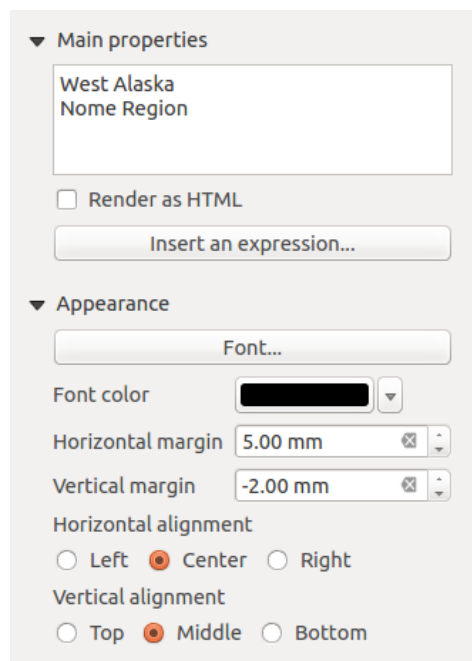



图 18.23: Label Item Properties Panel

Main properties

The *Main properties* group is the place to provide the text (it can be in HTML) or the expression to build the label. Expressions need to be surrounded by [% and %] in order to be interpreted as such.

- Labels can be interpreted as HTML code: check *Render as HTML*. You can now insert a URL, a clickable image that links to a web page or something more complex.
- You can also use *expressions*: click on *Insert or Edit an expression...* button, write your formula as usual and when the dialog is applied, QGIS automatically adds the surrounding characters.

注解: Clicking the *Insert or Edit an Expression...* button when no selection is made in the textbox will append the new expression to the existing text. If you want to update an existing text, you need to select it the part of interest beforehand.

You can combine HTML rendering with expressions, leading to advanced labeling. The following code will output  18.24:

```

<html>
<head>
  <style>
    /* Define some custom styles, with attribute-based size */
    name {color:red; font-size: [% ID %]px; font-family: Verdana; text-shadow:
↪grey 1px 0 10px;}
    use {color:blue;}
  </style>
</head>

<body>
  <!-- Information to display -->
  <u>Feature Information</u>
  <ul style="list-style-type:disc">
    <li>Feature Id: [% ID %]</li>
    <li>Airport: <name>[% NAME %]</name></li>
    <li>Main use: <use>[% USE %]</use></li>
  </ul>
  Last check: [% concat( format_date( "control_date", 'yyyy-MM-dd'), ' by <b><i>',
↪ @user_full_name, '</i></b>' ) %]

  <!-- Insert an image -->
  <p align=center>
</body>
</html>

```



图 18.24: Leveraging a label with HTML styling

Appearance

- Define *Font* by clicking on the *Font...* button or a *Font color* by pushing the *color widget*.
- You can specify different horizontal and vertical margins in mm. This is the margin from the edge of the layout item. The label can be positioned outside the bounds of the label e.g. to align label items with other items. In this case you have to use negative values for the margin.
- Using the text alignment is another way to position your label. It can be:
 - *Left, Center, Right* or *Justify* for *Horizontal alignment*
 - and *Top, Middle, Bottom* for *Vertical alignment*.

Exploring expressions in a label item

Below some examples of expressions you can use to populate the label item with interesting information - remember that the code, or at least the calculated part, should be surrounded by [% and %] in the *Main properties* frame:

- Display a title with the current atlas feature value in "field1":

```
'This is the map for ' || "field1"
```

or, written in the *Main properties* section:

```
This is the map for [% "field1" %]
```

- Add a pagination for processed atlas features (eg, Page 1/10):

```
concat( 'Page ', @atlas_featurenumber, '/', @atlas_totalfeatures )
```

- Return the name of the airports of the current atlas region feature, based on their common attributes:

```
aggregate( layer := 'airports',
  aggregate := 'concatenate',
  expression := "NAME",
  filter := fk_regionId = attribute( @atlas_feature, 'ID' ),
  concatenator := ', '
)
```

Or, if an *attributes relation* is set:

```
relation_aggregate( relation := 'airports_in_region_relation',
  aggregate := 'concatenate',
  expression := "NAME",
  concatenator := ', '
)
```

- Return the name of the airports contained in the current atlas region feature, based on their spatial relationship:

```
aggregate( layer := 'airports',
  aggregate := 'concatenate',
  expression := "NAME",
  filter := contains( geometry( @parent ), $geometry ),
  concatenator := ', '
)
```

OR:

```
array_to_string( array:= overlay_contains( layer := 'airports',
  expression := "NAME" ),
  delimiter:= ', '
)
```

- Return the lower X coordinate of the Map 1 item's extent:

```
x_min( map_get( item_variables( 'Map 1' ), 'map_extent' ) )
```


- Retrieve the name of the layers in the current layout Map 1 item, and formats in one name by line:

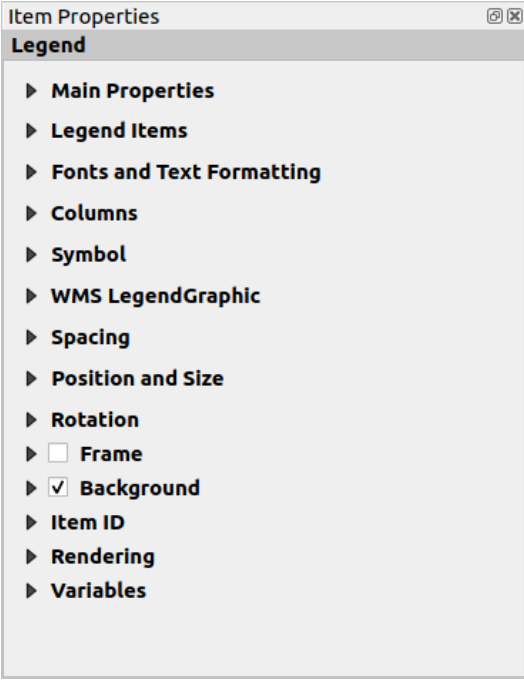
```
array_to_string(
  array_foreach(
    map_get( item_variables( 'Map 1' ), 'map_layers' ), -- retrieve the layers_
    →list
    layer_property( @element, 'name' ) -- retrieve each layer name
  ),
```

(下页继续)

```
'\n' -- converts the list to string separated by breaklines
)
```

18.2.5 The Legend Item

The *Legend* item is a box or a table that explains the meanings of the symbols used on the map. A legend is then bound to a map item. You can add a legend item with the  *Add Legend* tool following *items creation instructions* and manipulate it the same way as exposed in *Interacting with layout items*.

By default, the legend item displays all available layers and can be refined using its *Item Properties* panel. Other than the *items common properties*, this feature has the following functionalities (see  18.25):

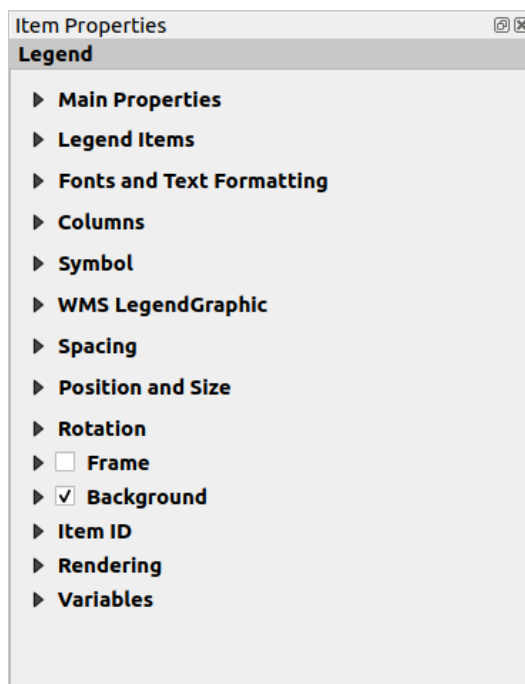
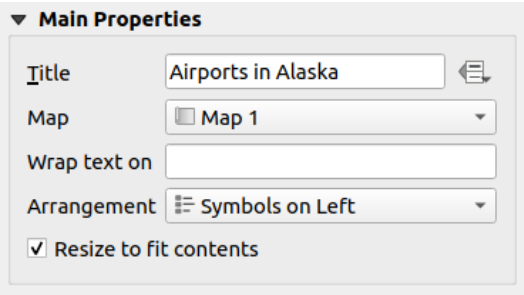


图 18.25: Legend Item Properties Panel

Main properties

The *Main properties* group of the legend *Item Properties* panel provides the following functionalities (see  18.26):

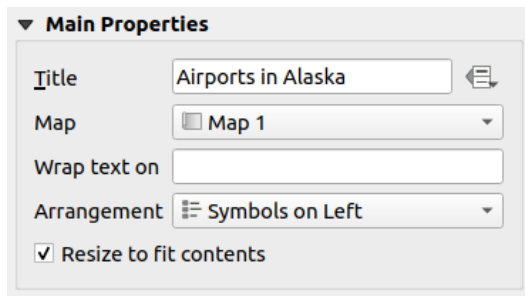


图 18.26: Legend Main properties group

In Main properties you can:

- Change the *Title* of the legend. It can be made dynamic using the *data-defined override* setting, useful for example when generating an atlas;
- Choose which *Map* item the current legend will refer to. By default, the map over which the legend item is drawn is picked. If none, then it falls back to the *reference map*.

注解: *Variables* of the linked map item (@map_id, @map_scale, @map_extent...) are also accessible from data-defined properties of the legend.

- Wrap the text of the legend on a given character: each time the character appears, it's replaced with a line break;
- Set the symbols and text placement in the legend: the *Arrangement* can be *Symbols on left* or *Symbols on right*. The default value depends on the locale in use (right-to-left based or not).
- Use *Resize to fit contents* to control whether or not a legend should be automatically resized to fit its contents. If unchecked, then the legend will never resize and instead just stick to whatever size the user has set. Any content which doesn't fit the size is cropped out.

Legend items

The *Legend items* group of the legend *Item Properties* panel provides the following functionalities (see 图 18.27):

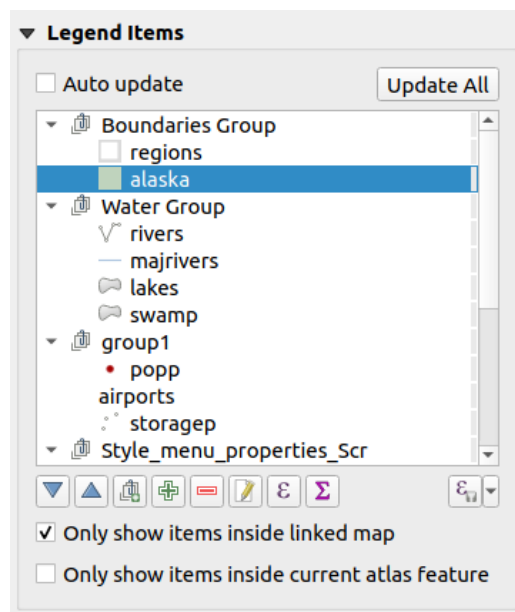






图 18.27: Legend Items group

- The legend will be updated automatically if *Auto update* is checked. When *Auto update* is unchecked this will give you more control over the legend items. All the icons below the legend items list will be activated.
- The legend items window lists all legend items and allows you to change item order, group layers, remove and restore items in the list, edit layer names and add a filter.
 - The item order can be changed using the and buttons or with 'drag-and-drop' functionality. The order can not be changed for WMS legend graphics.
 - Use the button to add a legend group.
 - Use the button to add layers and button to remove groups, layers or symbol classes.


- The  button is used to edit the layer, group name or title. First you need to select the legend item. Double-clicking the item also opens the text box to rename it.
- The  button uses expressions to customize each symbol label of the selected layer (see *Data-define the legend labels*)
- The  button adds a feature count for each class of vector layer.
- The  Filter legend by expression helps you filter which of the legend items of a layer will be displayed, i.e. using a layer that has different legend items (e.g., from a rule-based or categorized symbology), you can specify a boolean expression to remove from the legend tree, styles that have no feature satisfying a condition. Note that the features are nevertheless kept and shown in the layout map item.

While the default behavior of the legend item is to mimic the *Layers* panel tree, displaying the same groups, layers and classes of symbology, right-click any item offers you options to hide layer's name or raise it as a group or subgroup. In case you have made some changes to a layer, you can revert them by choosing *Reset to defaults* from the contextual menu of the legend entry.


After changing the symbology in the QGIS main window, you can click on *Update All* to adapt the changes in the legend element of the print layout.

- With the *Only show items inside linked map*, only the legend items visible in the linked map will be listed in the legend. This tool remains available when *Auto-update* is active
- While generating an atlas with polygon features, you can filter out legend items that lie outside the current atlas feature. To do that, check the *Only show items inside current atlas feature* option.

Data-define the legend labels

 allows you to add *expressions* to each symbol label of a given layer. New variables (@symbol_label, @symbol_id and @symbol_count) help you interact with the legend entry.

For example, given a *regions* layer categorized by its *type* field, you can append to each class in the legend their number of features and total area, e.g. *Borough (3) - 850ha*:

1. Select the layer entry in the legend tree
2. Press the  button, opening the *Expression String Builder* dialog
3. Enter the following expression (*assuming symbol labels have not been edited*):

```
concat ( @symbol_label,
         ' (', @symbol_count, ') - ',
         round( aggregate(@layer, 'sum', $area, filter:= "type"=@symbol_label)/
         ↪10000 ),
         'ha'
       )
```

4. Press *OK*

Fonts

The *Fonts* group of the legend *Item Properties* panel provides the following functionalities:

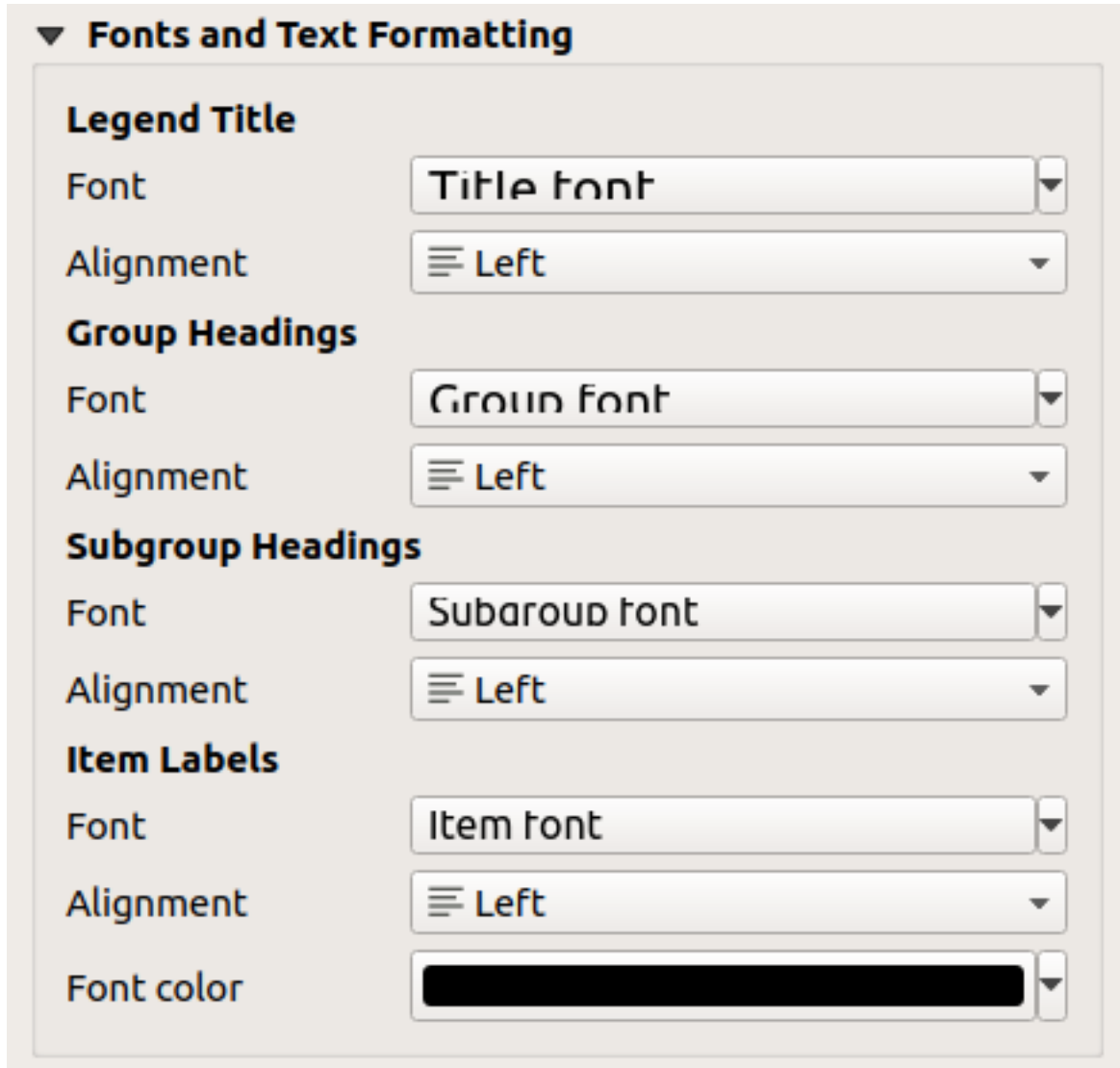


图 18.28: Legend Fonts properties

- You can change the font of the legend title, group, subgroup and item (feature) in the legend item using the *font selector* widget
- For each of these levels you can set the text *Alignment*: it can be *Left* (default for left-to-right based locales), *Center* or *Right* (default for right-to-left based locales).
- You set the *Color* of the labels using the *color selector* widget. The selected color will apply to all the font items in the legend.

Columns

Under the *Columns* group of the legend *Item Properties* panel, legend items can be arranged over several columns:

- Set the number of columns in the *Count* field. This value can be made dynamic e.g., following atlas features, legend contents, the frame size...
- *Equal column widths* sets how legend columns should be adjusted.
- The *Split layers* option allows a categorized or a graduated layer legend to be divided between columns.

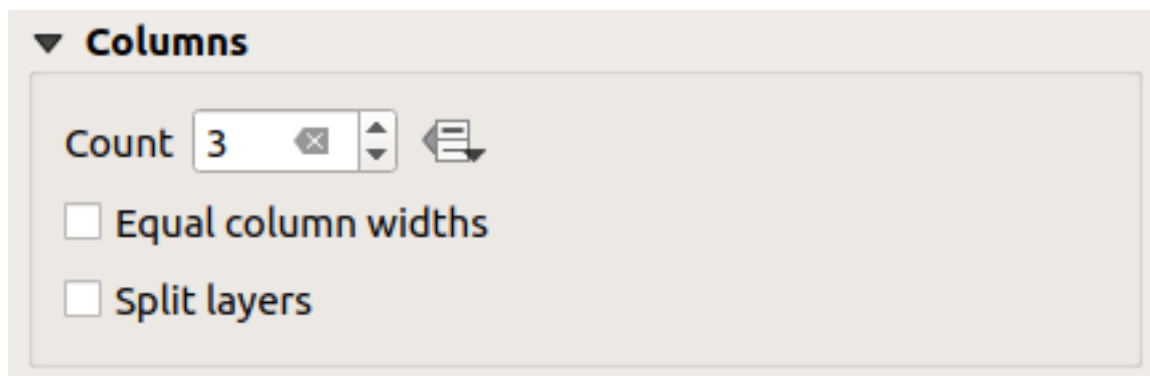


图 18.29: Legend Columns settings

Symbol

The *Symbol* group of the legend *Item Properties* panel configures the size of symbols displayed next to the legend labels. You can:

- Set the *Symbol width* and *Symbol height*
- Set the markers' *Min symbol size* and *Max symbol size*: 0.00mm means there is no value set.
- *Draw stroke for raster symbols*: this adds an outline to the symbol representing the band color of the raster layer; you can set both the *Stroke color* and *Tickness*.

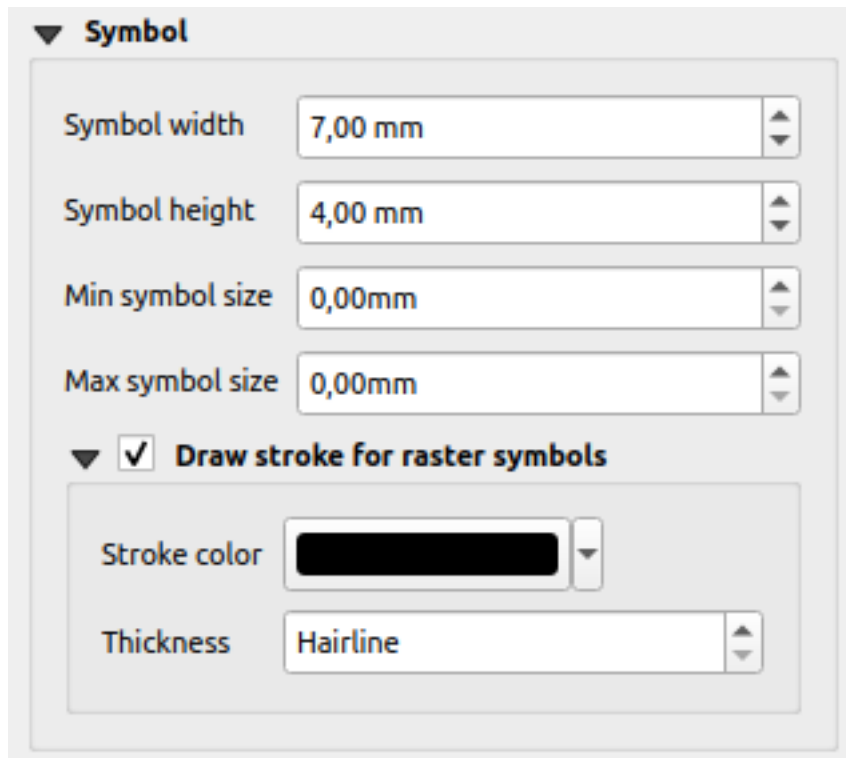


图 18.30: Legend Symbol configuration

WMS LegendGraphic and Spacing

The *WMS LegendGraphic* and *Spacing* groups of the legend *Item Properties* panel provide the following functionalities (see 图 18.31):

▼ WMS LegendGraphic

Legend width

Legend height

▼ Spacing

Legend Title

Space below

Groups

Above group

Below group heading

Subgroups

Above subgroup

Below subgroup heading

Legend Items

Space between symbols

Symbol label space

General

Box space

Column space

Line space

图 18.31: WMS LegendGraphic and Spacing groups


When you have added a WMS layer and you insert a legend item, a request will be sent to the WMS server to provide a WMS legend. This Legend will only be shown if the WMS server provides the GetLegendGraphic capability. The WMS legend content will be provided as a raster image.


WMS LegendGraphic is used to be able to adjust the *Legend width* and the *Legend height* of the WMS legend raster

image.

Spacing around title, groups, subgroups, symbols, labels, boxes, columns and lines can be customized through this dialog.

18.2.6 The Scale Bar Item

Scale bars provide a visual indication of the size of features, and distance between features, on the map item. A scale bar item requires a map item. Use the  *Add Scale Bar* tool following *items creation instructions* to add a new scale bar item that you can later manipulate the same way as exposed in *Interacting with layout items*.

By default, a new scale bar item shows the scale of the map item over which it is drawn. If there is no map item below, the *reference map* is used. You can customize it in the *Item Properties* panel. Other than the *items common properties*, this feature has the following functionalities (see  18.32):

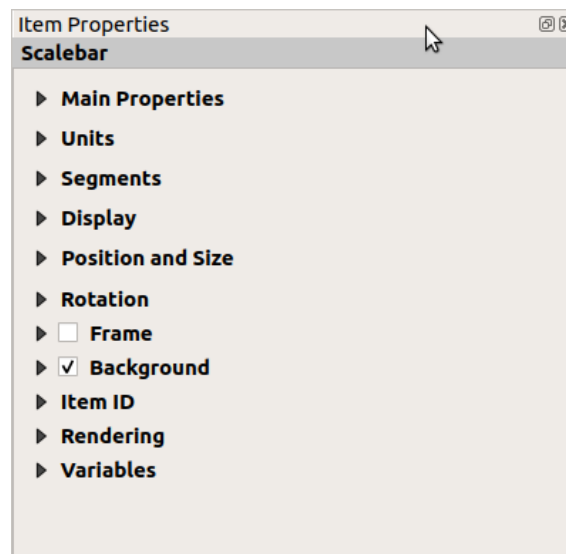



图 18.32: Scale Bar Item Properties Panel

Main properties

The *Main properties* group of the scale bar *Item Properties* panel provides the following functionalities (see  18.33):

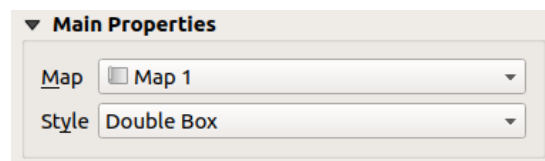


图 18.33: Scale Bar Main properties group

1. First, choose the map the scale bar will be attached to
2. Then, choose the style of the scale bar. Available styles are:
 - **Single box** and **Double box** styles, which contain one or two lines of boxes alternating colors;
 - **Middle**, **Up** or **Down** line ticks;
 - **Stepped line** style that draws a stepped line representation of a scalebar
 - **Hollow** style that draws a single box with alternating color for the segments, with horizontal lines through alternating segments

- **Numeric**, where the scale ratio is printed (e.g., 1 : 50000).
3. Set properties as appropriate

Units

The *Units* group of the scale bar *Item Properties* panel provides the functionalities to set the units of display and some text formatting (see 图 18.34):

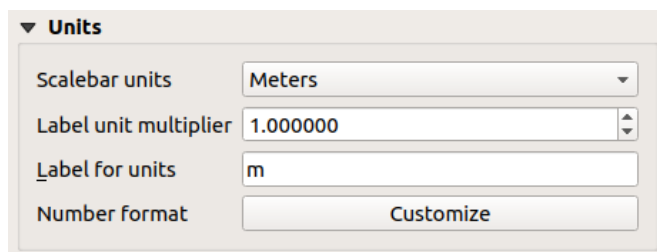


图 18.34: Scale Bar Units group

- Select the units you want to use with *Scalebar units*. There are many possible choices: **Map Units** (the default one), **Meters**, **Feet**, **Miles** or **Nautical Miles...** and some derivatives. Units conversion is handled automatically.
- The *Label unit multiplier* specifies how many scale bar units per labeled unit. Eg, if your scale bar units are set to "meters", a multiplier of 1000 will result in the scale bar labels in "kilometers".
- The *Label for units* field defines the text used to describe the units of the scale bar, eg m or km. This should be matched to reflect the multiplier above.
- Press *Customize* next to *Number format* to have control over all the formatting properties for the numbers in the scale bar, including thousand separators, decimal places, scientific notation, etc. (see [Number Formatting](#) for more details). Very useful in the case of making maps for audiences outside of the current QGIS locale, or when you would like to vary the style from the locale defaults (e.g. adding thousands separators when the locale default is to hide them).

Segments

The *Segments* group of the scale bar *Item Properties* panel provides the functionalities to configure the number and size of segments and subdivisions (see 图 18.35):

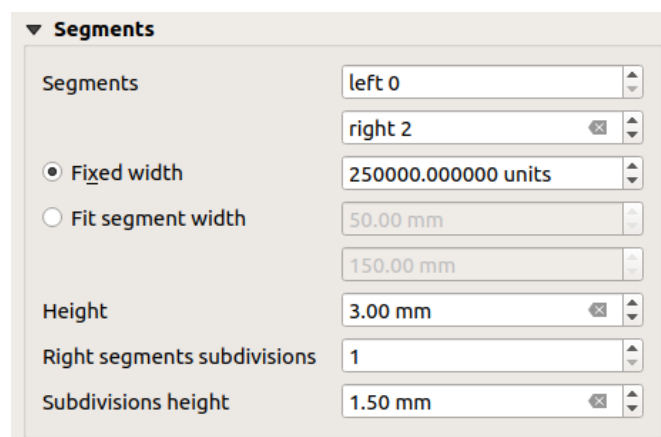


图 18.35: Scale Bar Segments group

- You can define the number of *Segments* that will be drawn at the left and right sides of the 0 of the scale bar:

- number of subdivisions of a unique segment on the *Left* side
- number of segments on the *Right* side
- You can set how long a segment will be (*Fixed width*), or limit the scale bar size in mm with *Fit segment width* option. In the latter case, each time the map scale changes, the scale bar is resized (and its label updated) to fit the range set.
- *Height* is used to define the height of the bar.
- *Right segment subdivisions* is used to define the number of sections the right-side segments of the scale bar can have (for *Line Ticks Down*, *Line Ticks Middle* and *Line Ticks Up* scale bar styles) .
- *Subdivision height* is used to define the height of the subdivision segment.

Display

The *Display* group of the scale bar *Item Properties* panel provides the following functionalities:

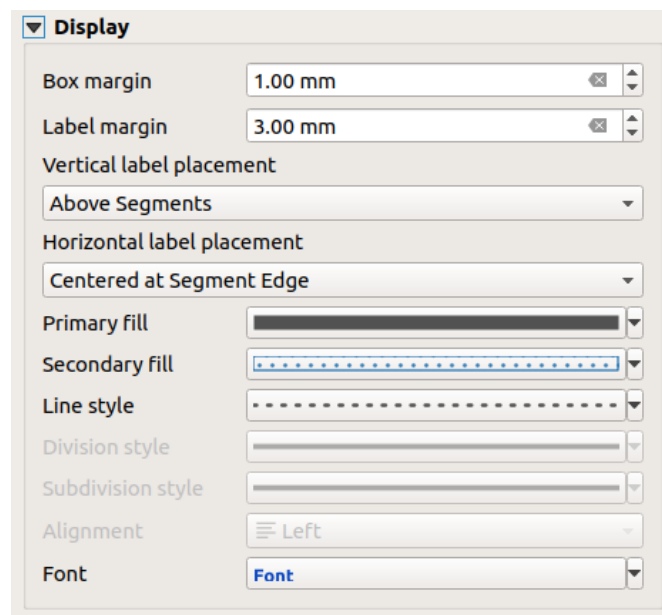


图 18.36: Scale Bar Display group

You can define how the scale bar will be displayed in its frame.

- *Box margin* : space between text and frame borders
- *Label margin* : space between text and scale bar drawing
- *Vertical label placement*: it can be above or below the scale bar segment
- *Horizontal label placement*: which would be centered at the scale bar segment's edge or center
- *Primary fill* and *Secondary fill* of the scale bar drawing using *fill symbols properties* (color, opacity, patterns, effects...) --- for *Single Box*, *Double Box* and *Hollow* styles
- *Line style* of the scale bar drawing using *line symbols properties* (color, stroke, join, cap style, patterns, effects...) --- for all but *Numeric* style
- *Division style* and *Subdivision style* respectively for division and subdivision segments in *Line Ticks Up*, *Line Ticks Middle* and *Line Ticks Down* scale bar styles using *line symbols properties* (color, stroke, join, cap style, patterns, effects...)
- *Alignment* puts text on the left, center or right side of the frame (only for *Numeric* scale bar style)
- *Font* to set the *properties* (size, font, color, letter spacing, shadow, background...) of the scale bar label.

Since most of the display properties of the scale bar rely on symbols whose properties can be data-defined, it's possible to render data-defined scale bars.

Example: The following code applied to the bold property of the scale labels will display numbers in bold when they are a multiple of 500:


```
-- returns True (or 1) if the value displayed on the bar  
-- is a multiple of 500  
  
@scale_value % 500 = 0
```


18.2.7 The Table Items

You can use table items to decorate and explain your map:

- *Attribute table*: shows a subset of the attributes of a layer, based on predefined rules
- *Fixed table*: inserts a manual text table where information can be independent from the layers.

The attribute table item

Any layer in the project can have its attributes shown in the print layout. Use the  *Add Attribute Table* tool following *items creation instructions* to add a new table item that you can later manipulate the same way as exposed in *Interacting with layout items*.

By default, a new attribute table item loads first rows of the first (alphabetically sorted) layer, with all the fields. You can however customize the table thanks to its *Item Properties* panel. Other than the *items common properties*, this feature has the following functionalities (see  18.37):

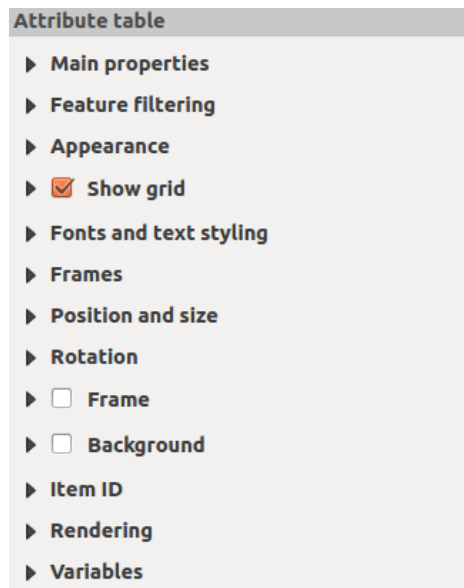


图 18.37: Attribute table Item Properties Panel

Main properties

The *Main properties* group of the attribute table provides the following functionalities (see 图 18.38):

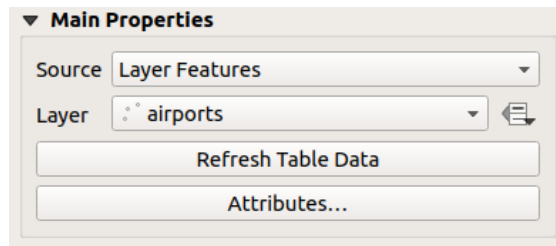



图 18.38: Attribute table Main properties Group

- For *Source* you can by default only select **Layer features** allowing you to select a *Layer* from the vector layers loaded in the project.

The  *data-defined override* button near the layer list allows you to dynamically change the layer which is used to populate the table, e.g. you could fill the attribute table with different layer attributes per atlas page. Note that the table structure used (图 18.41) is the one of the layer shown in the *Layer* drop-down list and it is left intact, meaning that setting a data defined table to a layer with different field(s) will result in empty column(s) in the table.

In case you activate the *Generate an atlas* option in the *Atlas* panel (see *Generate an Atlas*), there are two additional *Source* possible:

- **Current atlas feature** (see 图 18.39): you won't see any option to choose the layer, and the table item will only show a row with the attributes from the current feature of the atlas coverage layer.
 - and **Relation children** (see 图 18.40): an option with the relation names will show up. This feature can only be used if you have defined a *relation* using your atlas coverage layer as parent, and the table will show the children rows of the atlas coverage layer's current feature.
- The button *Refresh Table Data* can be used to refresh the table when the actual contents of the table has changed.

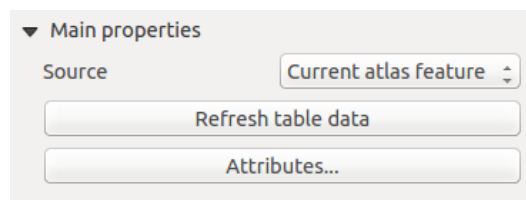


图 18.39: Attribute table Main properties for 'Current atlas feature'

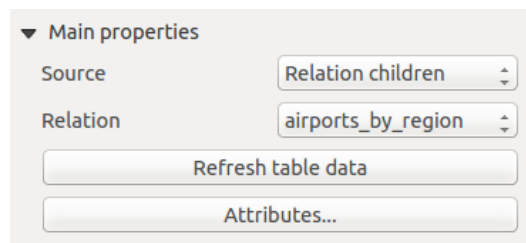


图 18.40: Attribute table Main properties for 'Relation children'

- The button *Attributes...* starts the *Select Attributes* dialog, (see 图 18.41) that can be used to change the visible contents of the table. The upper part of the window shows the list of the attributes to display and the lower part helps you sort the data.

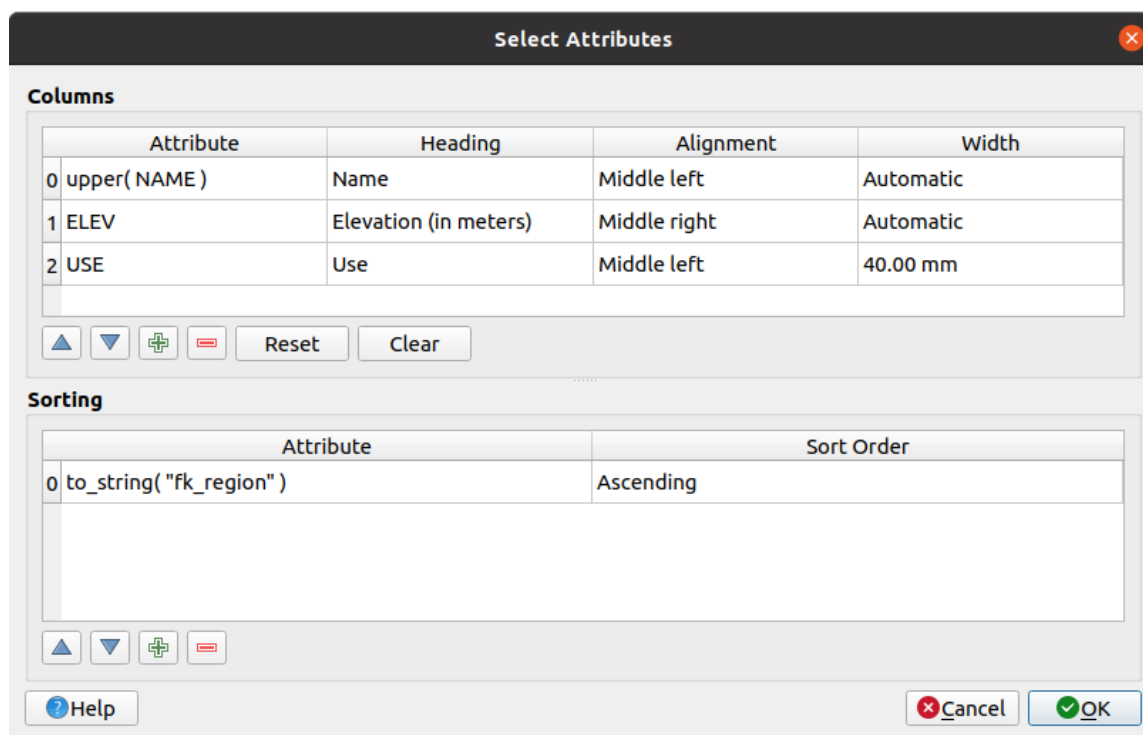










图 18.41: Attribute table Select attributes Dialog

In the *Columns* section you can:

- Move attributes up or down the list by selecting the rows and then using the  and  buttons to shift the rows. Multiple rows can be selected and moved at any one time.
- Add an attribute with the  button. This will add an empty row at the bottom of the table where you can select a field to be the attribute value or create an attribute via a regular expression.
- Remove an attribute with the  button. Multiple rows can be selected and removed at any one time.
- Reset the attribute table back to its default state with the *Reset* button.
- Clear the table using the *Clear* button. This is useful when you have a large table but only want to show a small number of attributes. Instead of manually removing each row, it may be quicker to clear the table and add the rows needed.
- Cell headings can be altered by adding the custom text in the *Heading* column.
- Cell alignment can be managed with the *Alignment* column which will dictate the texts position within the table cell.
- Cell width can be manually managed by adding custom values to the *width* column.

In the *Sorting* section you can:

- Add an attribute to sort the table with: press the  button and a new empty row is added. Insert a field or an expression in the *Attribute* column and set the *Sort order* to **Ascending** or **Descending**.
- Select a row in the list and use the  and  buttons to change the sort priority on attribute level. Selecting a cell in the *Sort Order* column helps you change the sorting order of the attribute field.
- Use the  button to remove an attribute from the sorting list.

Feature filtering

The *Feature filtering* group of the attribute table provides the following functionalities (see 图 18.42):

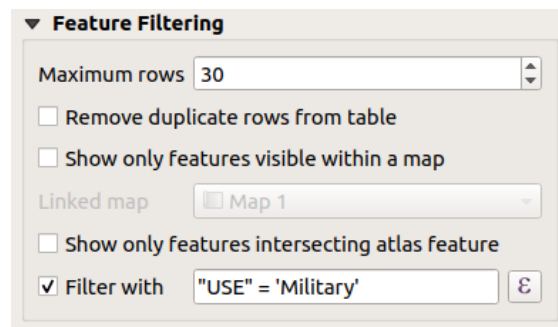



图 18.42: Attribute table Feature filtering Group

You can:

- Define the *Maximum rows* to be displayed.
- Activate *Remove duplicate rows from table* to show unique records only.
- Activate *Show only visible features within a map* and select the corresponding *Linked map* whose visible features attributes will be displayed.
- Activate *Show only features intersecting Atlas feature* is only available when *Generate an atlas* is activated. When activated it will show a table with only the features which intersect the current atlas feature.
- Activate *Filter with* and provide a filter by typing in the input line or insert a regular expression using the given  expression button. A few examples of filtering statements you can use when you have loaded the airports layer from the Sample dataset:

- `ELEV > 500`
- `NAME = 'ANIAC'`
- `NAME NOT LIKE 'AN%'`
- `regexp_match(attribute($currentfeature, 'USE') , '[i]')`

The last regular expression will include only the airports that have a letter 'i' in the attribute field 'USE'.

Appearance

The *Appearance* group of the attribute table provides the following functionalities (see 图 18.43):

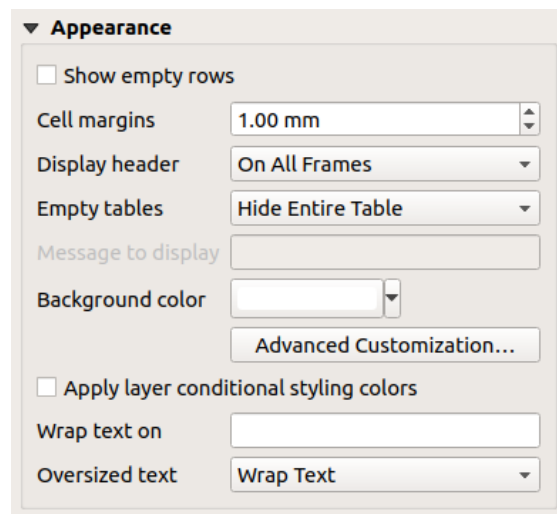


图 18.43: Attribute table appearance Group

- Click *Show empty rows* to fill the attribute table with empty cells. This option can also be used to provide additional empty cells when you have a result to show!
- With *Cell margins* you can define the margin around text in each cell of the table.
- With *Display header* you can select from a list one of 'On first frame', 'On all frames' default option, or 'No header'.
- The option *Empty table* controls what will be displayed when the result selection is empty.
 - **Draw headers only**, will only draw the header except if you have chosen 'No header' for *Display header*.
 - **Hide entire table**, will only draw the background of the table. You can activate *Don't draw background if frame is empty* in *Frames* to completely hide the table.
 - **Show set message**, will draw the header and adds a cell spanning all columns and display a message like 'No result' that can be provided in the option *Message to display*
- The option *Message to display* is only activated when you have selected **Show set message** for *Empty table*. The message provided will be shown in the table in the first row, when the result is an empty table.
- With *Background color* you can set the background color of the table using the *color selector* widget. The *Advanced customization* option helps you define different background colors for each cell (see 图 18.44)

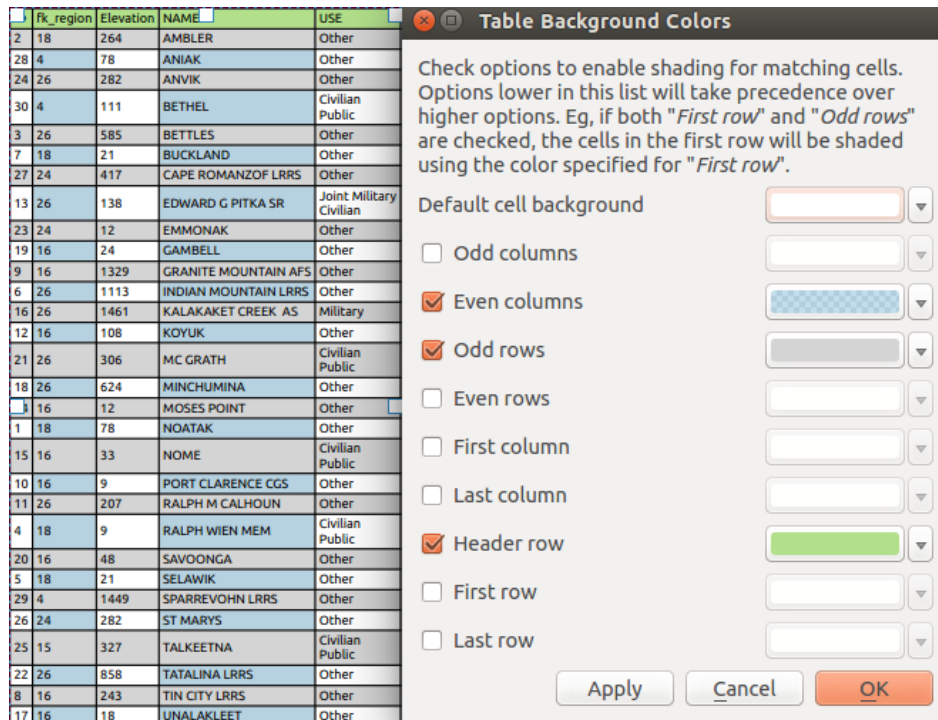



图 18.44: Attribute table Advanced Background Dialog

- *Apply layer conditional styling colors*: the *conditional table formatting* present in the layer is applied inside the layout attribute table (*only background and foreground colors are currently supported*). Conditional formatting rules take precedence over other layout table formatting settings, e.g. they will override other cell background color settings such as alternating row colors.
- With the *Wrap text on* option, you can define a character on which the cell content will be wrapped each time it is met
- With *Oversized text* you define the behavior when the width set for a column is smaller than its content's length. It can be **Wrap text** or **Truncate text**.

注解: More properties of the attribute table item are described in the *Tables common functionalities* section.

The fixed table item

Additional information about the map can be inserted manually into a table by choosing  *Add Fixed Table* and by following *items creation instructions* to add a new table item that you can later manipulate the same way as exposed in *Interacting with layout items*.

By default, an empty table with two minimized columns and rows appears in the map layout. You have to customize the table in the *Item Properties* panel. Other than the *items common properties*, this feature has the following functionalities:

Main properties

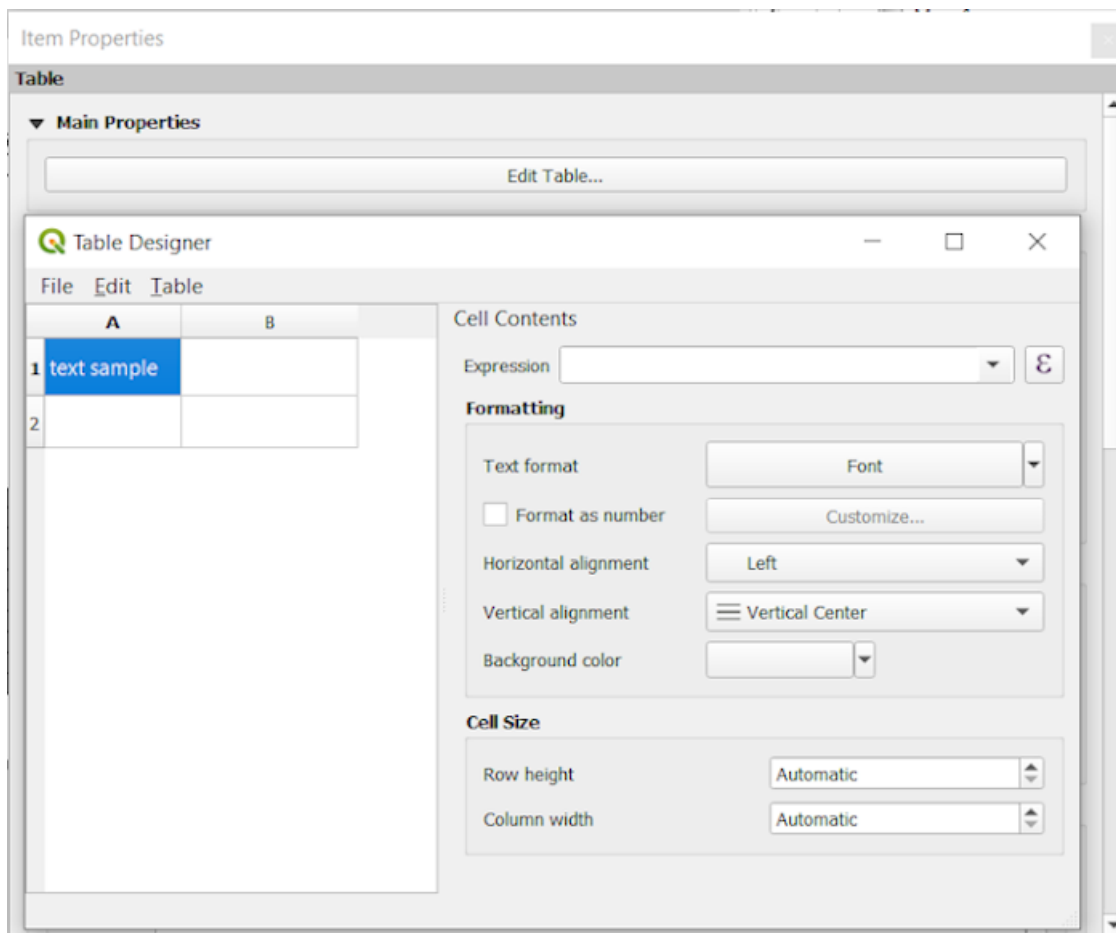


图 18.45: Fixed table Item Properties Panel with Table designer

In *Main properties* you can work with the *Table designer* when clicking the *Edit table ...*:

- You can click into the table and insert texts manually.
- Through the menus on top it is possible to:
 - *Import Content From Clipboard* by going to *File* (it overrides given inputs).
 - work with selection functionalities for rows and columns by going to *Edit*.
 - *Insert rows, Insert columns, Delete Rows, Delete Columns* as well as using the option to *Include Header Row*.
- You can work with the *Cell Contents* section on the right and:
 - Define the text format of selected cells in *Formatting*
 - * by clicking on the given \mathcal{E} expression button and using a regular expression for the input of the cell
 - * by choosing the *Text format*
 - * by *Format as number* (several formats are available)
 - * by defining the *Horizontal alignment* and the *Vertical alignment*
 - * by choosing a *Background color*

- Define the *Cell Size* with *Row height* and *Column width*.

Appearance

The *Appearance* group of the fixed table provides the following functionalities:

- Click *Show empty rows* to fill the attribute table with empty cells.
- With *Cell margins* you can define the margin around text in each cell of the table.
- With *Display header* you can select from a list one of 'On first frame', 'On all frames' default option, or 'No header'.
- With *Background color* you can set the background color of the table using the *color selector* widget. The *Advanced customization* option helps you define different background colors for each cell.
- With *Oversized text* you define the behavior when the width set for a column is smaller than its content's length. It can be **Wrap text** or **Truncate text**.

注解: More properties of the fixed table item are described in the *Tables common functionalities* section.

Tables common functionalities

Show grid

The *Show grid* group of the table items provides the following functionalities (see 图 18.46):

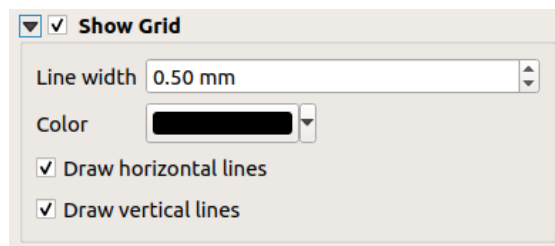


图 18.46: Attribute table Show grid Group

- Activate *Show grid* when you want to display the grid, the outlines of the table cells. You can also select to either *Draw horizontal lines* or *Draw vertical lines* or both.
- With *Line width* you can set the thickness of the lines used in the grid.
- The *Color* of the grid can be set using the color selection widget.

Fonts and text styling

The *Fonts and text styling* group of the table items provides the following functionalities (see 图 18.47):



图 18.47: Attribute table Fonts and text styling Group

- You can define *Font* properties for *Table heading* and *Table contents*, using the advanced *text settings* widget (with buffer, shadow, paint effects, transparency, background, coloring, ...). Note that these changes do not affect the cells that have custom font assigned, either from the *Appearance* section or the *Table Designer* dialog. Only cells with the default rendering are overwritten.
- For *Table heading* you can additionally set the *Alignment* to *Follow column alignment* or override this setting by choosing *Left*, *Center* or *Right*. The column alignment is set using the *Select Attributes* dialog (see 图 18.41).

Frames

The *Frames* group of the table item properties provides the following functionalities (see 图 18.48):

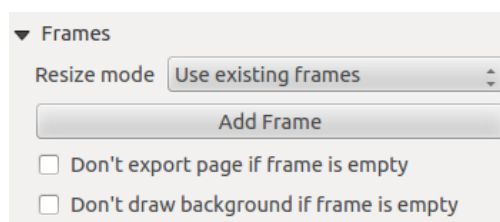



图 18.48: Attribute table Frames Group


- With *Resize mode* you can select how to render the attribute table contents:
 - *Use existing frames* displays the result in the first frame and added frames only.
 - *Extend to next page* will create as many frames (and corresponding pages) as necessary to display the full selection of attribute table. Each frame can be moved around on the layout. If you resize a frame, the resulting table will be divided up between the other frames. The last frame will be trimmed to fit the table.
 - *Repeat until finished* will also create as many frames as the *Extend to next page* option, except all frames will have the same size.
- Use the *Add Frame* button to add another frame with the same size as selected frame. The result of the table that will not fit in the first frame will continue in the next frame when you use the *Resize mode Use existing frames*.
- Activate *Don't export page if frame is empty* prevents the page to be exported when the table frame has no contents. This means all other layout items, maps, scalebars, legends etc. will not be visible in the result.
- Activate *Don't draw background if frame is empty* prevents the background to be drawn when the table frame has no contents.

18.2.8 The Picture and the North Arrow Items

The *Picture* item is a tool that helps decorate your map with pictures, logos... It can also be used to add north arrows, despite the dedicated *North arrow* tool.

The Picture Item

You can add a picture by dragging it from your file manager onto the canvas, or by using the  **Add Picture**, following *items creation instructions*. Then you can manipulate it, as explained in *Interacting with layout items*.

When using  **Add Picture**, the picture item will be a blank frame that you can customize using its *Item Properties* panel. Other than the *items common properties*, this feature has the following functionalities (see [图 18.49](#)):

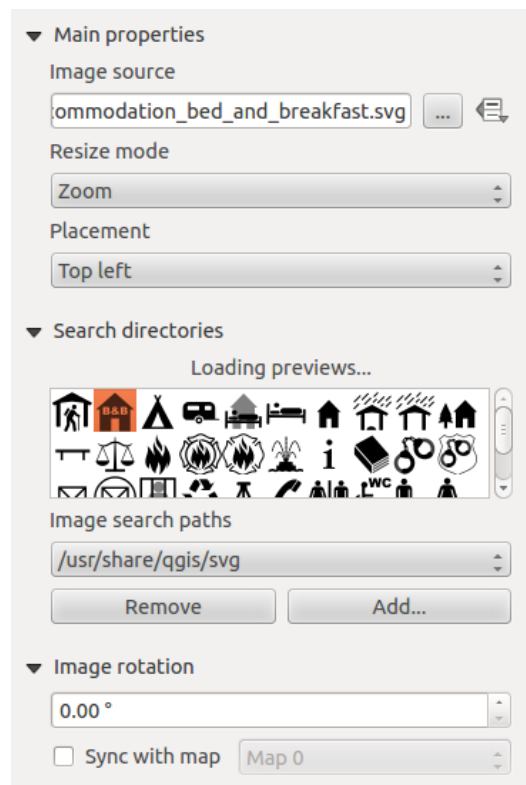



图 18.49: Picture Item Properties panel

There are several ways to set the *Image source* (to select the image you want to display):

1. In the *Main properties* group, use the ... **Browse** button of *image source* to select a file on your computer. The browser will start in the SVG-libraries provided with QGIS. You can also select other image formats (like .png or .jpg).
2. You can enter the source directly in the *Image source* text field. You can even provide a remote URL that points to a picture.
3. From the *Search directories* area you can select an image from the loaded previews to set the image source. These images are by default provided by folders set in *Settings ► Options ► System ► SVG Paths*.
4. Use the  **data defined override** button to set the image source from a feature attribute or using a regular expression.

注解: In the *Search directories* group, you can use the *Add* and *Remove* buttons to customize the list of folders to fetch and preview images from.

With the *Resize mode* option, you can set how the image is displayed when the frame is resized:

- **Zoom:** enlarges/reduces the image to the frame while maintaining the aspect ratio of picture
- **Stretch:** stretches the image to fit inside the frame
- **Clip:** use this mode for raster images only, it sets the size of the image to the original image size without scaling, and the frame is used to clip the image. So only the part of the image that is inside the frame will be visible.
- **Zoom and resize frame:** enlarges the image to fit the frame, and then resizes frame to fit the resulting image dimensions
- **Resize frame to image size:** sets the size of the frame to match the original size of the image (no scaling)


Depending on the selected *Resize mode*, the *Placement* and *Image rotation* options may be disabled. *Placement* lets you select the position of the image inside its frame.

The QGIS provided (default) *.SVG* files are customizable, meaning that you can easily apply other *Fill color*, *Stroke color* (including opacity) and *Stroke width* than the original, using their corresponding feature in the *SVG Parameters* group. These properties can also be *data-defined*.

If you add an *.SVG* file that does not enable these properties, you may need to add the following tags to the file in order to add support e.g. for transparency:

- *fill-opacity="param(fill-opacity)"*
- *stroke-opacity="param(outline-opacity)"*


You can read this [blog post](#) to see an example.

Images can be rotated with the *Image rotation* field. Activating the  *Sync with map* checkbox synchronizes the rotation of the image with the rotation applied to a selected map item. This is a convenient feature for north arrows that you can align with either:

- **Grid north:** the direction of a grid line which is parallel to the central meridian of the national/local grid
- **True north:** direction of a meridian of longitude.

You can also apply a declination *Offset* to the picture rotation.

The North Arrow Item

You can add a north arrow with the  **Add North Arrow** button, following *items creation instructions* and manipulate it the same way as exposed in *Interacting with layout items*.

Since north arrows are images, the *North Arrow* item has the same properties as the *picture item*. The main differences are:

- A default north arrow is used when adding the item, instead of a blank frame
- The north arrow item is synced with a map item by default: the *Sync with map* property is the map over which the north arrow item is drawn. If none, it falls back to the *reference map*.

注解: Many of the north arrows do not have an 'N' added in the north arrow. This is done on purpose, since there are languages that do not use an 'N' for North.

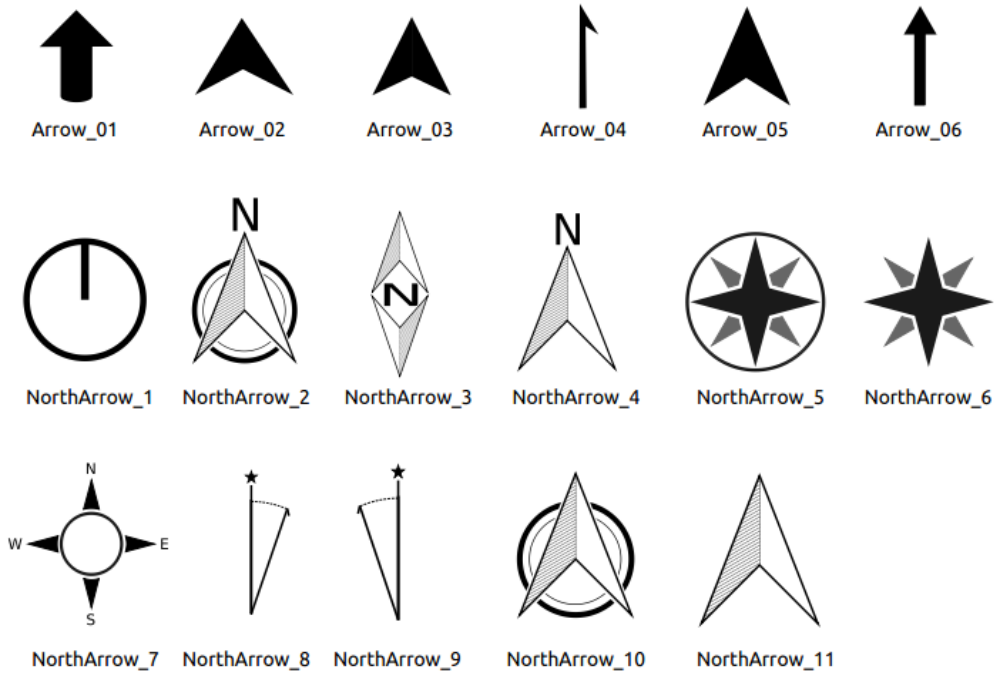



图 18.50: North arrows available for selection in provided SVG library

18.2.9 The HTML Frame Item

It is possible to add a frame that displays the contents of a website or even create and style your own HTML page and display it! You can add a picture with the  *Add HTML* following *items creation instructions* and manipulate it the same way as exposed in *Interacting with layout items*. Note that the HTML scale is controlled by the layout export resolution at the time the HTML frame is created.

The HTML item can be customized using its *Item Properties* panel. Other than the *items common properties*, this feature has the following functionalities (see 图 18.51):

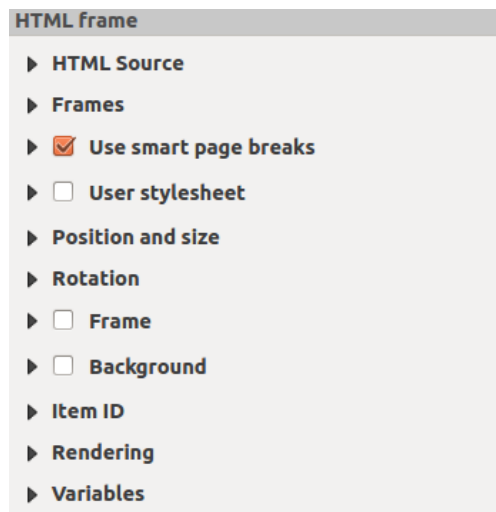


图 18.51: HTML Frame, the Item Properties Panel

HTML Source

The *HTML Source* group of the HTML frame *Item Properties* panel provides the following functionalities (see [图 18.52](#)):

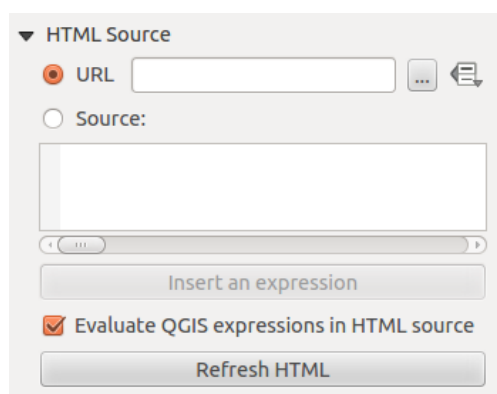


图 18.52: HTML frame, the HTML Source properties

- In *URL* you can enter the URL of a webpage you copied from your Internet browser or select an HTML file using the ... ^{Browse} button. There is also the option to use the ^{Data-defined override} button, to provide a URL from the contents of an attribute field of a table or using a regular expression.
- In *Source* you can enter text in the textbox with some HTML tags or provide a full HTML page.
- The *Insert or Edit an Expression...* button can be used to add an expression like [%Year(\$now)%] in the Source textbox to display the current year. This button is only activated when radiobutton *Source* is selected. After inserting the expression click somewhere in the textbox before refreshing the HTML frame, otherwise you will lose the expression.
- Activate *Evaluate QGIS expressions in HTML code* to see the result of the expression you have included, otherwise you will see the expression instead.
- Use the *Refresh HTML* button to refresh the HTML frame(s) and see the result of changes.

Frames

The *Frames* group of the HTML frame *Item Properties* panel provides the following functionalities (see [图 18.53](#)):

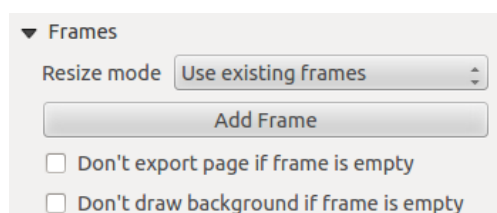


图 18.53: HTML frame, the Frames properties

- With *Resize mode* you can select how to render the HTML contents:
 - Use existing frames displays the result in the first frame and added frames only.
 - Extend to next page will create as many frames (and corresponding pages) as necessary to render the height of the web page. Each frame can be moved around on the layout. If you resize a frame, the webpage will be divided up between the other frames. The last frame will be trimmed to fit the web page.
 - Repeat on every page will repeat the upper left of the web page on every page in frames of the same size.

- Repeat until finished will also create as many frames as the Extend to next page option, except all frames will have the same size.
- Use the *Add Frame* button to add another frame with the same size as selected frame. If the HTML page does not fit in the first frame it will continue in the next frame when you use *Resize mode* or *Use existing frames*.
- Activate *Don't export page if frame is empty* prevents the page from being exported when the frame has no HTML contents. This means all other layout items, maps, scale bars, legends etc. will not be visible in the result.
- Activate *Don't draw background if frame is empty* prevents the HTML frame being drawn if the frame is empty.

Use smart page breaks and User style sheet

The *Use smart page breaks* dialog and *User style sheet* dialog of the HTML frame *Item Properties* panel provides the following functionalities (see 图 18.54):

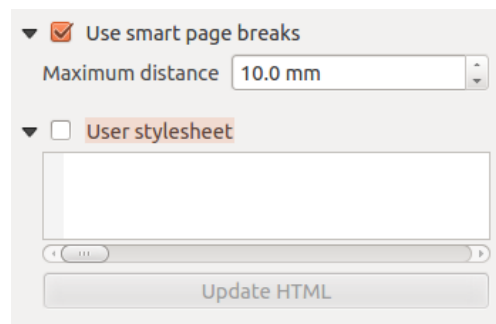


图 18.54: HTML frame, Use smart page breaks and User style sheet properties

- Activate *Use smart page breaks* to prevent the html frame contents from breaking mid-way a line of text so it continues nice and smooth in the next frame.
- Set the *Maximum distance* allowed when calculating where to place page breaks in the html. This distance is the maximum amount of empty space allowed at the bottom of a frame after calculating the optimum break location. Setting a larger value will result in better choice of page break location, but more wasted space at the bottom of frames. This is only used when *Use smart page breaks* is activated.
- Activate *User style sheet* to apply HTML styles that often is provided in cascading style sheets. An example of style code is provided below to set the color of `<h1>` header tag to green and set the font and font size of text included in paragraph tags `<p>`.

```
h1 {color: #00ff00;
}
p {font-family: "Times New Roman", Times, serif;
font-size: 20px;
}
```



- Use the *Update HTML* button to see the result of the style sheet settings.

18.2.10 图形项

QGIS 在打印布局中提供了一系列工具绘制常规或复杂图形

注解： 与其他打印布局项不同，你不能自定义框架风格，也不能自定义形状边界的背景颜色（默认设定为透明色）

常规图形项

布局:guiabel: 图形项是一个有助于布置你地图的工具, 包含三角形、矩形和椭圆形等常规图形... 你可以使用 `laddBasicShapel:sup:` 添加图形工具来添加一个常规图形, 点击该工具集可以访问特定的工具, 例如  添加矩形, `laddBasicCirclel` 添加椭圆 和  添加三角形. 选择了合适的工具之后, 你可以根据接下来的:ref:‘项目创建指令 `<create_layout_item>`’绘制图形. 和其他布局项一样, 常规形状的操作方法和:ref:‘interact_layout_item’中提示的操作一样.

注解： 在绘制基本图形时要按住:kbd:‘Shift’键, 使用单击 + 拖动的方式, 可以帮助你创建一个完美的正方形, 圆或者三角形.

默认的图形项可以使用它的:guiabel: ‘项目属性’ 面板来定制. 除了:ref: ‘项目基本属性 `<item_common_properties>`’, 该特性还有以下功能 (参见:numref: ‘figure_layout_basic_shape’):

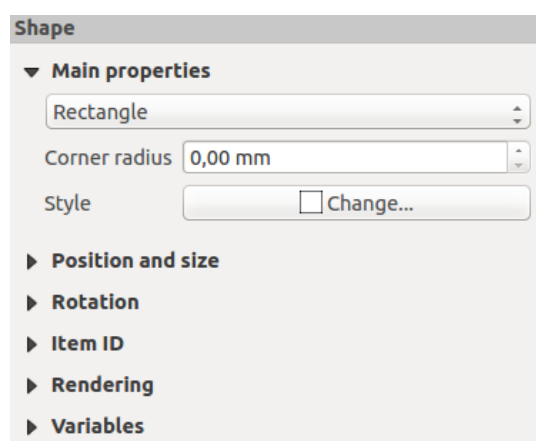


图 18.55: 图形项属性面板

主要特性组中, 并在图框中允许你转换形状项的类型 (椭圆, 矩形 or 三角形).

你可以使用高级选择器控件:ref: 符号 `<symbol-selector>` and 颜色 设置形状的风格

对于矩形图形, 你可以使用不同的单位来更改:guiabel: 圆角半径设置圆角.

基于结点的形状项

尽管 `laddBasicShapel` 添加形状工具提供了创建简单和预定义几何项的方式, [:gui-label:添加结点项](#) 工具帮助你创建一个自定义或更高级的几何项。对于折线或多边形, 你可以画任意数量的线或者边, 并且项目的顶点可以使用 [:gui-label:编辑结点项](#) 独立且直接地操作。项目本身也可以在 `:ref:interact_layout_item` 中操作。

添加一个基于结点的形状

1. 单击 [:gui-label:添加结点项](#) 图标
2. 选择 [:gui-label:添加多边形](#) 或 [:gui-label:添加折线](#) 工具
3. Perform consecutive left clicks to add nodes of your item. If you hold down the `Shift` key while drawing a segment, it is constrained to follow an orientation multiple of 45° .
4. 完成绘制后, 右键单击, 结束图形绘制。

你可以在 `:gui-label:` 项目特性面板自定义形状的外观。

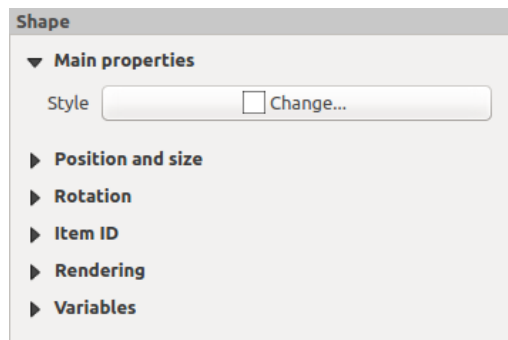


图 18.56: 多边形结点形状项目特性面板

在主要特性‘中, 你可以使用高级选择器控件 `:ref:符号 <symbol-selector>` 和 `:ref:颜色 <color-selector>` 设置样式...

对于折线结点项, 你同样可以使用 `:gui-label:线路标记` 参数化后添加

- 使用选项开始/结束标记:
 - 无: 绘制一个简单折线
 - 箭头: 添加一个自定义箭头指向的常规三角形.
 - SVG 标记: 使用文件 `file:SVG` 文件作为项目目标.
- 自定义箭头指向
 - *Arrow stroke color*: sets the stroke color of the arrow head.
 - *Arrow fill color*: sets the fill color of the arrow head.
 - *Arrow stroke width*: sets the stroke width of the arrow head.
 - *Arrow head width*: sets the size of the arrow head.

SVG images are automatically rotated with the line. Stroke and fill colors of QGIS predefined SVG images can be changed using the corresponding options. Custom SVG may require some tags following this [instruction](#).

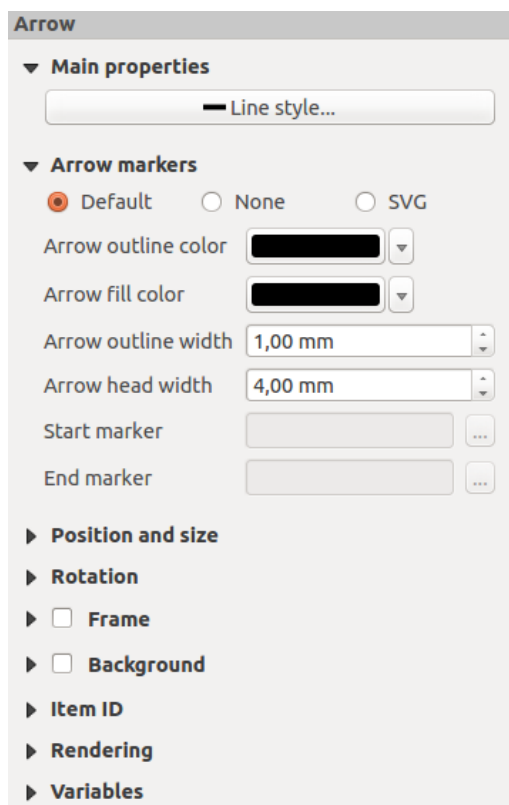




图 18.57: Polyline Node Shape Item Properties Panel

The Arrow Item

The  **Add Arrow** tool is a shortcut to create an arrow-enabled polyline by default and thus has the same properties and behavior as a *polyline node item*.

Actually, the arrow item can be used to add a simple arrow, for example, to show the relation between two different print layout items. However, to create a north arrow, the *image item* should be considered first as it gives access to a set of north arrows in *.SVG* format that you can sync with a map item so that it rotates automatically with it.

Editing a node item geometry

A specific tool is provided to edit node-based shapes through  **Edit Nodes Item**. Within this mode, you can select a node by clicking on it (a marker is displayed on the selected node). A selected node can be moved either by dragging it or by using the arrow keys. Moreover, in this mode, you are able to add nodes to an existing shape: double-click on a segment and a node is added at the place you click. Finally, you can remove the currently selected node by hitting the `Del` key.

18.3 Creating an Output

图 18.58 shows an example print layout including all the types of layout items described in the previous section.

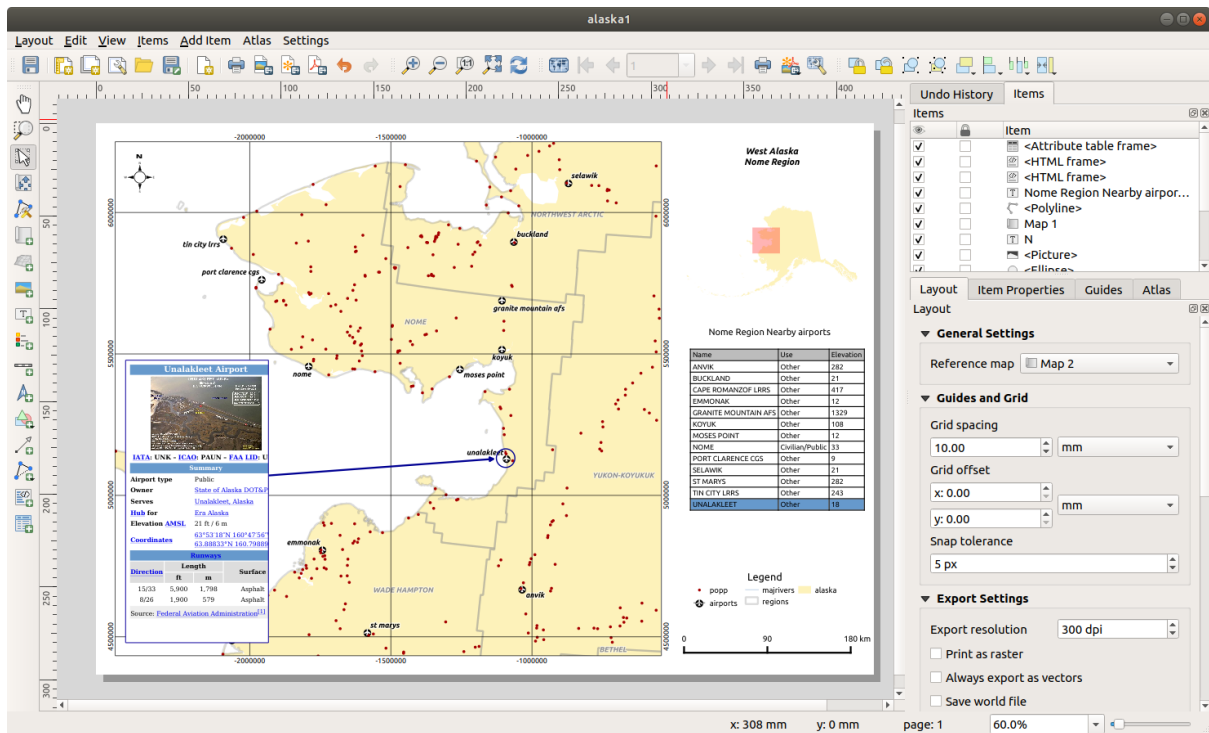


图 18.58: Print Layout with map view, legend, image, scale bar, coordinates, text and HTML frame added

From the *Layout* menu or toolbar, you can output the print layout to different file formats, and it is possible to modify the resolution (print quality) and paper size:

- The Print icon allows you to print the layout to a connected printer or a PostScript file, depending on the installed printer drivers.
- The Export as image icon exports the print layout image formats such as PNG, BMP, TIF, JPG, and many others...
- The Export as SVG icon saves the print layout as an SVG (Scalable Vector Graphic).
- The Export as PDF icon saves the defined print layout directly as a PDF (Portable Document Format) file.


18.3.1 Export settings

Whenever you export a print layout, there are a selection of export settings QGIS needs to check in order to produce the most appropriate output. These configurations are:

- The *Export settings* of the *Layout* panel, such as *Export resolution*, *Print as raster* *Always export as vectors* or *Save world file*
- *Exclude page from exports* in the *page item properties* panel
- *Exclude item from exports* in the *item properties* panel

18.3.2 Export as Image

To export a layout as an image:

1. Click the  Export as image icon
2. Select the image format, the folder and filename (e.g. `myill.png`) to use. If the layout contains more than one page, each page will be exported to a file with the given filename with the page number appended (e.g. `myill_2.png`).
3. In the next (*Image Export Options*) dialog:
 - You can override the print layout *Export resolution* and the exported page dimensions (as set in *Layout panel*).
 - Image rendering can also be improved with the *Enable antialiasing* option.
 - If you want to export your layout as a **georeferenced image** (e.g., to share with other projects), check the *Generate world file* option, and an *ESRI World File* with the same name as the exported image, but a different extension (`.tifw` for TIFF, `.pnw` for PNG, `.jgw` for JPEG, ...) will be created when exporting. This option can also be checked by default in the *layout panel*.

注解: For multi-page output, only the page that contains the *reference map* will get a world file (assuming that the *Generate world file* option is checked).

- By checking *Crop to content* option, the image output by the layout will include the minimal area enclosing all the items (map, legend, scale bar, shapes, label, image...) of each page of the composition:
 - If the composition includes a single page, then the output is resized to include EVERYTHING on the composition. The page can then be reduced or extended to all items depending on their position (on, above, below, left or right of the page).
 - In case of a multi-page layout, each page will be resized to include items in its area (left and right sides for all pages, plus top for the first page and bottom for the last page). Each resized page is exported to a separate file.

The *Crop to content* dialog also lets you add margins around the cropped bounds.

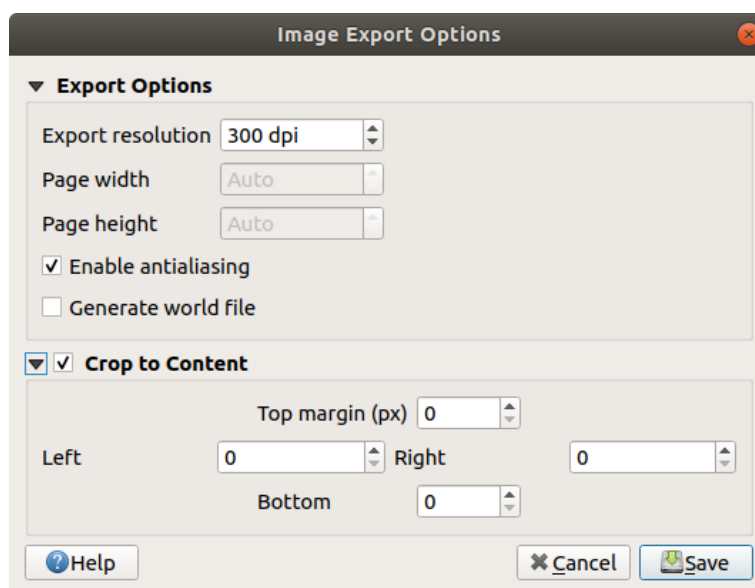


图 18.59: Image Export Options, output is resized to items extent


小技巧: Use image formats that support transparency when items extend beyond the paper extent

Layout items may be placed outside the paper extent. When exporting with the *Crop to content* option, the resulting image may therefore extend beyond the paper extent. Since the background outside of the paper extent will be transparent, for image formats that do not support transparency (e.g. BMP and JPG) the transparent background will be rendered as full black, "corrupting" the image. Use transparency-compatible formats (e.g. TIFF and PNG) in such cases.

注解: When supported by the format (e.g. PNG) and the underlying Qt library, the exported image may include *project metadata* (author, title, date, description...)

18.3.3 Export as SVG

To export a layout as SVG:

1. Click the  Export as SVG icon
2. Fill in the path and filename (used as a base name for all the files in case of multi-page composition, as for image export)
3. In the next *SVG Export Options* dialog, you can override the layout default *export settings* or configure new ones:
 - *Export map layers as SVG groups*: exported items are grouped within layers whose name matches the layer names from QGIS, making it much easier to understand the contents of the document.
 - *Always export as vectors*: some rendering options require items to be rasterized for a better rendering. Check this option to keep the objects as vectors with the risk that the appearance of the output file may not match the print layout preview (for more details, see *Export settings*).
 - *Export RDF metadata* of the document such as the title, author, date, description...
 - *Simplify geometries to reduce output file size*: this avoids exporting ALL geometry vertices, which can result in a ridiculously complex and large export file size that could fail to load in other applications. Geometries will be simplified while exporting the layout in order to remove any redundant vertices which are not discernably different at the export resolution (e.g. if the export resolution is 300 dpi, vertices that are less than 1/600 inch apart will be removed).
 - Set the *Text export*: controls whether text labels are exported as proper text objects (*Always export texts as text objects*) or as paths only (*Always export texts as paths*). If they are exported as text objects, they can be edited in external applications (e.g. Inkscape) as normal text. BUT the side effect is that the rendering quality is reduced, AND there are issues with rendering when certain text settings like buffers are in place. That's why exporting as paths is recommended.
 - Apply *Crop to content option*
 - *Disable tiled raster layer exports*: When exporting files, QGIS uses a built-in raster layer tiled rendering that saves memory. Sometimes, this can cause visible "seams" in the rasters for generated files. Checking this option would fix that, at the cost of a higher memory usage during exports.

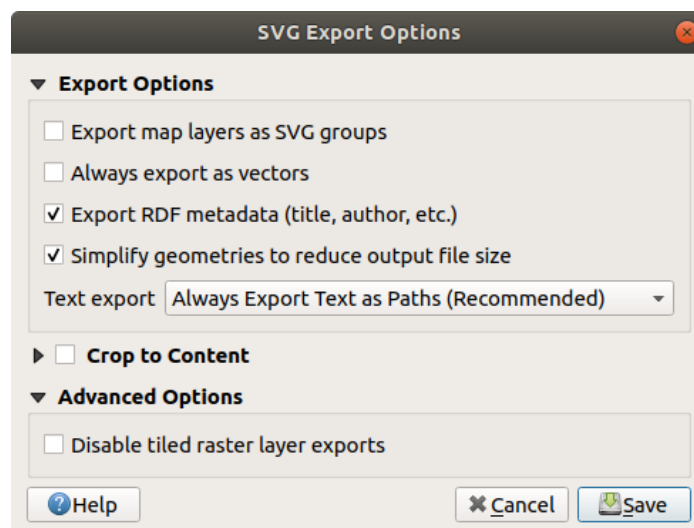



图 18.60: SVG Export Options

注解: Currently, the SVG output is very basic. This is not a QGIS problem, but a problem with the underlying Qt library. This will hopefully be sorted out in future versions.

18.3.4 Export as PDF

To export a layout as PDF:

1. Click the  Export as PDF icon
2. Fill in the path and filename: unlike for image and SVG export, all the pages in the layout are exported to a single PDF file.
3. In the next *PDF Export Options* dialog, you can override the layout default *export settings* or configure new ones:
 - *Always export as vectors*: some rendering options require items to be rasterized for a better rendering. Check this option to keep the objects as vectors with the risk that the appearance of the output file may not match the print layout preview (for more details, see *Export settings*).
 - *Append georeference information*: available only if the *reference map*, from which the information is taken, is on the first page.
 - *Export RDF metadata* of the document such as the title, author, date, description...
 - Set the *Text export*: controls whether text labels are exported as proper text objects (*Always export texts as text objects*) or as paths only (*Always export texts as paths*). If they are exported as text objects then they can be edited in external applications (e.g. Inkscape) as normal text. BUT the side effect is that the rendering quality is decreased, AND there are issues with rendering when certain text settings like buffers are in place. That's why exporting as paths is recommended.
 - Control the PDF *Image compression* using:
 - *Lossy (JPEG)*, which is the default compression mode
 - or *Lossless*, which creates bigger files in most cases, but is much more suitable for printing outputs or for post-production in external applications (requires Qt 5.13 or later).
 - *Create Geospatial PDF (GeoPDF)*: Generate a georeferenced PDF file (requires GDAL version 3 or later).

- *Disable tiled raster layer exports*: When exporting files, QGIS uses tiled based rendering that saves memory. Sometimes, this can cause visible "seams" in the rasters for generated files. Checking this option would fix that, at the cost of a higher memory usage during exports.
- *Simplify geometries to reduce output file size*: Geometries will be simplified while exporting the layout by removing vertices that are not discernably different at the export resolution (e.g. if the export resolution is 300 dpi, vertices that are less than 1/600 inch apart will be removed). This can reduce the size and complexity of the export file (very large files can fail to load in other applications).

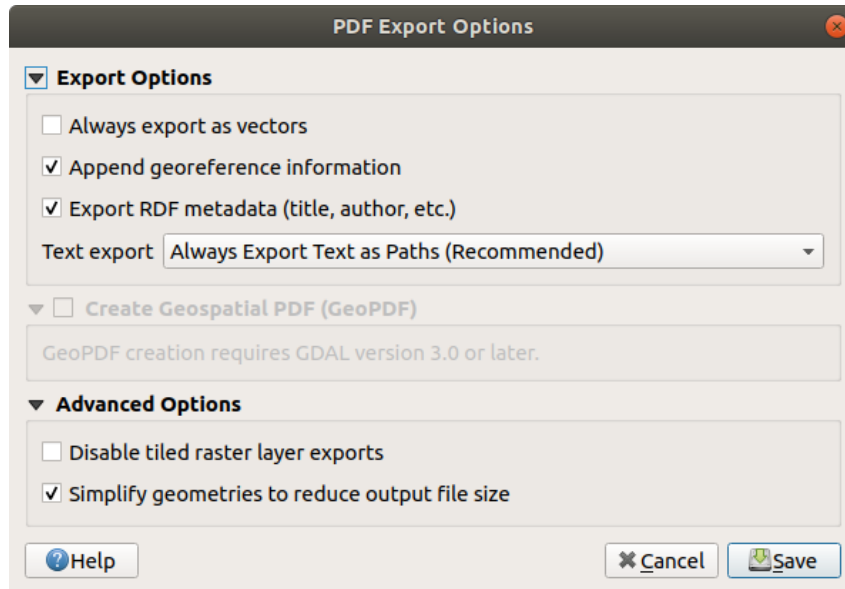


图 18.61: PDF Export Options

注解: Since QGIS 3.10, with GDAL 3, GeoPDF export is supported, and a number of GeoPDF specific options are available:

- *Format* (GeoPDF format - there are some GeoPDF variations),
- *Include multiple map themes* (specify map themes to include),
- *Include vector feature information* (choose the layers and group them into logical PDF groups).

注解: Exporting a print layout to formats that supports georeferencing (e.g. PDF and TIFF) creates a georeferenced output by default.

18.3.5 Generate an Atlas

Atlas functions allow you to create map books in an automated way. Atlas uses the features of a table or vector layer (*Coverage layer*) to create an output for each feature (**atlas feature**) in the table / layer. The most common usage is to zoom a map item to the current atlas feature. Further use cases include:

- a map item showing, for another layer, only features that share the same attribute as the atlas feature or are within its geometry.
- a label or HTML item whose text is replaced as features are iterated over
- a table item showing attributes of associated *parent or children* features of the current atlas feature...

For each feature, the output is processed for all pages and items according to their exports settings.

小技巧: Use variables for more flexibility

QGIS provides a large panel of functions and *variables*, including atlas related ones, that you can use to manipulate the layout items, but also the symbology of the layers, according to atlas status. Combining these features gives you a lot of flexibility and helps you easily produce advanced maps.

To enable the generation of an atlas and access atlas parameters, refer to the *Atlas* panel. This panel contains the following (see 图 18.62):

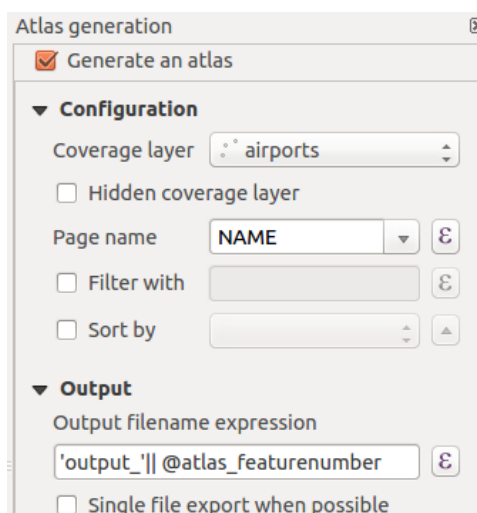



图 18.62: Atlas Panel

- *Generate an atlas* enables or disables atlas generation.
- *Configuration*
 - A *Coverage layer* combo box that allows you to choose the table or vector layer containing the features to iterate over.
 - An optional *Hidden coverage layer* that, if checked, will hide the coverage layer (but not the other layers) during the generation.
 - An optional *Page name* combo box to specify the name for the feature page(s). You can select a field of the coverage layer or set an *expression*. If this option is empty, QGIS will use an internal ID, according to the filter and/or the sort order applied to the layer.
 - An optional *Filter with* text area that allows you to specify an expression for filtering features from the coverage layer. If the expression is not empty, only features that evaluate to `True` will be processed.
 - An optional *Sort by* that allows you to sort features of the coverage layer (and the output), using a field of the coverage layer or an expression. The sort order (either ascending or descending) is set by the two-state *Sort direction* button that displays an up or a down arrow.
- *Output* - this is where the output of the atlas can be configured:
 - An *Output filename expression* textbox that is used to generate a filename for each atlas feature. It is based on expressions. is meaningful only for rendering to multiple files.
 - A *Single file export when possible* that allows you to force the generation of a single file if this is possible with the chosen output format (PDF, for instance). If this field is checked, the value of the *Output filename expression* field is meaningless.

- An *Image export format* drop-down list to select the output format when using the  Export atlas as Images... button.

Control map by atlas

The most common usage of atlas is with the map item, zooming to the current atlas feature, as iteration goes over the coverage layer. This behavior is set in the *Controlled by atlas* group properties of the map item. See *Controlled by atlas* for different settings you can apply on the map item.

Customize labels with expression

In order to adapt labels to the feature the atlas iterates over, you can include expressions. Make sure that you place the expression part (including functions, fields or variables) between [% and %] (see *The Label Item* for more details).

For example, for a city layer with fields CITY_NAME and ZIPCODE, you could insert this:

```
The area of [% concat( upper(CITY_NAME), ', ', ZIPCODE, ' is ',
format_number($area/1000000, 2) ) %] km2
```


or, another combination:

```
The area of [% upper(CITY_NAME)%], [%ZIPCODE%] is
[%format_number($area/1000000,2) %] km2
```


The information [% concat(upper(CITY_NAME), ', ', ZIPCODE, ' is ', format_number(\$area/1000000, 2)) %] is an expression used inside the label. Both expressions would result in the following type of label in the generated atlas:


```
The area of PARIS,75001 is 1.94 km2
```

Explore Data-defined override buttons with atlas


There are several places where you can use a  Data defined override button to override the selected setting. This is particularly useful with atlas generation. See *Data defined override setup* for more details on this widget.

For the following examples the Regions layer of the QGIS sample dataset is used and selected as *Coverage layer* for the atlas generation. We assume that it is a single page layout containing a map item and a label item.

When the height (north-south) of a region extent is greater than its width (east-west), you should use *Portrait* instead of *Landscape* orientation to optimize the use of paper. With a  Data Defined Override button you can dynamically set the paper orientation.

Right-click on the page and select *Page Properties* to open the panel. We want to set the orientation dynamically, using an expression depending on the region geometry, so press the  button of field *Orientation*, select *Edit...* to open the *Expression string builder* dialog and enter the following expression:

```
CASE WHEN bounds_width(@atlas_geometry) > bounds_height(@atlas_geometry)
THEN 'Landscape' ELSE 'Portrait' END
```


Now if you *preview the atlas*, the paper orients itself automatically, but item placements may not be ideal. For each Region you need to reposition the location of the layout items as well. For the map item you can use the  button of its *Width* property to set it dynamic using the following expression:

```
@layout_pagewidth - 20
```

Likewise, use the  button of the *Height* property to provide the following expression to constrain map item size:

```
@layout_pageheight - 20
```

To ensure the map item is centered in the page, set its *Reference point* to the upper left radio button and enter 10 for its *X* and *Y* positions.

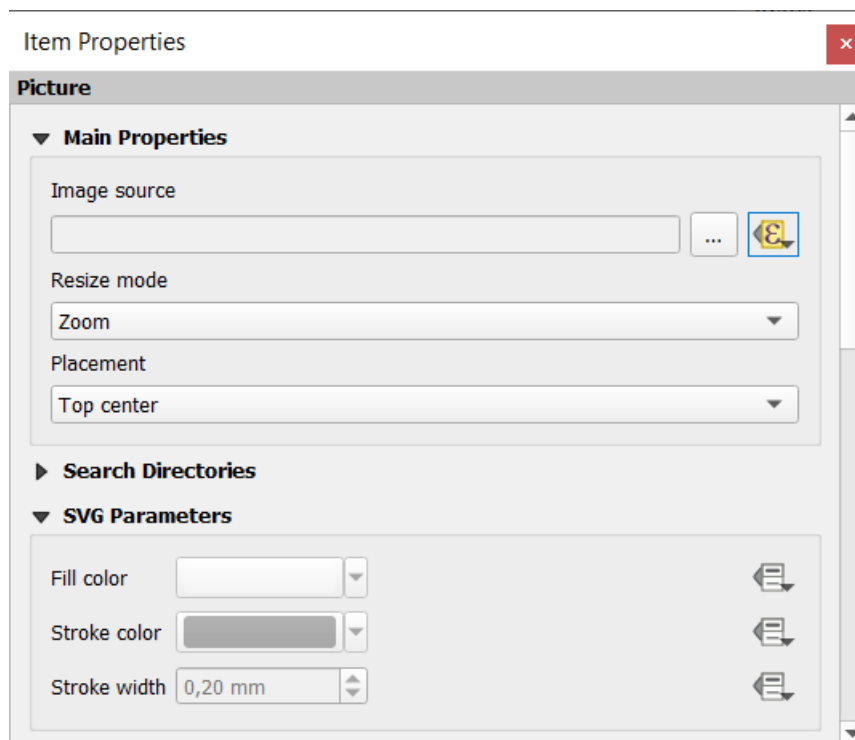
Let's add a title above the map in the center of the page. Select the label item and set the horizontal alignment to  *Center*. Next move the label to the right position, choose the middle button for the *Reference point*, and provide the following expression for field *X*:

```
@layout_pagewidth / 2
```

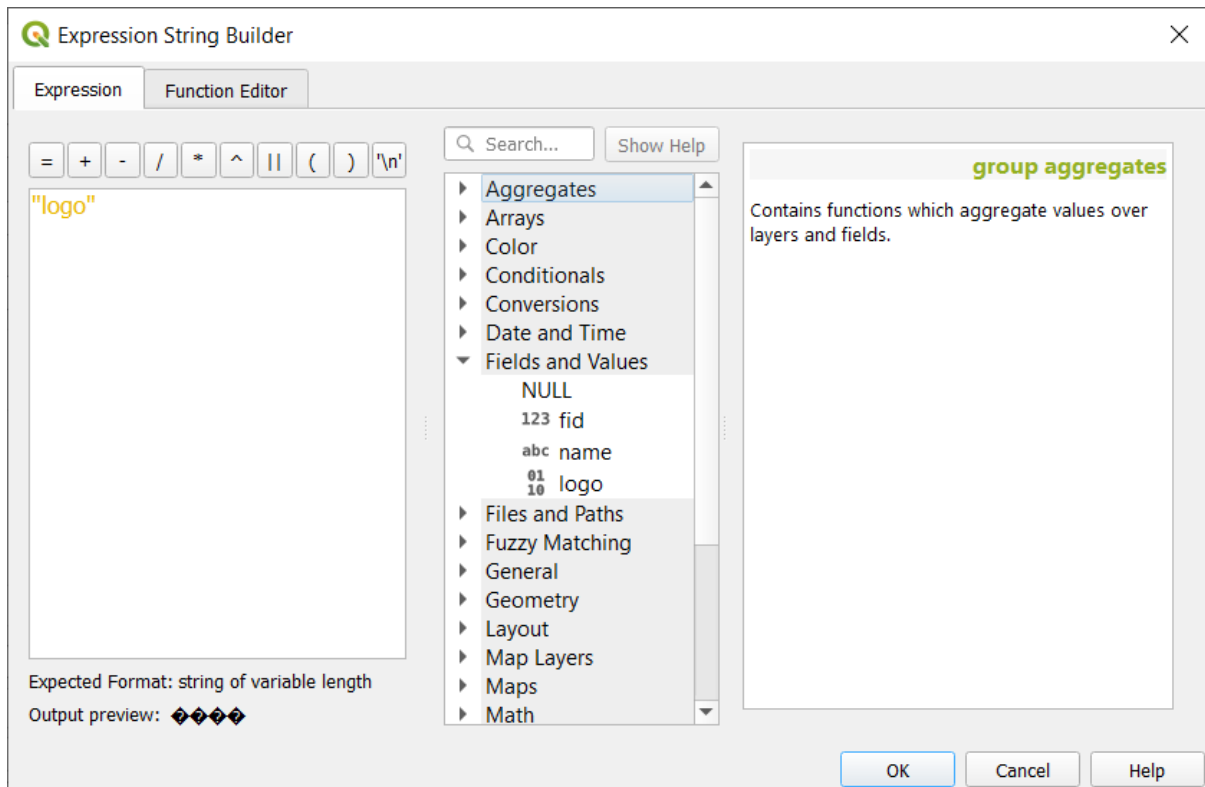
For all other layout items you can set the position in a similar way so they are correctly positioned both for portrait and landscape. You can also do more tweaks such as customizing the title with feature attributes (see [Customize labels with expression](#) example), changing images, resizing the number of legend columns number according to page orientation, ...

The information provided here is an update of the excellent blog (in English and Portuguese) on the Data Defined Override options [Multiple_format_map_series_using_QGIS_2.6](#) .

Another example for using data-defined override buttons is the usage of a dynamic picture. For the following examples we use a geopackage layer containing a BLOB field called `logo` with the field type binary (see [Creating a new GeoPackage layer](#)). For every feature there is defined a different picture so that the atlas can iterate over as described in [Preview and generate an atlas](#). All you need to do is add a picture in the print layout and go to its *Item properties* in the atlas context. There you can find a data-defined override button in the *Image source* section of the *Main Properties*.



In the following window choose *Edit* so that the *Expression String Builder* opens. From the *Fields and values* section you can find the BLOB field that was defined in the geopackage layer. Double-click the field name `logo` and click *OK*.




The atlas iterates over the entries in the BLOB field provided that you choose the geopackage layer as *Coverage layer* (further instructions you can find in *Preview and generate an atlas*).





These are just two examples of how you can use some advanced settings with atlas.

Preview and generate an atlas



图 18.63: Atlas Preview toolbar

Once the atlas settings have been configured, and layout items (map, table, image...) linked to it, you can create a preview of all the pages by choosing *Atlas* ► *Preview Atlas* or clicking the  *Preview Atlas* icon. You can then use the arrows to navigate through all the features:

-  First feature
-  Previous feature
-  Next feature
-  Last feature

You can also use the combo box to select and preview a specific feature. The combo box shows atlas feature names according to the expression set in the atlas *Page name* option.

As for simple compositions, an atlas can be generated in different ways (see *Creating an Output* for more information - just use tools from the *Atlas* menu or toolbar instead of the *Layout* menu).

This means that you can directly print your compositions with *Atlas* ► *Print Atlas*. You can also create a PDF using *Atlas* ► *Export Atlas as PDF...*: You will be asked for a directory to save all the generated PDF files, except if the *Single file export when possible* has been selected. In that case, you'll be prompted to give a filename.

With *Atlas* ► *Export Atlas as Images...* or *Atlas* ► *Export Atlas as SVG...* tool, you're also prompted to select a folder. Each page of each atlas feature composition is exported to the image file format set in *Atlas* panel or to SVG.

注解: With multi-page output, an atlas behaves like a layout in that only the page that contains the *General settings* will get a world file (for each feature output).

小技巧: Print a specific atlas feature

If you want to print or export the composition of only one feature of the atlas, simply start the preview, select the desired feature in the drop-down list and click on *Layout* ► *Print* (or *Export...* to any supported file format).

Use project defined relations for atlas creation

For users with HTML and Javascript knowledge it is possible to operate on GeoJSON objects and use project defined relations from the QGIS project. The difference between this approach and using expressions directly inserted into the HTML is that it gives you a full, unstructured GeoJSON feature to work with. This means that you can use existing Javascript libraries and functions that operate on GeoJSON feature representations.

The following code includes all related child features from the defined relation. Using the JavaScript `setFeature` function it allows you to make flexible HTML which represents relations in whatever format you like (lists, tables, etc). In the code sample, we create a dynamic bullet list of the related child features.

```
// Declare the two HTML div elements we will use for the parent feature id
// and information about the children
<div id="parent"></div>
<div id="my_children"></div>

<script type="text/javascript">
  function setFeature(feature)
  {
    // Show the parent feature's identifier (using its "ID" field)
    document.getElementById('parent').innerHTML = feature.properties.ID;
    //clear the existing relation contents
    document.getElementById('my_children').innerHTML = '';
    feature.properties.my_relation.forEach(function(child_feature) {
      // for each related child feature, create a list element
      // with the feature's name (using its "NAME" field)
      var node = document.createElement("li");
      node.appendChild(document.createTextNode(child_feature.NAME));
      document.getElementById('my_children').appendChild(node);
    });
  }
</script>
```

During atlas creation there will be an iteration over the coverage layer containing the parent features. On each page, you will see a bullet list of the related child features following the parent's identifier.

18.4 Creating a Report

This section will help you set up a report in QGIS.

18.4.1 What is it?

By definition, a GIS report is a document containing information organized in a narrative way, containing maps, text, graphics, tables, etc. A report can be prepared ad hoc, periodic, recurring, regular, or as required. Reports may refer to specific periods, events, occurrences, subjects or locations.

In QGIS, a *Report* is an extension of a *Layouts*.

Reports allow users to output their GIS projects in a simple, quick and structured way.

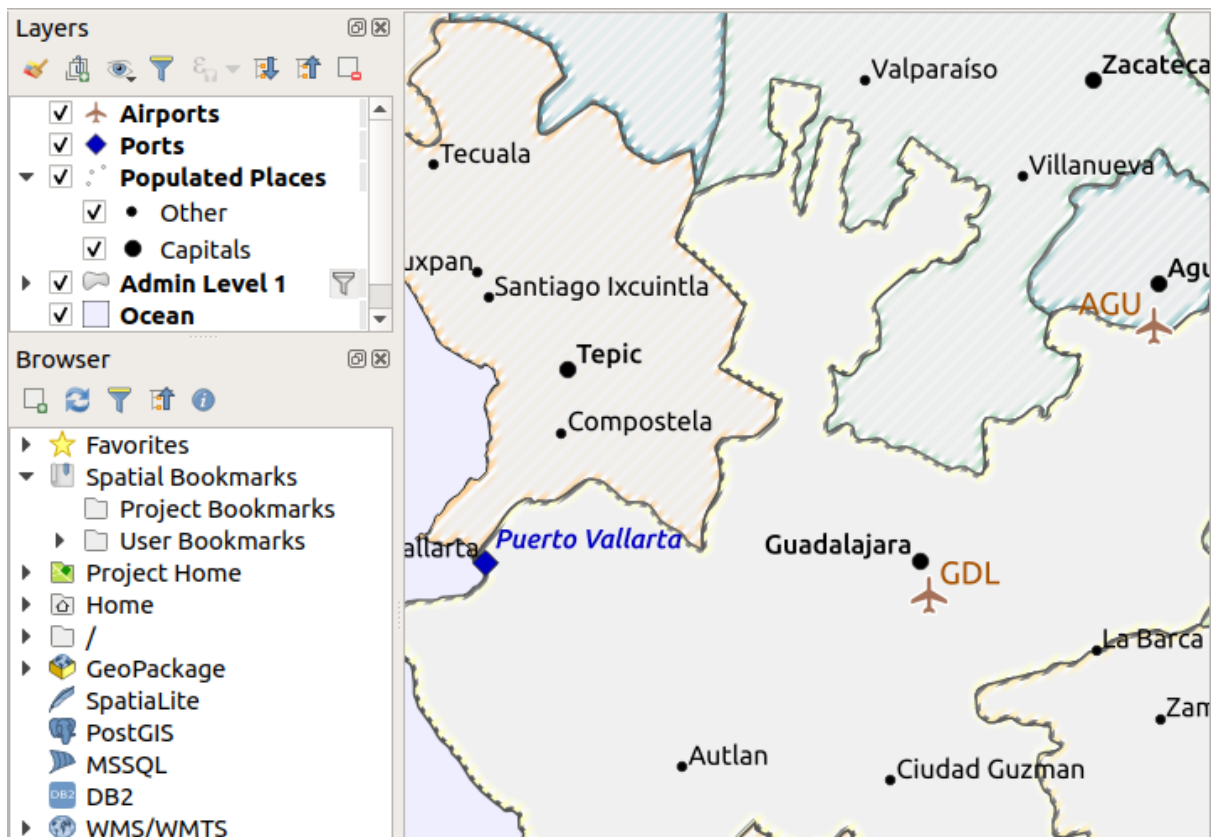
A report can be created with *Project* ► *New Report* or inside the *Project* ► *Layout Manager*.

注解: The maps in QGIS reports behave in the same way as maps in print layouts and atlases. We will concentrate on the specifics of QGIS reports. For details on map handling, see the sections on *print layouts* and *atlases*.

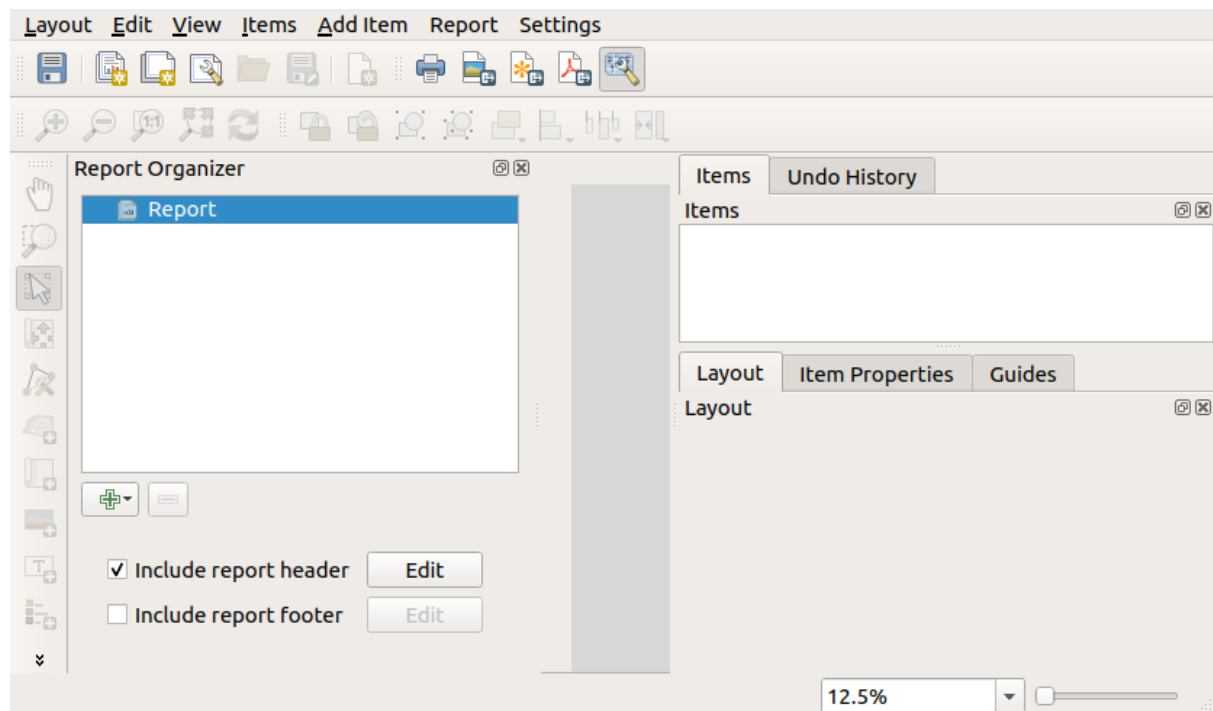
18.4.2 Get started

In the *Layout Manager* dialog a report can be created through *New from template* by selecting the dropdown option *Empty Report* and hitting the *Create...* button.

For this example, we use some administrative boundaries, populated places, ports and airports from the [Natural Earth dataset](#) (1:10M).

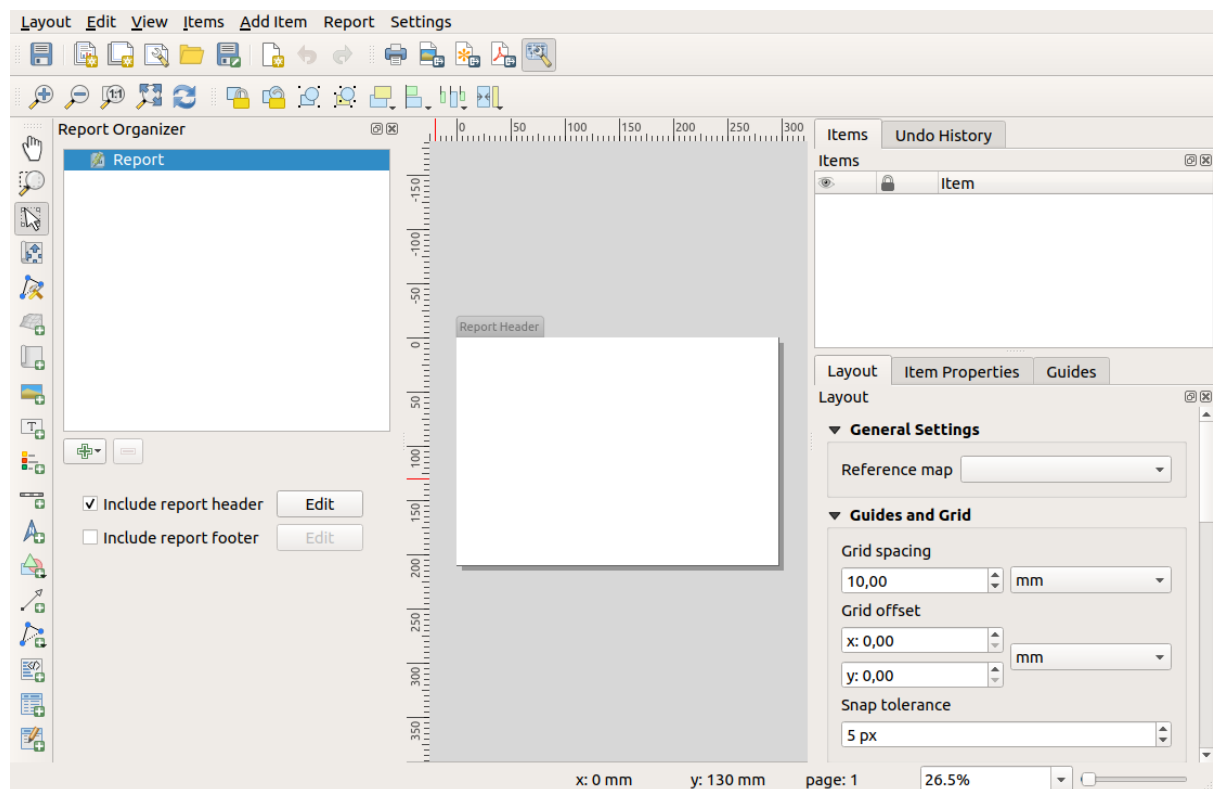


Using the *Project* ► *New Report* command, we create a blank report. Initially, there is not much to look at –the dialog which is displayed looks much like the print layout designer, except for the *Report Organizer* panel to the left:



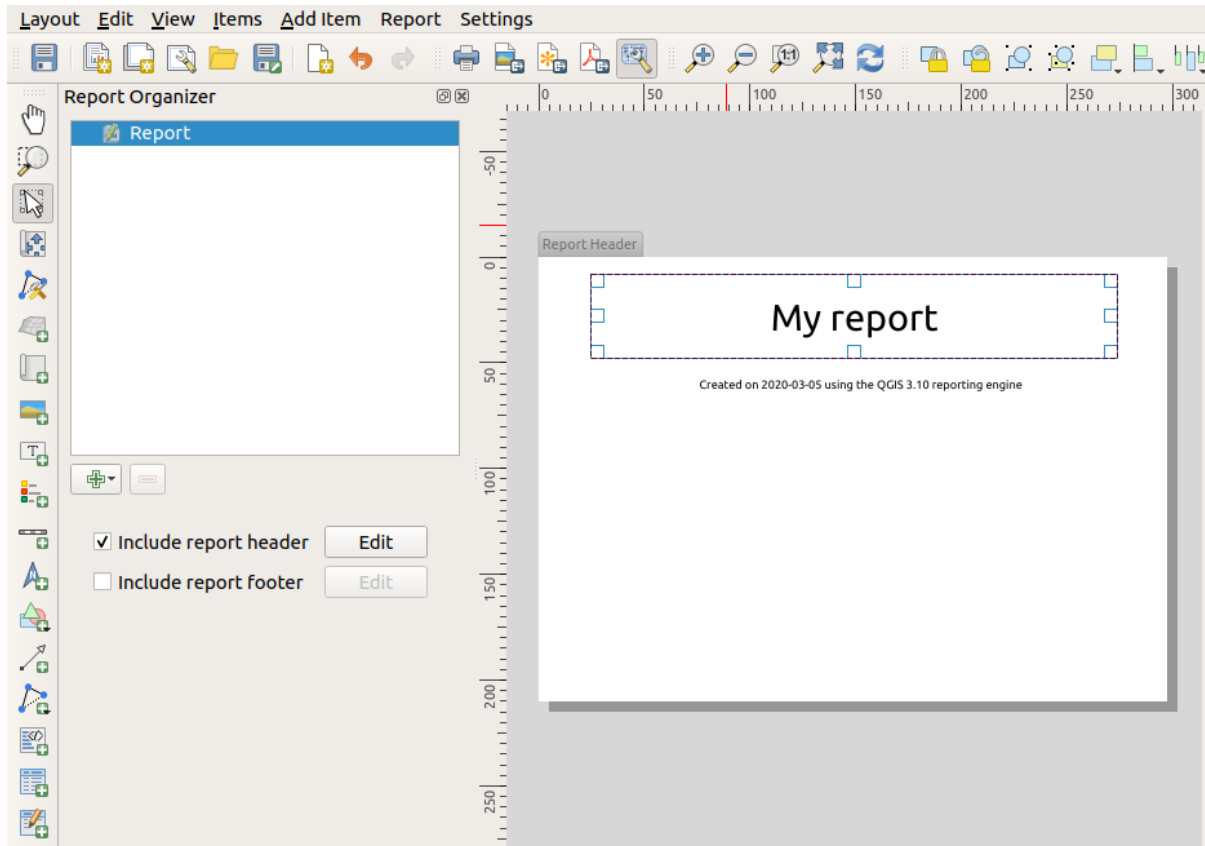
18.4.3 Layout Report Workspace

QGIS reports can consist of multiple, nested sections. In our new blank report we initially only have the main report section. The only options for this report section is *Include report header* and *Include report footer*. If we enable these options, a header will be included as the first page(s) (individual parts of reports can be multi-page if desired) in the report, and a footer will constitute the last page(s). Enable the header (*Include report header*), and hit the *Edit* button next to it:

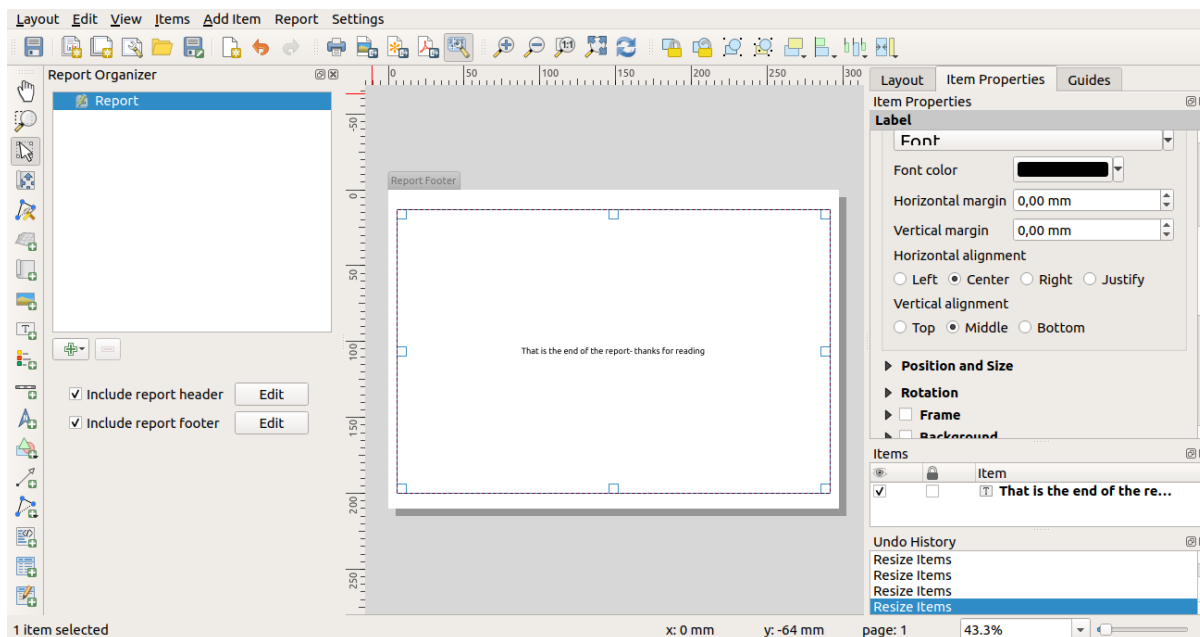


A few things happen as a result. Firstly, an edit pencil is shown next to *Report* in the *Report Organizer*, indicating that the report section is currently being edited in the designer. We also see a new page with a small *Report Header* title. The page has *landscape* orientation by default, but this (and other properties of the page) can be changed by right-clicking on the page and choosing *Page properties*. This will bring up the *Item properties* tab for the page, and page *Size*, *Width*, *Height*, and more can be specified.

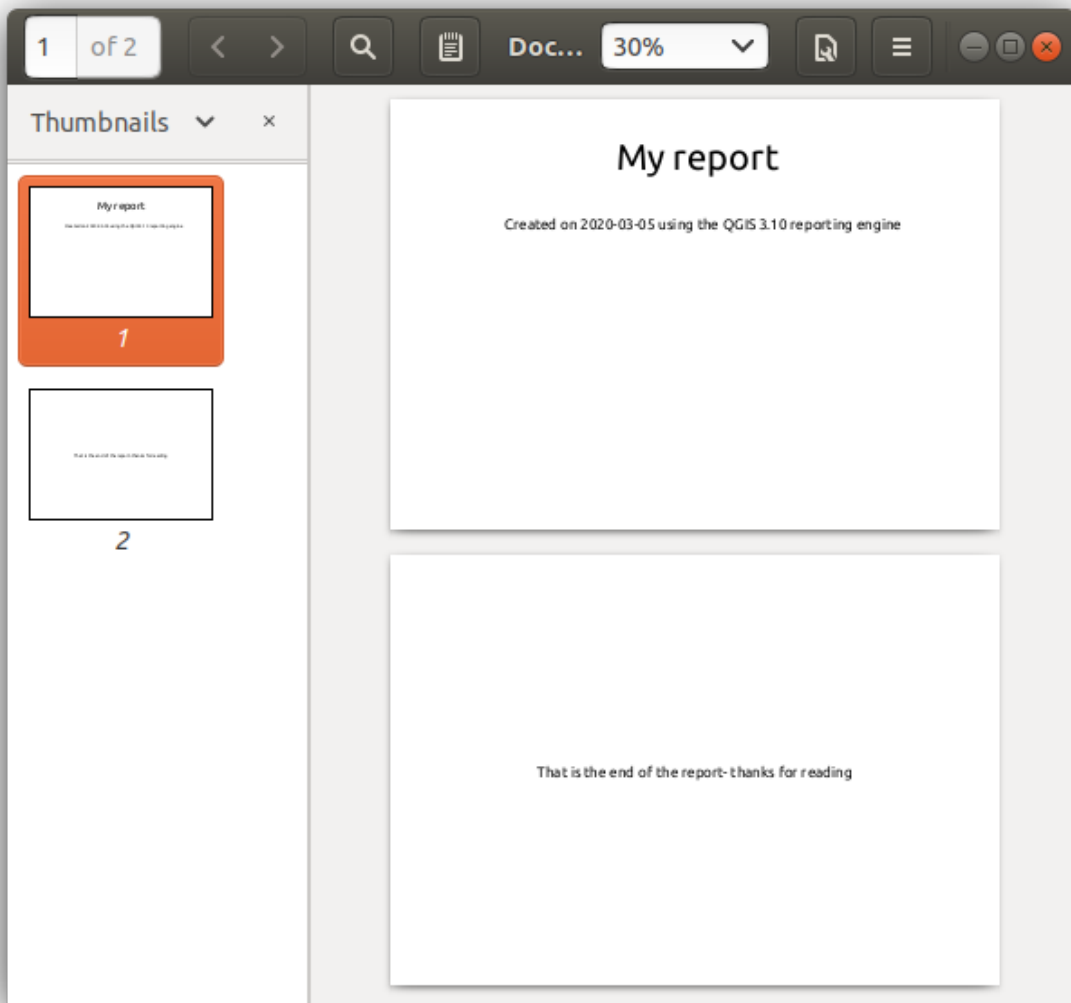
In QGIS reports, every component of the report is made up of individual layouts. They can be created and modified using the same tools as for standard print layouts –so you can use any desired combination of labels, pictures, maps, tables, etc. Let us add some items to our report header to demonstrate:



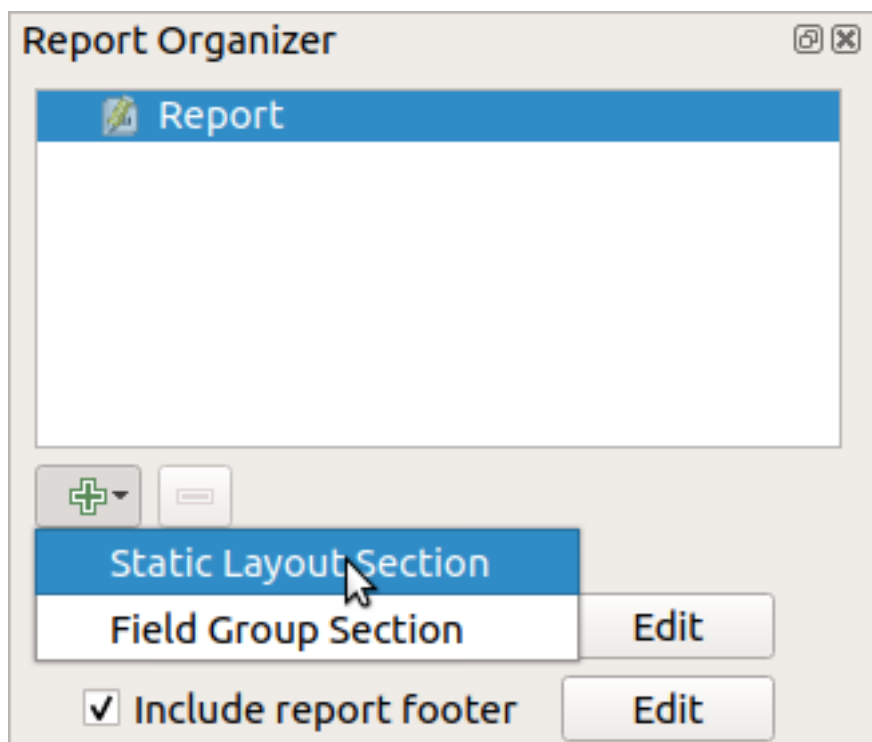
We will also create a simple footer for the report by checking the *Include report footer* option and hitting *Edit*.



Before proceeding further, let us export this report and see what we get. Exporting is done from the *Report* menu –in this case we select *Export Report as PDF...* to render the whole report to a PDF file. Here is the not-very-impressive result –a two page PDF consisting of our header and footer:



Let us make things more interesting. By hitting the  Add Section button in the *Report Organizer*, we are given a choice of new sections to add to our report.

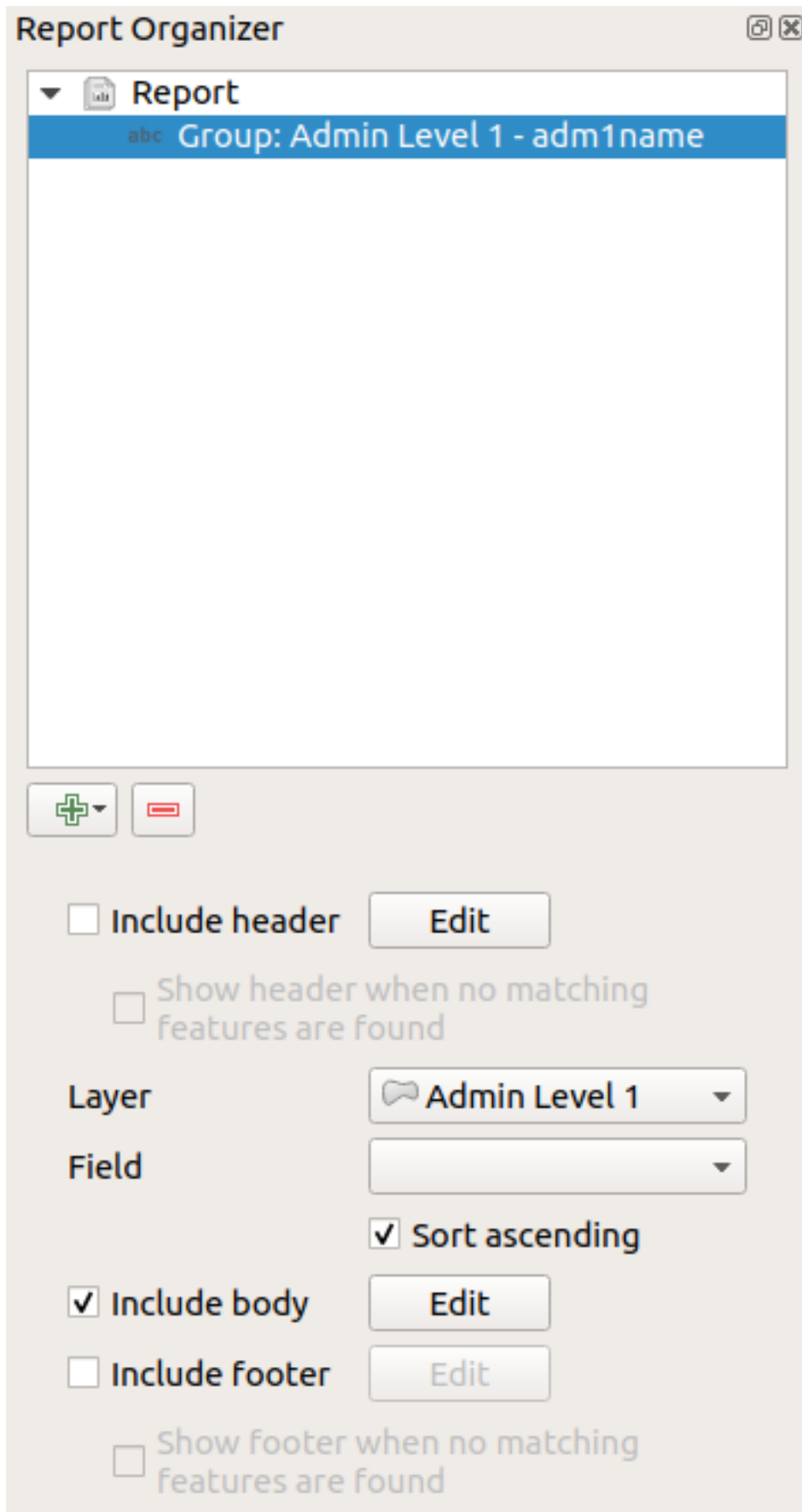


There are two options: *Static Layout Section* and *Field Group Section*.

The *Add Static Layout Section* is a single, static body layout. This can be used to embed static layouts mid-way through a report.

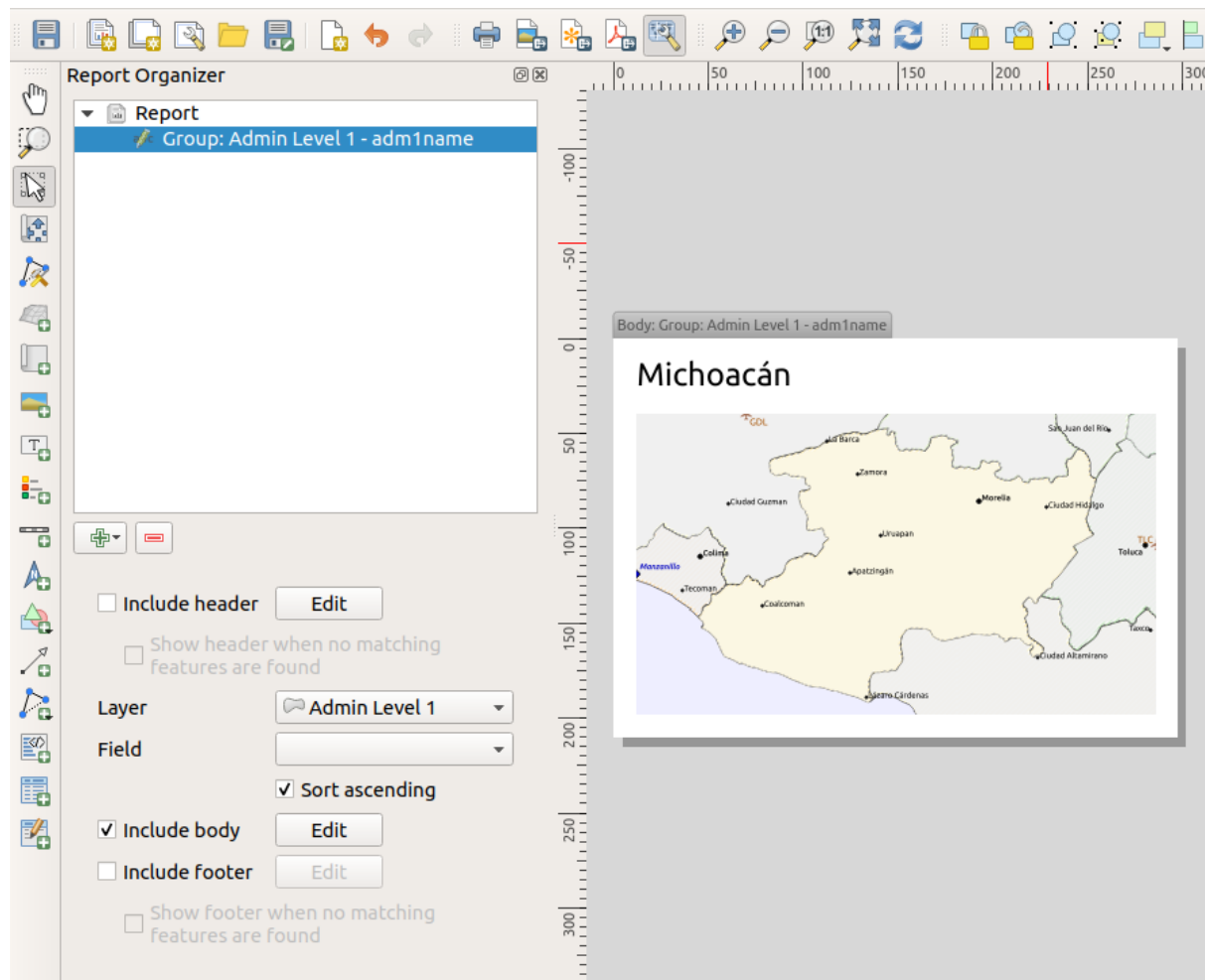
The *Field Group Section* repeats its body layout for every feature of a layer. The features are sorted by the selected grouping feature (with an option for ascending/descending sort). If a field group section has child sections (e.g. another field group section with a different field), then only features with unique values for the group feature are iterated over. This allows nested reports.

For now we will add a *Field Group Section* to our report. At its most basic level, you can think of a *Field Group Section* as the equivalent of a *print atlas*: you select a layer to iterate over, and the report will insert a section for each feature found. Selecting the new *Field Group Section* reveals a number of new related settings:



In this case we've setup our Field Group so that we iterate over all the states from the *Admin Level 1* layer, using

the values from the *adm1name* field. The same options to include header and footer are present, together with a new option to include a *body* for this section. We' ll do that, and edit the body:

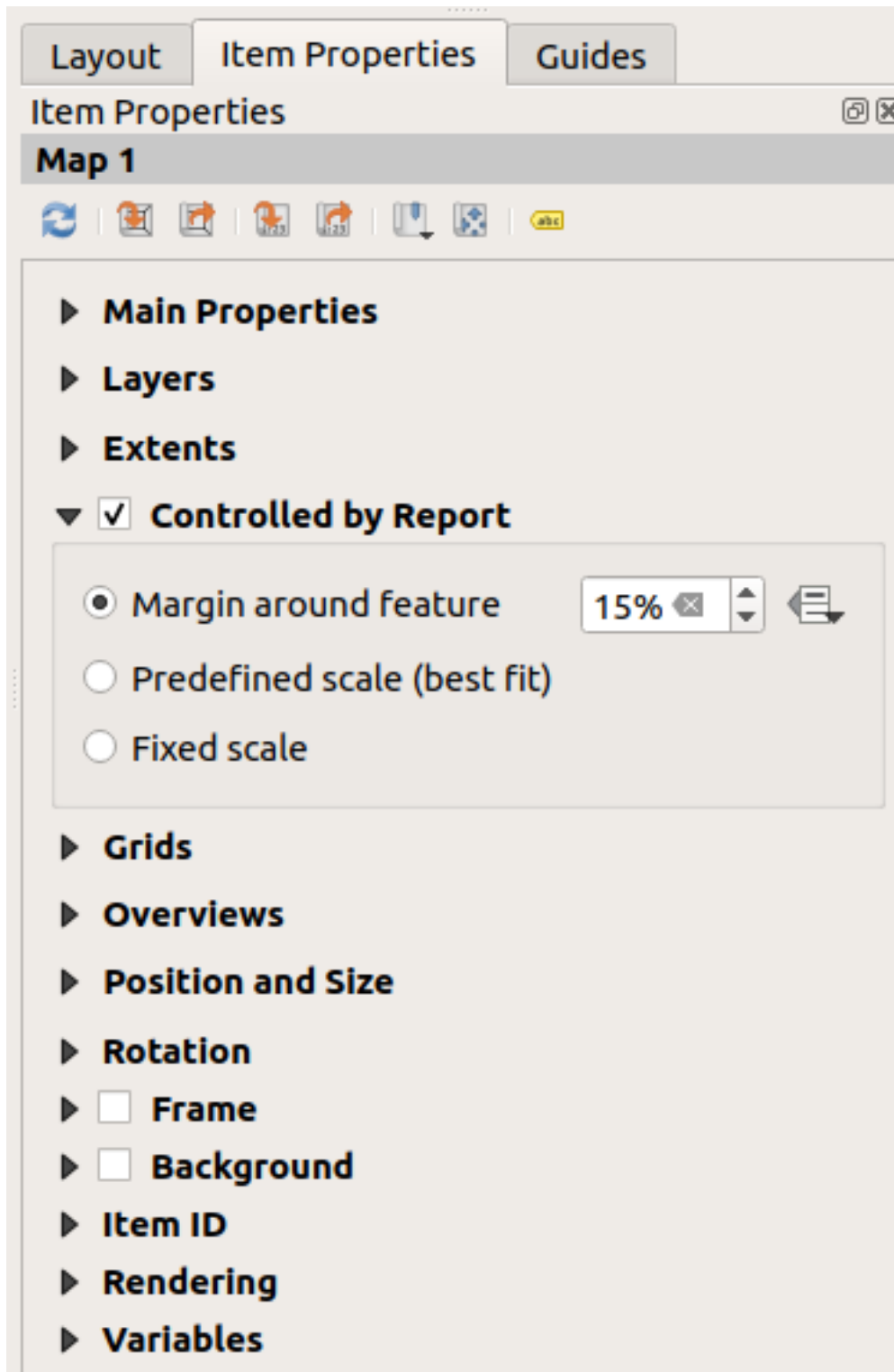


Our body now consists of a map and a label showing the name of the state. To include the name of the state, we selected *Add Item* ► *Add Label* and data defined the text under *Main Properties* with the help of *Insert or Edit an Expression...*

The result was the following expression (*name* is the name of the attribute in the *Admin Level 1* layer that contains the name of the state):

```
[% "name" %]
```

The map is set to follow the current report feature (enabled by checking *Controlled by Report* –just like a map item in an atlas will follow the current atlas feature when *Controlled by Atlas* is checked):



If we went ahead and exported our report now, we'd get something like this:

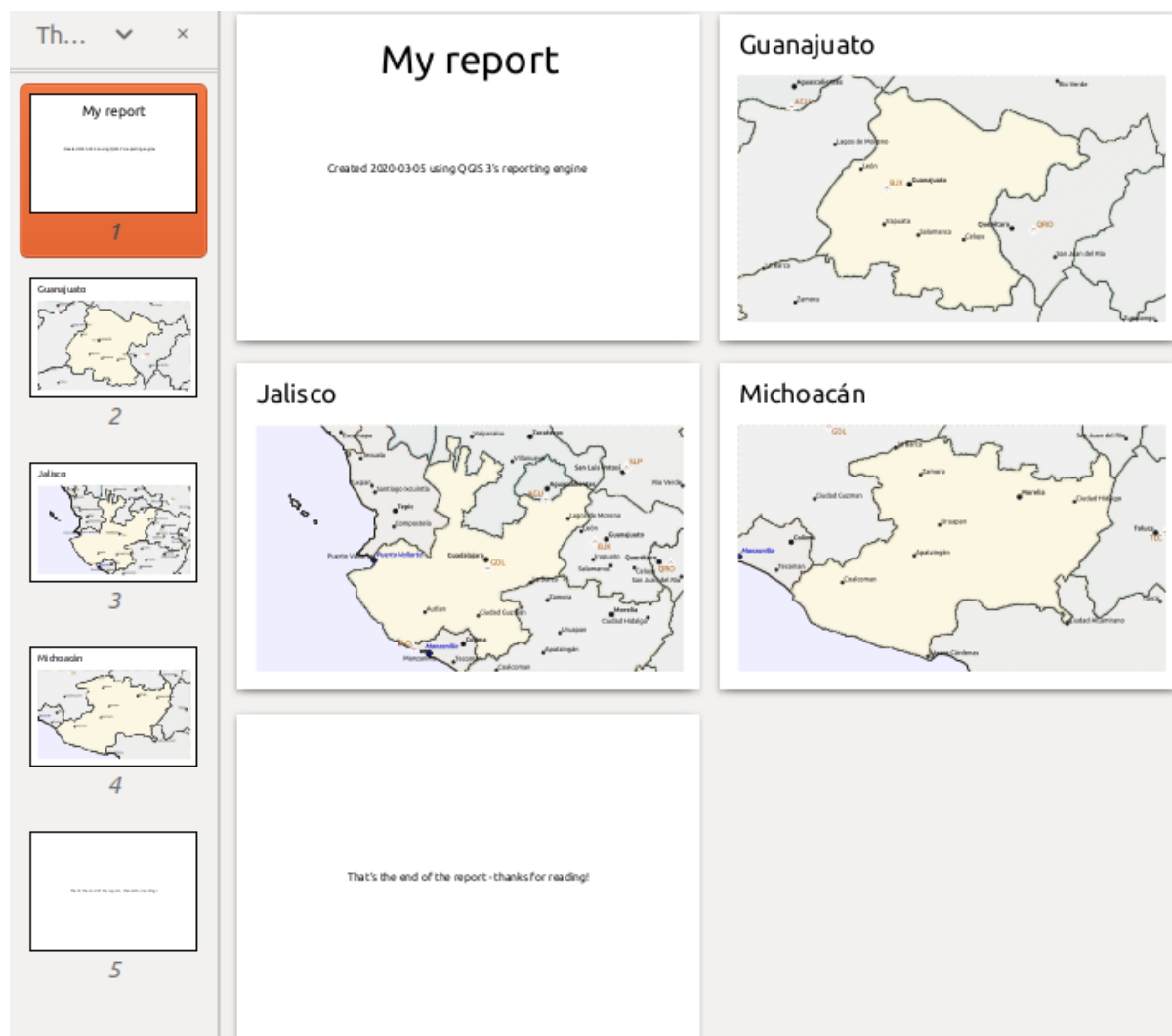
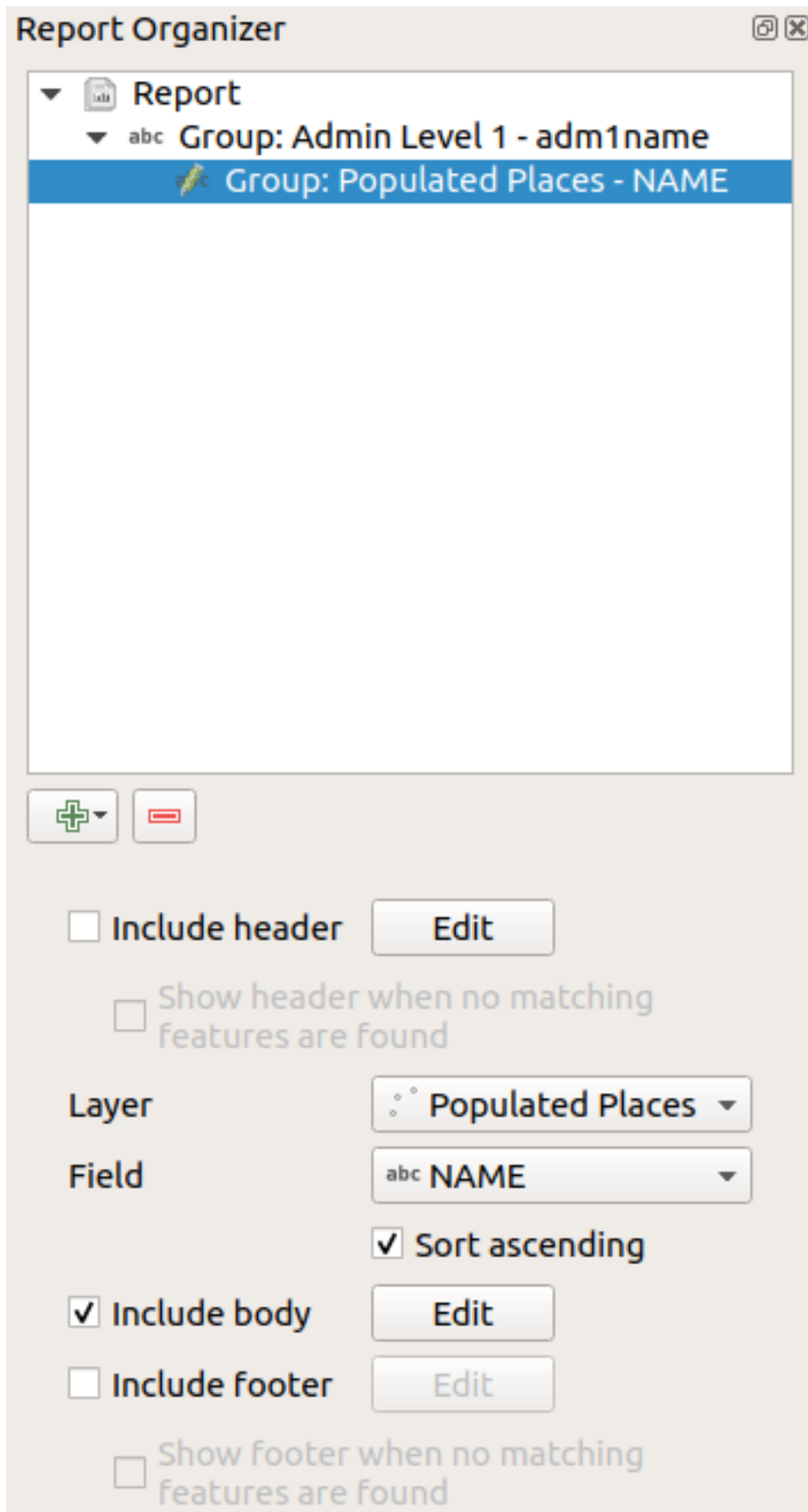


图 18.64: The report header, a page for each state, and the report footer.

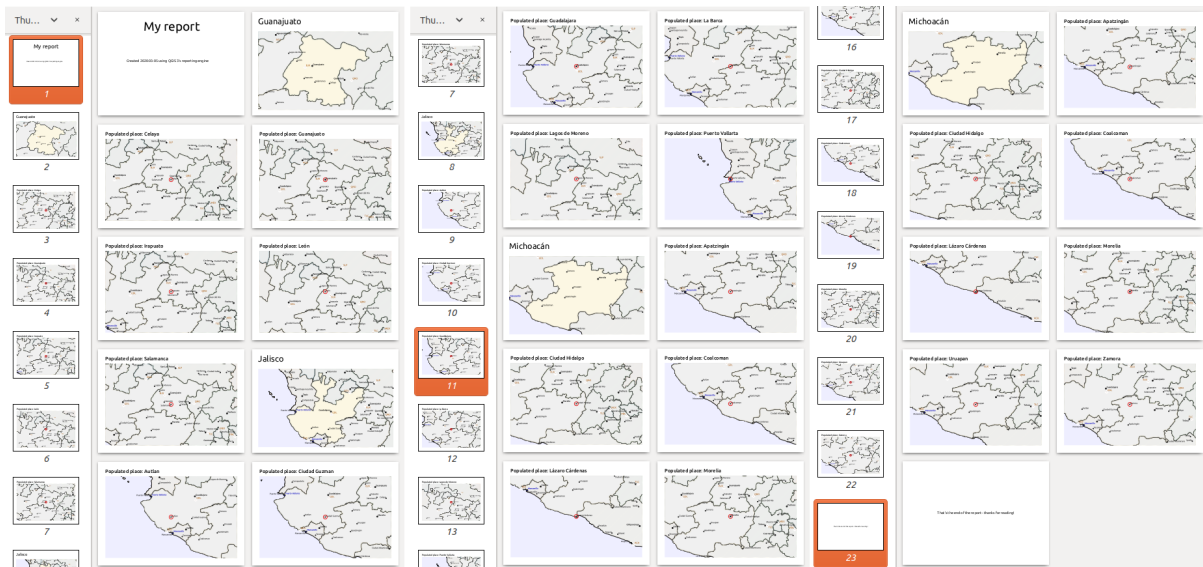
So more or less an atlas, but with a header and footer page.

Let us make things more interesting by adding a subsection to our state group. We do this by first selecting the *Admin Level 1* field group in the organizer, then hitting the  Add Field button and adding a new *Field Group Section*:



When iterating over the features of a *Field Group Section*, the features will be filtered to match the defining field

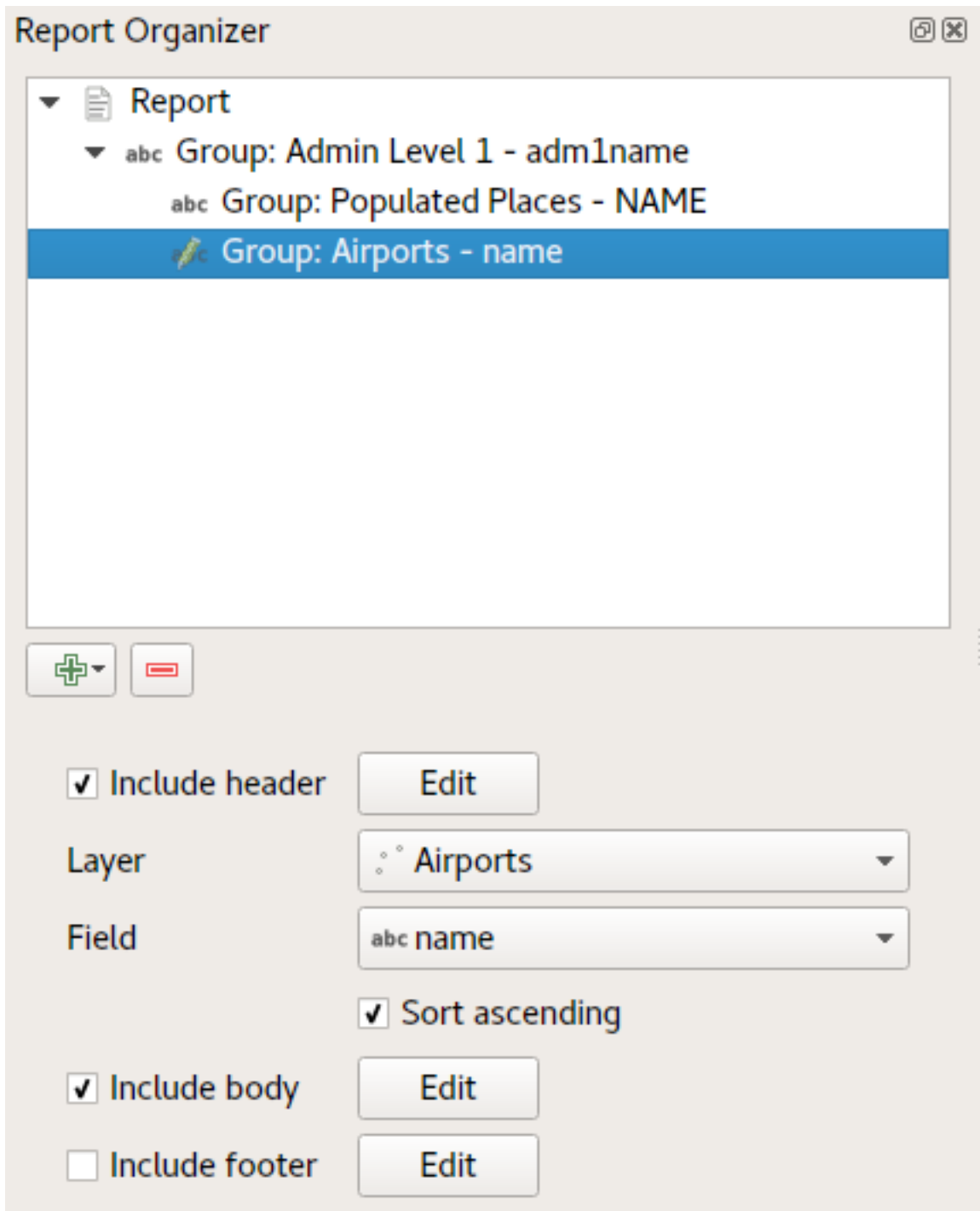
of its parent group (`adm1name` in this case). Here, the subsection we added will iterate over a *Populated Places* layer, including a body section for each place encountered. The magic here is that the *Populated Places* layer has an attribute with the same name as the defining field in the parent layer, `adm1name`, tagging each place with the state it is contained within (if you're lucky your data will already be structured like this –if not, run the *Join Attributes by Location* Processing algorithm and create your own field). When we export this report, QGIS will grab the first state from the *Admin Level 1* layer, and then iterate over all the *Populated Places* with a matching `adm1name` value. Here's what we get:



Here we created a basic body for the *Populated Places* group, including a map of the place and a table of some place attributes. So our report is now a report header, a page for the first state, followed by a page for every populated place within that state, then the rest of the states with their populated places, and finally the report footer. If we were to add a header for the *Populated Places* group, it would be included just before listing the populated places for each state, as shown in the illustration below.

Similarly, a footer for the *Populated Places* group would be inserted after the final place for each state is included.

In addition to nested subsections, subsections in a report can also be included consecutively. If we add a second subsection to the *Admin Level 1* group for *Airports*, then (if the *Airports* layer has an attribute `adm1name` that can link it to the parent group) our report will first list ALL the populated places for each state, followed by all the airports within that state, before proceeding to the next state.



The key point here is that our *Airports group* is a subsection of the *Admin Level 1 group* –not the *Populated Places group*.

In this case our report would be structured like this (note that state flags have also been included - the procedure for adding feature specific pictures in this way is described below):


The screenshot displays the QGIS Desktop 3.16 report engine interface. On the left, a vertical sidebar shows a list of report thumbnails, numbered 1 through 9. The main workspace is a grid of report panels. The top-left panel is titled 'My report' and contains the text 'Created: 2020-03-05 using QGIS 3's report engine'. To its right is a map of Guanajuato. Below these are several panels for 'Populated places in Guanajuato', each showing a map with a specific location highlighted: Celaya, Guanajuato, Irapuato, León, and Salamanca. Further down, there is a panel for 'Airports in Guanajuato' and a map of 'Airport Del Bajío Int'l (BJX)'. The bottom section of the report features a map of Jalisco, followed by panels for 'Populated places in Jalisco' and specific locations: Autlan and Ciudad Guzman.

Including pictures in a report

Pictures can be quite useful in reports, and QGIS allows pictures in both the static and dynamic parts of a report. Pictures are added in the same way as for standard print layouts, and for the static report parts (and static pictures in dynamic parts) there is not more to it.

But if you want illustrations that are tailored to the report features, your layer must have an attribute that can be used to define the picture to include.

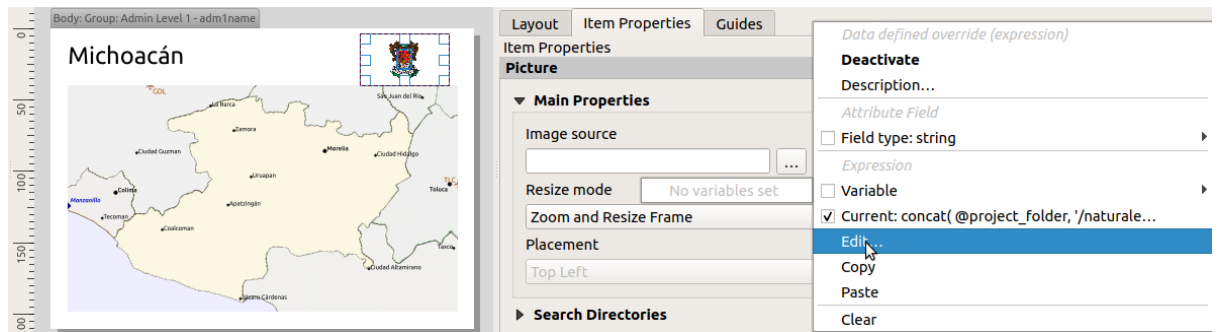
QGIS depends on absolute file names for images in reports.

For dynamic pictures, you first add a picture to the body part of the group, as usual. In the *Item properties* of the picture, you set the *Image Source* using the  Data defined override button, and either select an attribute that contains the absolute path of the images or *Edit...* (to enter an expression that generates the absolute image path).

Below is an example expression that uses string concatenation to specify the absolute path to the pictures, using the directory where the project file is located (@project_path) and an attribute (admlname) from which the file name is generated (in this case by transforming the string in the admlname attribute to uppercase, and appending '_flag.png'):

```
concat (@project_folder, '/natureearth/pictures/' ,
        upper ("admlname"), '_flag.png')
```

This means that the pictures are located in the natureearth/pictures subdirectory of the project file directory.



Highlighting the current report feature in a map

In the above report, the report features are emphasized in the maps using highlighting (state) and circles (populated places). To emphasize the report features in the maps (apart from placing them at the centre of the maps), you must data define the style using a comparison between its @id and the @atlas_featureid, as for atlases.

For instance, if you would like to use a thicker line / border for the report feature than the other features you can data define the line width:

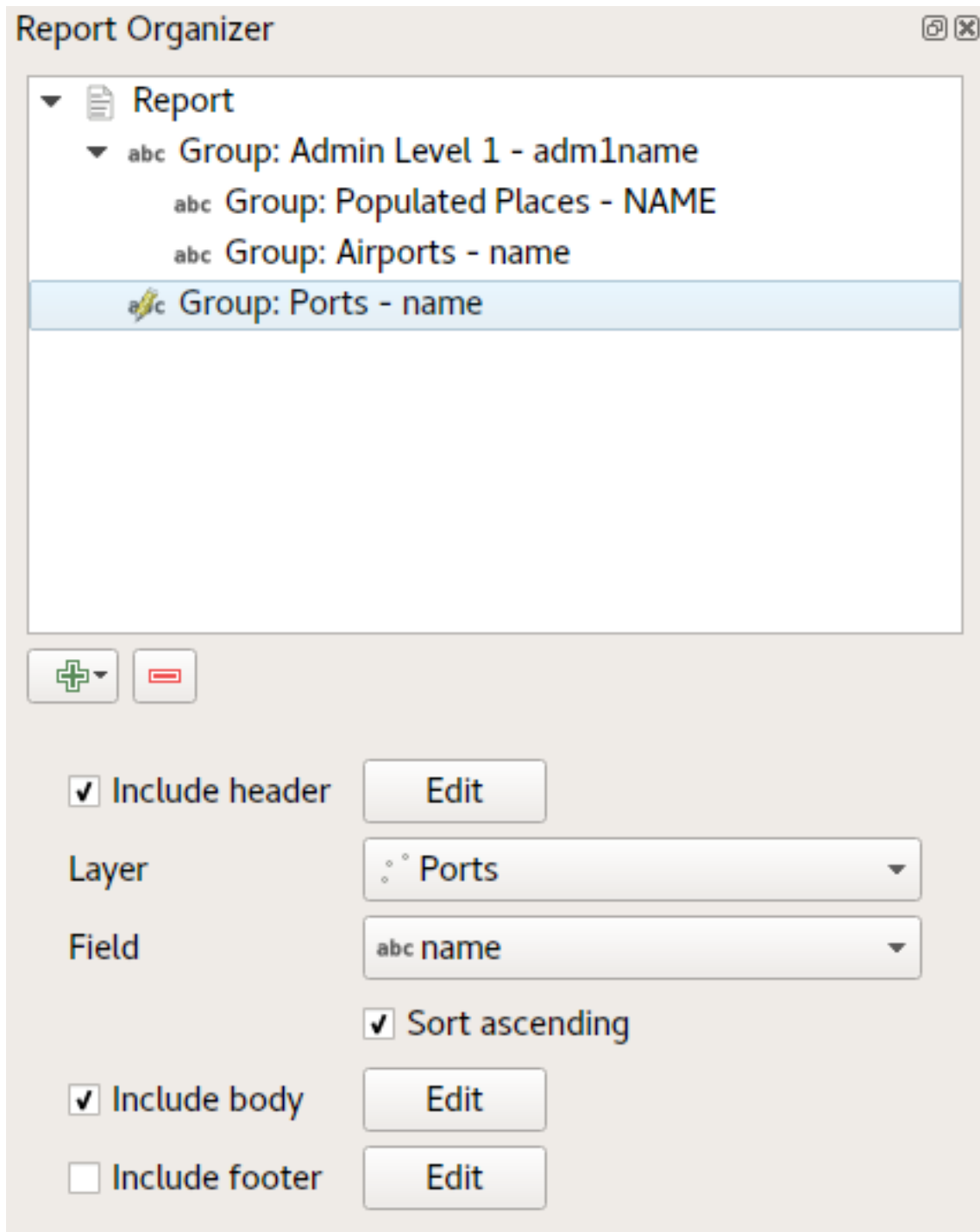
```
if($id=@atlas_featureid, 2.0, 0.1)
```

The report feature will get a 2 units wide polygon outline, while all other features will get a 0.1 units wide line. It is also possible to data define the colour (non-transparent dark magenta for the report feature and semi-transparent light gray for the other features):

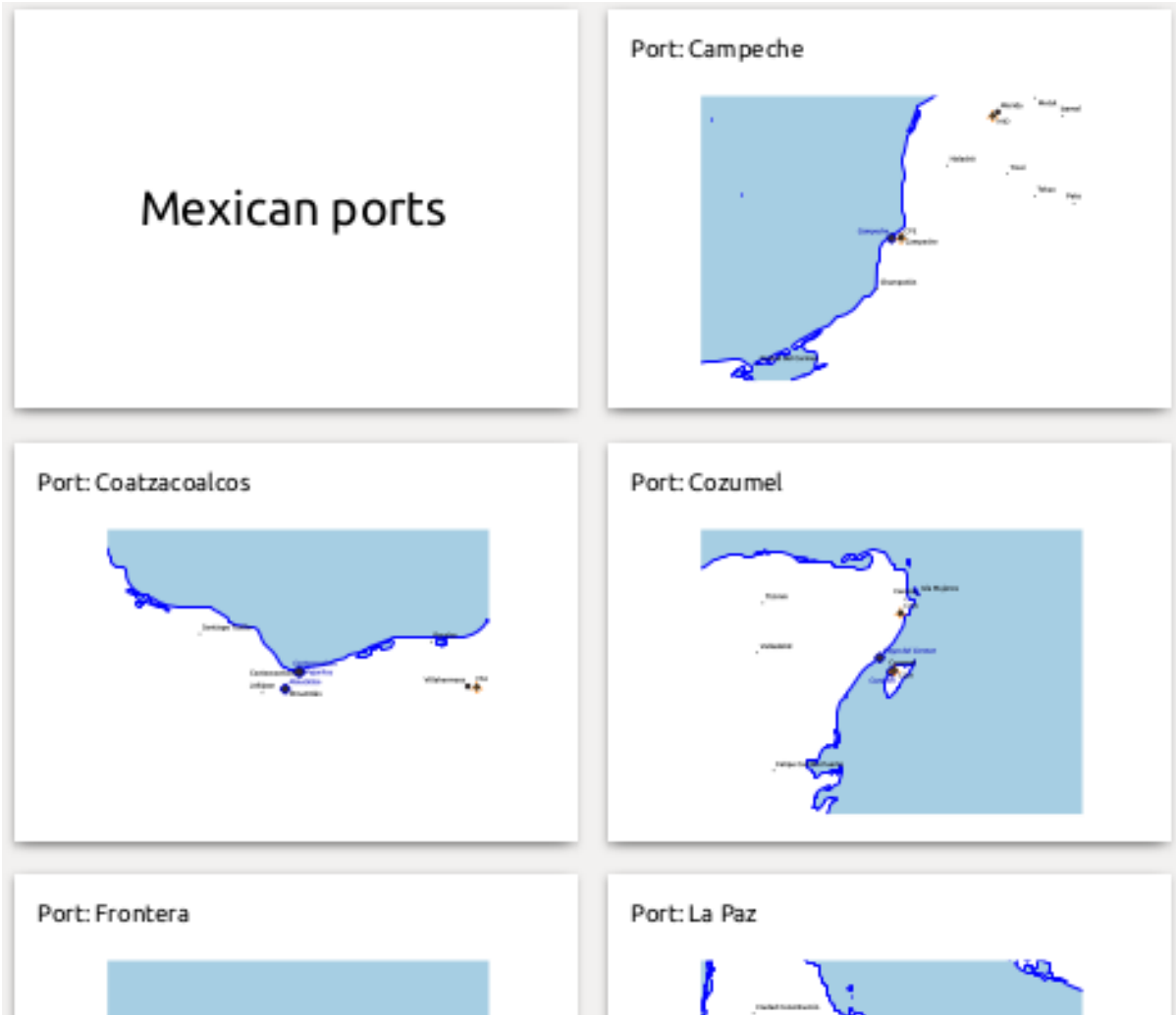
```
if($id=@atlas_featureid, '#FF880088', '#88CCCC')
```

More level 1 groups

Combining nested and consecutive sections, together with section headers and footers allows for tons of flexibility. For instance, in the below report we add another field group as a child of the main report for the :guiabel'Ports' layer. Now, after listing the states together with their populated places and airports, we' ll get a summary list of all the ports in the region:



This results in the last part of our report exporting as:



18.4.4 Export settings

When you export a report (*Report* ► *Export Report as Images...* / *SVG...* / *PDF...*), you will be asked for a file name, and then you get the opportunity to tune the export settings to get the most appropriate output.

As you see, reports in QGIS are extremely powerful and flexible!

注解: The current information was adapted from a North Road blog, [Exploring Reports in QGIS 3.0 - the Ultimate Guide!](#)

Working with OGC / ISO protocols

The Open Geospatial Consortium (OGC) is an international organization with membership of more than 300 commercial, governmental, nonprofit and research organizations worldwide. Its members develop and implement standards for geospatial content and services, GIS data processing and exchange.

Describing a basic data model for geographic features, an increasing number of specifications are developed by OGC to serve specific needs for interoperable location and geospatial technology, including GIS. Further information can be found at <https://www.opengeospatial.org/>.

Important OGC specifications supported by QGIS are:

- **WMS** --- Web Map Service (*WMS/WMTS Client*)
- **WMTS** --- Web Map Tile Service (*WMS/WMTS Client*)
- **WFS** --- Web Feature Service (*WFS and WFS-T Client*)
- **WFS-T** --- Web Feature Service - Transactional (*WFS and WFS-T Client*)
- **WCS** --- Web Coverage Service (*WCS Client*)
- **WPS** --- Web Processing Service
- **CSW** --- Catalog Service for the Web
- **SFS** --- Simple Features for SQL (*PostGIS Layers*)
- **GML** --- Geography Markup Language

OGC services are increasingly being used to exchange geospatial data between different GIS implementations and data stores. QGIS can deal with the above specifications as a client, being **SFS** (through support of the PostgreSQL / PostGIS data provider, see section *PostGIS Layers*).

You can also share your maps and data through the WMS, WMTS, WFS, WFS-T and WCS protocols using a web-server with QGIS Server, UMN MapServer or GeoServer installed.

19.1 WMS/WMTS Client

19.1.1 Overview of WMS Support

QGIS currently can act as a WMS client that understands WMS 1.1, 1.1.1 and 1.3 servers. In particular, it has been tested against publicly accessible servers such as DEMIS.

A WMS server acts upon requests by the client (e.g., QGIS) for a raster map with a given extent, set of layers, symbolization style, and transparency. The WMS server then consults its local data sources, rasterizes the map, and sends it back to the client in a raster format. For QGIS, this format would typically be JPEG or PNG.

WMS is generically a REST (Representational State Transfer) service rather than a full-blown Web service. As such, you can actually take the URLs generated by QGIS and use them in a web browser to retrieve the same images that QGIS uses internally. This can be useful for troubleshooting, as there are several brands of WMS server on the market and they all have their own interpretation of the WMS standard.

WMS layers can be added quite simply, as long as you know the URL to access the WMS server, you have a serviceable connection to that server, and the server understands HTTP as the data transport mechanism.

Additionally, QGIS will cache your WMS responses (i.e. images) for 24h as long as the GetCapabilities request is not triggered. The GetCapabilities request is triggered everytime the *Connect* button in the *Add Layer(s) from WMS(T) Server* dialog is used to retrieve the WMS server capabilities. This is an automatic feature meant to optimize project loading time. If a project is saved with a WMS layer, the corresponding WMS tiles will be loaded from the cache the next time the project is opened as long as they are no older than 24H.

19.1.2 Overview of WMTS Support

QGIS can also act as a WMTS client. WMTS is an OGC standard for distributing tile sets of geospatial data. This is a faster and more efficient way of distributing data than WMS because with WMTS, the tile sets are pre-generated, and the client only requests the transmission of the tiles, not their production. A WMS request typically involves both the generation and transmission of the data. A well-known example of a non-OGC standard for viewing tiled geospatial data is Google Maps.

In order to display the data at a variety of scales close to what the user might want, the WMTS tile sets are produced at several different scale levels and are made available for the GIS client to request them.

This diagram illustrates the concept of tile sets:

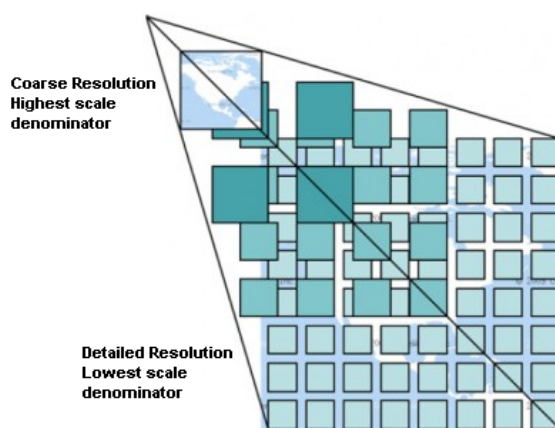


图 19.1: Concept of WMTS tile sets

The two types of WMTS interfaces that QGIS supports are via Key-Value-Pairs (KVP) and RESTful. These two interfaces are different, and you need to specify them to QGIS differently.

1. In order to access a **WMTS KVP** service, a QGIS user must open the WMS/WMTS interface and add the following string to the URL of the WMTS tile service:

```
"?SERVICE=WMTS&REQUEST=GetCapabilities"
```

An example of this type of address is

```
https://opencache.statkart.no/gatekeeper/gk/gk.open_wmts?\  
service=WMTS&request=GetCapabilities
```

For testing the topo2 layer in this WMTS works nicely. Adding this string indicates that a WMTS web service is to be used instead of a WMS service.

- The **RESTful WMTS** service takes a different form, a straightforward URL. The format recommended by the OGC is:

```
{WMTSBaseURL}/1.0.0/WMTSCapabilities.xml
```

This format helps you to recognize that it is a RESTful address. A RESTful WMTS is accessed in QGIS by simply adding its address in the WMS setup in the URL field of the form. An example of this type of address for the case of an Austrian basemap is <https://maps.wien.gv.at/basemap/1.0.0/WMTSCapabilities.xml>.





注解: You can still find some old services called WMS-C. These services are quite similar to WMTS (i.e., same purpose but working a little bit differently). You can manage them the same as you do WMTS services. Just add `?tiled=true` at the end of the url. See https://wiki.osgeo.org/wiki/Tile_Map_Service_Specification for more information about this specification.


When you read WMTS, you can often think WMS-C also.

19.1.3 Selecting WMS/WMTS Servers

The first time you use the WMS feature in QGIS, there are no servers defined.

You then need to create connections to the server you are targeting:

- Go to the  **WMS/WMTS** tab of the *Data Source Manager* dialog, either by:
 - clicking the  **Open Data Source Manager** button (or pressing `Ctrl+L`) and enabling the tab
 - clicking the  **Add WMS layer** button on the *Manage Layers* toolbar
 - or selecting *Layer* ► *Add Layer* ►  **Add WMS/WMTS Layer...** menu
- Press *New* from the *Layers* tab. The *Create a New WMS/WMTS Connection...* dialog appears.

小技巧: Right-click the  **WMS/WMTS** entry from within the *Browser panel* and select *New Connection...* also opens the *Create a New WMS/WMTS Connection...* dialog.

- Then enter the parameters to connect to your desired WMS server, as listed below:

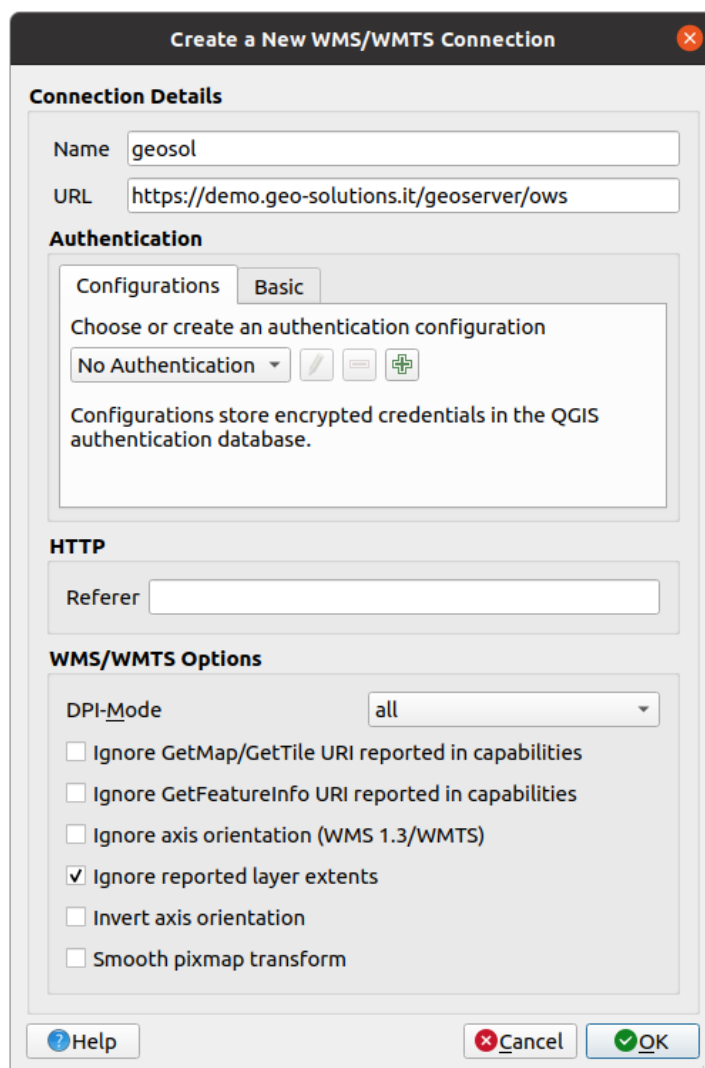


图 19.2: Creating a connection to a WMS server

- *Name*: A name for the connection. This name will be used in the Server Connections drop-down box so that you can distinguish it from other WMS servers.
- *URL*: URL of the server providing the data. This must be a resolvable host name -- the same format as you would use to open a telnet connection or ping a host, i.e. the base URL only. For example, you shouldn't have fragments such as `request=GetCapabilities` or `version=1.0.0` in your URL.
- *Authentication* (optional): using a *stored configuration* or a basic authentication with *Username* and *Password*.


警告: Entering **username** and **password** in the *Authentication* tab will keep unprotected credentials in the connection configuration. Those **credentials will be visible** if, for instance, you shared the project file with someone. Therefore, it's advisable to save your credentials in a *Authentication configuration* instead (*configurations* tab). See [认证系统](#) for more details.

- *HTTP Referer*
- *DPI-Mode*: Available options are **all**, **off**, **QGIS**, **UMN** and **GeoServer**
- *Ignore GetMap/GetTile URI reported in capabilities*: if checked, use given URI from the *URL* field above.

- *Ignore GetFeatureInfo URI reported in capabilities*: if checked, use given URI from the *URL* field above.
- *Ignore axis orientation (WMS 1.3/WMTS)*
- *Ignore reported layer extents*: because the extent reported by raster layers may be smaller than the actual area which can be rendered (notably for WMS servers with symbology which takes more space than the data extent), check this option to avoid cropping raster layers to their reported extents, resulting in truncated symbols on the borders of these layers.
- *Invert axis orientation*
- *Smooth pixmap transformation*


4. Press *OK*

Once the new WMS server connection has been created, it will be preserved for future QGIS sessions.

If you need to set up a proxy server to be able to receive WMS services from the internet, you can add your proxy server in the options. Choose *Settings* ► *Options* and click on the *Network* tab. There, you can add your proxy settings and enable them by setting *Use proxy for web access*. Make sure that you select the correct proxy type from the *Proxy type*  drop-down menu.

19.1.4 Loading WMS/WMTS Layers

Once you have successfully filled in your parameters, you can use the *Connect* button to retrieve the capabilities of the selected server. This includes the image encoding, layers, layer styles and projections. Since this is a network operation, the speed of the response depends on the quality of your network connection to the WMS server. While downloading data from the WMS server, the download progress is visualized in the lower left corner of the main QGIS dialog.

Your screen should now look a bit like , which shows the response provided by a WMS server.

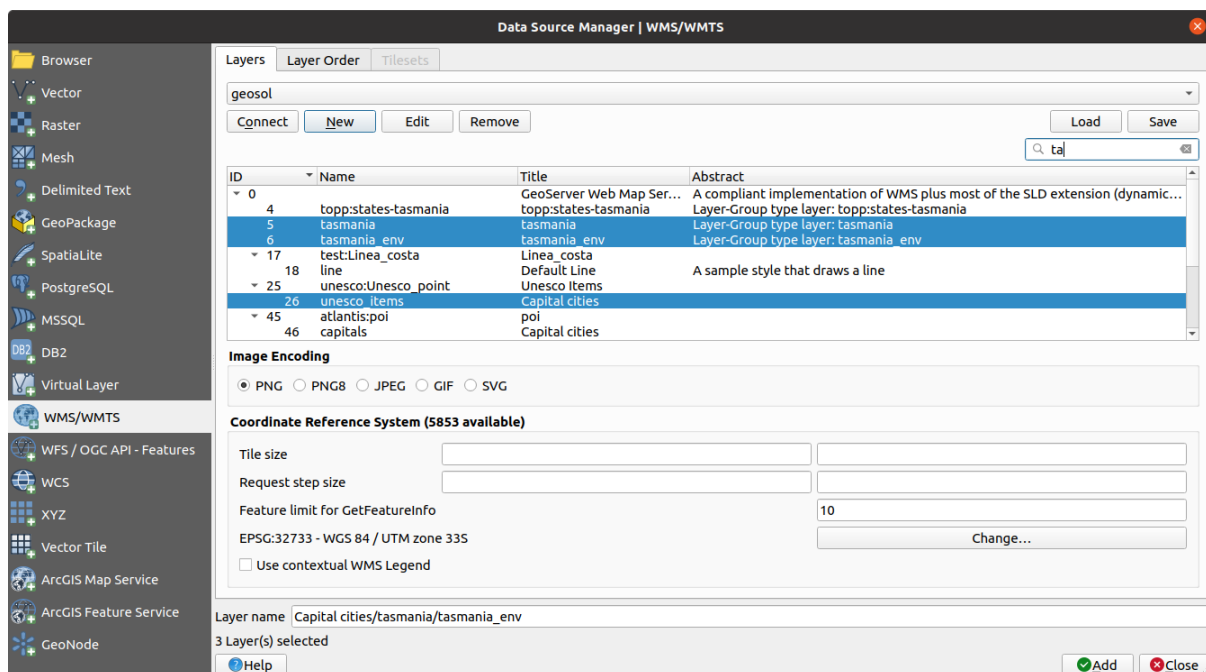


图 19.3: Dialog for adding a WMS server, with filter on available layers

The upper part of the *Layers* tab of the dialog shows a tree structure that can include layer groups embedding layers with their associated image style(s) served by the server. Each item can be identified by:

- an *ID*
- a *Name*
- a *Title*
- and an *Abstract*.


The list can be filtered using the  widget in the top right corner.

Image Encoding

The *Image encoding* section lists the formats that are supported by both the client and server. Choose one depending on your image accuracy requirements.

小技巧: Image Encoding

You will typically find that a WMS server offers you the choice of JPEG or PNG image encoding. JPEG is a lossy compression format, whereas PNG faithfully reproduces the raw raster data.

Use JPEG if you expect the WMS data to be photographic in nature and/or you don't mind some loss in picture quality. This trade-off typically reduces by five times the data transfer requirement compared with PNG.

Use PNG if you want precise representations of the original data and you don't mind the increased data transfer requirements.

Options

The Options area of the dialog provides means to configure the WMS requests. You can define:

- *Tile size* if you want to set tile sizes (e.g., 256x256) to split up the WMS request into multiple requests.
- The *Request step size*
- The *Feature limit for GetFeatureInfo* defines the maximum number of GetFeatureInfo results from the server.
- If you select a WMS from the list, a field with the default projection provided by the web server appears. Press the *Change...* button to replace the default projection of the WMS with another CRS supported by the WMS server.
- Finally you can activate *Use contextual WMS Legend* if the WMS Server supports this feature. Then only the relevant legend for your current map view extent will be shown and thus will not include legend items for items you can't see in the current map.

At the bottom of the dialog, a *Layer name* text field displays the selected item's *Title*. You can change the name at your will. This name will appear in the *Layers* panel after you pressed the *Add* button and loaded the layer(s) in QGIS.

You can select several layers at once, but only one image style per layer. When several layers are selected, they will be combined at the WMS server and transmitted to QGIS in one go, as a single layer. The default name is a slash (/) separated list of their original title.

Layer Order

The *Layer Order* tab lists the selected layers available from the current connected WMS server.

WMS layers rendered by a server are overlaid in the order listed in the *Layers* tab, from top to bottom of the list. If you want to change the overlay order, you can use the *Up* and *Down* buttons of the *Layer Order* tab.

Transparency

The *Global transparency* setting from the *Layer Properties* is hard coded to be always on, where available.

小技巧: WMS Layer Transparency

The availability of WMS image transparency depends on the image encoding used: PNG and GIF support transparency, whilst JPEG leaves it unsupported.

Coordinate Reference System

A coordinate reference system (CRS) is the OGC terminology for a QGIS projection.

Each WMS layer can be presented in multiple CRSs, depending on the capability of the WMS server.


To choose a CRS, select *Change...* and a dialog similar to the one shown in [图 10.3](#) will appear. The main difference with the WMS version of the dialog is that only those CRSs supported by the WMS server will be shown.

19.1.5 图块集


When using WMTS (Cached WMS) services like

```
https://opencache.statkart.no/gatekeeper/gk/gk.open_wmts?
  service=WMTS&request=GetCapabilities
```

you are able to browse through the *Tilesets* tab given by the server. Additional information like tile size, formats and supported CRS are listed in this table. In combination with this feature, you can use the tile scale slider by selecting

View ► *Panels* (or  *Settings* ► *Panels*), then choosing *Tile Scale Panel*. This gives you the available scales from the tile server with a nice slider docked in.


19.1.6 Using the Identify Tool

Once you have added a WMS server, and if any layer from a WMS server is queryable, you can then use the  Identify tool to select a pixel on the map canvas. A query is made to the WMS server for each selection made. The results of the query are returned in plain text. The formatting of this text is dependent on the particular WMS server used.

Format selection

If multiple output formats are supported by the server, a combo box with supported formats is automatically added to the identify results dialog and the selected format may be stored in the project for the layer.

GML format support

The  Identify tool supports WMS server response (GetFeatureInfo) in GML format (it is called Feature in the QGIS GUI in this context). If "Feature" format is supported by the server and selected, results of the Identify tool are vector features, as from a regular vector layer. When a single feature is selected in the tree, it is highlighted in the map and it can be copied to the clipboard and pasted to another vector layer. See the example setup of the UMN Mapserver below to support GetFeatureInfo in GML format.

```
# in layer METADATA add which fields should be included and define geometry_
↳ (example) :

"gml_include_items"    "all"
"ows_geometries"      "mygeom"
"ows_mygeom_type"     "polygon"

# Then there are two possibilities/formats available, see a) and b):

# a) basic (output is generated by Mapserver and does not contain XSD)
# in WEB METADATA define formats (example):
"wms_getfeatureinfo_formatlist" "application/vnd.ogc.gml,text/html"

# b) using OGR (output is generated by OGR, it is send as multipart and contains_
↳ XSD)
# in MAP define OUTPUTFORMAT (example):
OUTPUTFORMAT
  NAME "OGRGML"
```

(下页继续)

```
MIMETYPE "ogr/gml"
DRIVER "OGR/GML"
FORMATOPTION "FORM=multipart"
END

# in WEB METADATA define formats (example):
"wms_getfeatureinfo_formatlist" "OGRGML,text/html"
```

Viewing Properties

Once you have added a WMS server, you can view its properties by right-clicking on it in the legend and selecting *Properties*.

Metadata Tab

The tab *Metadata* displays a wealth of information about the WMS server, generally collected from the capabilities statement returned from that server. Many definitions can be gleaned by reading the WMS standards (see OPEN-GEOSPATIAL-CONSORTIUM in *Literature and Web References*), but here are a few handy definitions:

- **Server Properties**

- **WMS Version** --- The WMS version supported by the server.
- **Image Formats** --- The list of MIME-types the server can respond with when drawing the map. QGIS supports whatever formats the underlying Qt libraries were built with, which is typically at least `image/png` and `image/jpeg`.
- **Identity Formats** --- The list of MIME-types the server can respond with when you use the Identify tool. Currently, QGIS supports the `text-plain` type.

- **Layer Properties**

- **Selected** --- Whether or not this layer was selected when its server was added to this project.
- **Visible** --- Whether or not this layer is selected as visible in the legend (not yet used in this version of QGIS).
- **Can Identify** --- Whether or not this layer will return any results when the Identify tool is used on it.
- **Can be Transparent** --- Whether or not this layer can be rendered with transparency. This version of QGIS will always use transparency if this is `Yes` and the image encoding supports transparency.
- **Can Zoom In** --- Whether or not this layer can be zoomed in by the server. This version of QGIS assumes all WMS layers have this set to `Yes`. Deficient layers may be rendered strangely.
- **Cascade Count** --- WMS servers can act as a proxy to other WMS servers to get the raster data for a layer. This entry shows how many times the request for this layer is forwarded to peer WMS servers for a result.
- **Fixed Width, Fixed Height** --- Whether or not this layer has fixed source pixel dimensions. This version of QGIS assumes all WMS layers have this set to `nothing`. Deficient layers may be rendered strangely.
- **WGS 84 Bounding Box** --- The bounding box of the layer, in WGS 84 coordinates. Some WMS servers do not set this correctly (e.g., UTM coordinates are used instead). If this is the case, then the initial view of this layer may be rendered with a very 'zoomed-out' appearance by QGIS. The WMS webmaster should be informed of this error, which they may know as the WMS XML elements `LatLonBoundingBox`, `EX_GeographicBoundingBox` or the `CRS:84 BoundingBox`.
- **Available in CRS** --- The projections that this layer can be rendered in by the WMS server. These are listed in the WMS-native format.
- **Available in style** --- The image styles that this layer can be rendered in by the WMS server.

19.1.7 Show WMS legend graphic in table of contents and layout

The QGIS WMS data provider is able to display a legend graphic in the table of contents' layer list and in the print layout. The WMS legend will be shown only if the WMS server has GetLegendGraphic capability and the layer has getCapability url specified, so you additionally have to select a styling for the layer.

If a legendGraphic is available, it is shown below the layer. It is little and you have to click on it to open it in real dimension (due to QgsLegendInterface architectural limitation). Clicking on the layer's legend will open a frame with the legend at full resolution.


In the print layout, the legend will be integrated at its original (downloaded) dimension. Resolution of the legend graphic can be set in the item properties under *Legend* ► *WMS LegendGraphic* to match your printing requirements.

The legend will display contextual information based on your current scale. The WMS legend will be shown only if the WMS server has GetLegendGraphic capability and the layer has getCapability url specified, so you have to select a styling.

19.1.8 WMS Client Limitations

Not all possible WMS client functionality had been included in this version of QGIS. Some of the more noteworthy exceptions follow.

Editing WMS Layer Settings

Once you've completed the  Add WMS layer procedure, there is no way to change the settings. A work-around is to delete the layer completely and start again.

WMS Servers Requiring Authentication

Currently, publicly accessible and secured WMS services are supported. The secured WMS servers can be accessed by public authentication. You can add the (optional) credentials when you add a WMS server. See section *Selecting WMS/WMTS Servers* for details.


小技巧: Accessing secured OGC-layers

If you need to access secured layers with secured methods other than basic authentication, you can use InteProxy as a transparent proxy, which does support several authentication methods. More information can be found in the InteProxy manual at <https://inteproxy.wald.intevation.org>.

小技巧: QGIS WMS Mapserver

Since Version 1.7.0, QGIS has its own implementation of a WMS 1.3.0 Mapserver. Read more about this in QGIS-Server-manual.

19.2 WCS Client

 A Web Coverage Service (WCS) provides access to raster data in forms that are useful for client-side rendering, as input into scientific models, and for other clients. The WCS may be compared to the WFS and the WMS. As WMS and WFS service instances, a WCS allows clients to choose portions of a server's information holdings based on spatial constraints and other query criteria.

QGIS has a native WCS provider and supports both version 1.0 and 1.1 (which are significantly different), but currently it prefers 1.0, because 1.1 has many issues (i.e., each server implements it in a different way with various particularities).

The native WCS provider handles all network requests and uses all standard QGIS network settings (especially proxy). It is also possible to select cache mode ('always cache', 'prefer cache', 'prefer network', 'always network'), and the provider also supports selection of time position, if temporal domain is offered by the server.

警告: Entering **username** and **password** in the *Authentication* tab will keep unprotected credentials in the connection configuration. Those **credentials will be visible** if, for instance, you shared the project file with someone. Therefore, it's advisable to save your credentials in a *Authentication configuration* instead (*configurations* tab). See [认证系统](#) for more details.

19.3 WFS and WFS-T Client



In QGIS, a WFS layer behaves pretty much like any other vector layer. You can identify and select features, and view the attribute table. QGIS supports WFS 1.0.0, 1.1.0, 2.0 and WFS3 (OGC API - Features), including editing (through WFS-T).

In general, adding a WFS layer is very similar to the procedure used with WMS. There are no default servers defined, so you have to add your own. You can find WFS servers by using the *MetaSearch plugin* or your favourite web search engine. There are a number of lists with public URLs, some of them maintained and some not.

Loading a WFS Layer

As an example, we use the Gateway Geomatics WFS server and display a layer. https://demo.gatewaygeomatics.com/cgi-bin/wfs_gateway?REQUEST=GetCapabilities&VERSION=1.0.0&SERVICE=WFS

To be able to load a WFS Layer, first create a connection to the WFS server:

1. Open the *Data Source Manager* dialog by pressing the  Open Data Source Manager button
2. Enable the  *WFS/OGC API-Features* tab
3. Click on *New...* to open the *Create a New WFS Connection* dialog
4. Enter *Gateway Geomatics* as name
5. Enter the URL (see above)

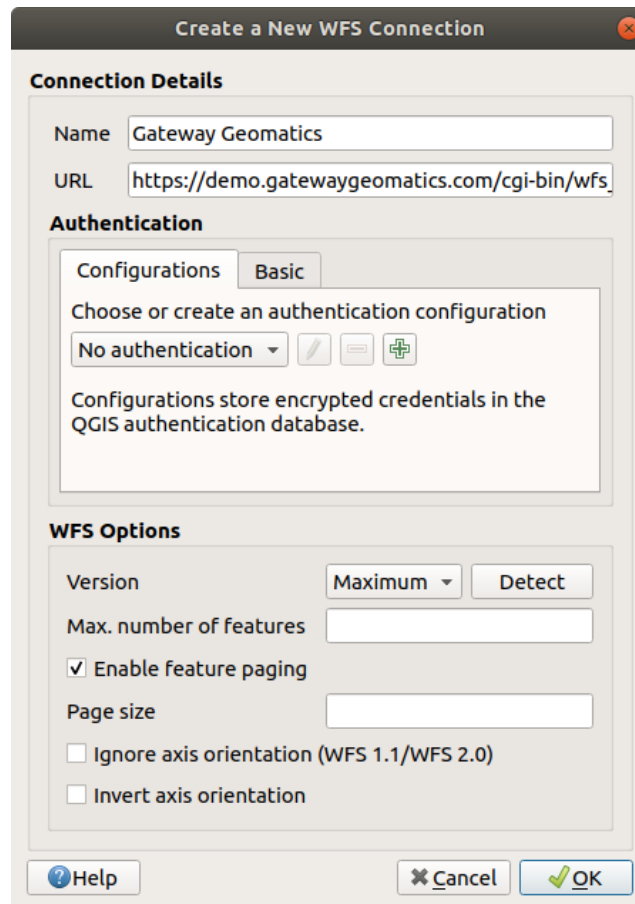


图 19.4: Creating a connection to a WFS server

注解: In case of an OGC API - Features (WFS3), the URL to provide should be the landing page, ie the main page from which it is possible to navigate to all the available service endpoints.

6. In the WFS settings dialog, you can:


- Indicate the WFS version of the server. If unknown, press the *Detect* button to automatically retrieve it.
- Define the *maximum number of features* retrieved in a single GetFetFeature request. If empty, no limit is set.
- *Invert axis orientation*.
- And depending on the WFS version:
 - Force to *Ignore axis orientation (WFS 1.1/WFS 2.0)*
 - *Enable feature paging* and specify the maximum number of features to retrieve with *Page size*. If no limit is defined, then the server default is applied.

警告: Entering **username** and **password** in the *Authentication* tab will keep unprotected credentials in the connection configuration. Those **credentials will be visible** if, for instance, you shared the project file with someone. Therefore, it's advisable to save your credentials in an *Authentication configuration* instead (*Configurations* tab). See [认证系统](#) for more details.

7. Press *OK* to create the connection.

Note that any proxy settings you may have set in your preferences are also recognized.

Now we are ready to load WFS layers from the above connection.

1. Choose 'Gateway Geomatics' from the *Server Connections*  drop-down list.
2. Click *Connect*
3. Select the *Parks* layer in the list
4. You can also choose whether to:
 - *Use title for layer name*, showing the layer's title as defined on the server in the *Layers* panel instead of its *Name*
 - *Only request features overlapping the view extent*
 - *Change* the layer's CRS
 - or *Build query* to specify particular features to retrieve, by either using the corresponding button or double-clicking the target layer.
5. Click *Add* to add the layer to the map.

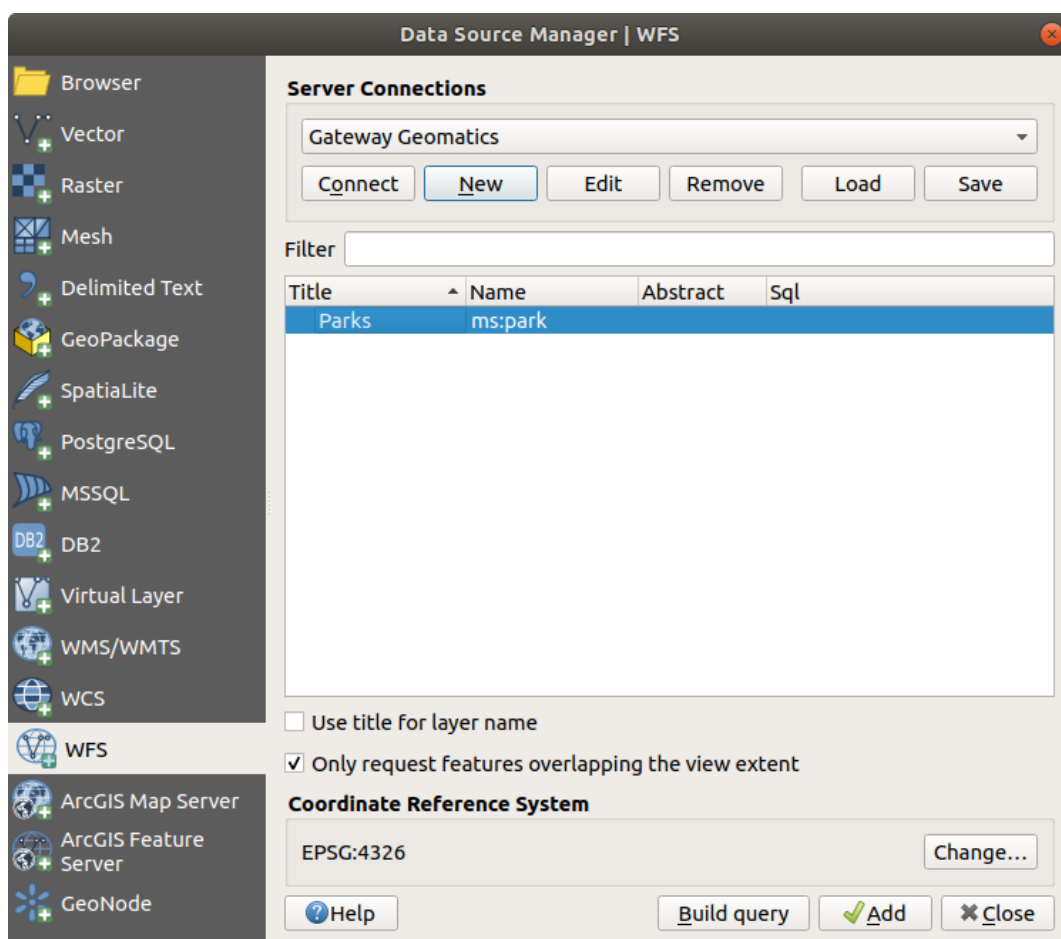


图 19.5: Adding a WFS layer

You'll notice the download progress is visualized in the lower left of the QGIS main window. Once the layer is loaded, you can identify and select a couple of features and view the attribute table.

注解: QGIS supports different versions of the WFS protocol, with background download and progressive rendering, on-disk caching of downloaded features and version autodetection.

20.1 GPS Plugin


20.1.1 What is GPS?

GPS, the Global Positioning System, is a satellite-based system that allows anyone with a GPS receiver to find their exact position anywhere in the world. GPS is used as an aid in navigation, for example in airplanes, in boats and by hikers. The GPS receiver uses the signals from the satellites to calculate its latitude, longitude and (sometimes) elevation. Most receivers also have the capability to store locations (known as **waypoints**), sequences of locations that make up a planned **route** and a tracklog or **track** of the receiver's movement over time. Waypoints, routes and tracks are the three basic feature types in GPS data. QGIS displays waypoints in point layers, while routes and tracks are displayed in linestring layers.

注解: QGIS supports also GNSS receivers. But we keep using the term GPS in this documentation.


20.1.2 Loading GPS data from a file

There are dozens of different file formats for storing GPS data. The format that QGIS uses is called GPX (GPS eXchange format), which is a standard interchange format that can contain any number of waypoints, routes and tracks in the same file.

To load a GPX file, you first need to load the plugin. *Plugins* ►  *Plugin Manager...* opens the Plugin Manager Dialog. Activate the *GPS Tools* checkbox. When this plugin is loaded, a button with a small handheld GPS device will show up in the toolbar and in *Layer* ► *Create Layer* ► :

-  GPS Tools
-  *Create new GPX Layer*

For working with GPS data, we provide an example GPX file available in the QGIS sample dataset: `qgis_sample_data/gps/national_monuments.gpx`. See section [下载示例数据](#) for more information about the sample data.

1. Select *Vector* ► *GPS Tools* or click the  icon in the toolbar and open the *Load GPX file* tab (see [图 20.1](#)).
2. Browse to the folder `qgis_sample_data/gps/`, select the GPX file `national_monuments.gpx` and click *Open*.

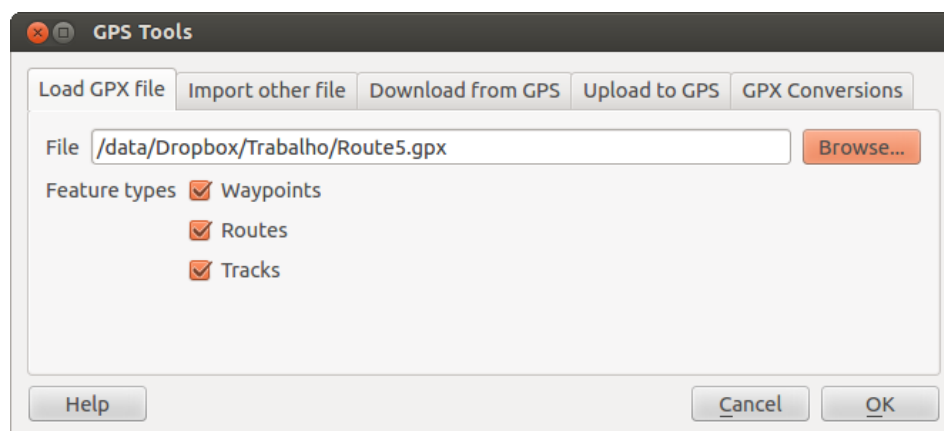


图 20.1: The *GPS Tools* dialog window

Use the *Browse...* button to select the GPX file, then use the checkboxes to select the feature types you want to load from that GPX file. Each feature type will be loaded in a separate layer when you click *OK*. The file `national_monuments.gpx` only includes waypoints.

注解: GPS units allow you to store data in different coordinate systems. When downloading a GPX file (from your GPS unit or a web site) and then loading it in QGIS, be sure that the data stored in the GPX file uses WGS 84 (latitude/longitude). QGIS expects this, and it is the official GPX specification. See <https://www.topografix.com/GPX/1/1/>.

20.1.3 GPSTabel

Since QGIS uses GPX files, you need a way to convert other GPS file formats to GPX. This can be done for many formats using the free program GPSTabel, which is available at <https://www.gpsbabel.org>. This program can also transfer GPS data between your computer and a GPS device. QGIS uses GPSTabel to do these things, so it is recommended that you install it. However, if you just want to load GPS data from GPX files you will not need it. Version 1.2.3 of GPSTabel is known to work with QGIS, but you should be able to use later versions without any problems.

20.1.4 Importing GPS data

To import GPS data from a file that is not a GPX file, you use the tool *Import other file* in the GPS Tools dialog. Here, you select the file that you want to import (and the file type), which feature type you want to import from it, where you want to store the converted GPX file and what the name of the new layer should be. Note that not all GPS data formats will support all three feature types, so for many formats you will only be able to choose between one or two types.

20.1.5 Downloading GPS data from a device

QGIS can use GPSTools to download data from a GPS device directly as new vector layers. For this we use the *Download from GPS* tab of the GPS Tools dialog (see [图 20.2](#)). Here, we select the type of GPS device, the port that it is connected to (or USB if your GPS supports this), the feature type that you want to download, the GPX file where the data should be stored, and the name of the new layer.

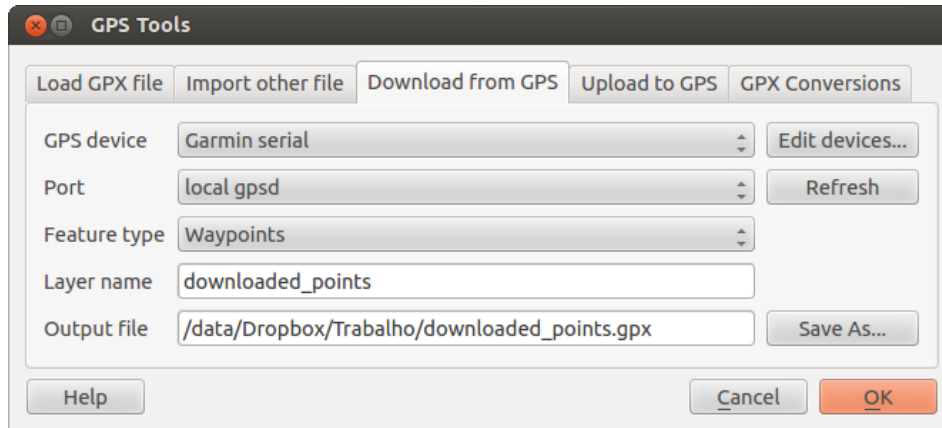


图 20.2: The download tool

The device type you select in the GPS device menu determines how GPSTools tries to communicate with your GPS device. If none of the available types work with your GPS device, you can create a new type (see section *Defining new device types*).

The port may be a file name or some other name that your operating system uses as a reference to the physical port in your computer that the GPS device is connected to. It may also be simply USB, for USB-enabled GPS units.

- On Linux, this is something like `/dev/ttyS0` or `/dev/ttyS1`.
- On Windows, it is COM1 or COM2.

When you click *OK*, the data will be downloaded from the device and appear as a layer in QGIS.

20.1.6 Uploading GPS data to a device

You can also upload data directly from a vector layer in QGIS to a GPS device using the *Upload to GPS* tab of the GPS Tools dialog. To do this, you simply select the layer that you want to upload (which must be a GPX layer), your GPS device type, and the port (or USB) that it is connected to. Just as with the download tool, you can specify new device types if your device isn't in the list.

This tool is very useful in combination with the vector-editing capabilities of QGIS. It allows you to load a map, create waypoints and routes, and then upload them and use them on your GPS device.

20.1.7 Defining new device types

There are lots of different types of GPS devices. The QGIS developers can't test all of them, so if you have one that does not work with any of the device types listed in the *Download from GPS* and *Upload to GPS* tools, you can define your own device type for it. You do this by using the GPS device editor, which you start by clicking the *Edit Devices* button in the download or the upload tab.

To define a new device, you simply click the *New Device* button, enter a name, enter download and upload commands for your device, and click the *Update Device* button. The name will be listed in the device menus in the upload and download windows -- it can be any string. The download command is the command that is used to download data from the device to a GPX file. This will probably be a GPSTools command, but you can use any other command

line program that can create a GPX file. QGIS will replace the keywords `%type`, `%in`, and `%out` when it runs the command.

`%type` will be replaced by `-w` if you are downloading waypoints, `-r` if you are downloading routes and `-t` if you are downloading tracks. These are command-line options that tell GPSBabel which feature type to download.

`%in` will be replaced by the port name that you choose in the download window and `%out` will be replaced by the name you choose for the GPX file that the downloaded data should be stored in. So, if you create a device type with the download command `gpsbabel %type -i garmin -o gpx %in %out` (this is actually the download command for the predefined device type 'Garmin serial') and then use it to download waypoints from port `/dev/ttyS0` to the file `output.gpx`, QGIS will replace the keywords and run the command `gpsbabel -w -i garmin -o gpx /dev/ttyS0 output.gpx`.

The upload command is the command that is used to upload data to the device. The same keywords are used, but `%in` is now replaced by the name of the GPX file for the layer that is being uploaded, and `%out` is replaced by the port name.

You can learn more about GPSBabel and its available command line options at <https://www.gpsbabel.org>.

Once you have created a new device type, it will appear in the device lists for the download and upload tools.

20.1.8 Download of points/tracks from GPS units

As described in previous sections QGIS uses GPSBabel to download points/tracks directly in the project. QGIS comes out of the box with a pre-defined profile to download from Garmin devices. Unfortunately there is a [bug #6318](#) that does not allow create other profiles, so downloading directly in QGIS using the GPS Tools is at the moment limited to Garmin USB units.

Garmin GPSMAP 60cs

MS Windows

Install the Garmin USB drivers from https://www8.garmin.com/support/download_details.jsp?id=591

Connect the unit. Open GPS Tools and use `type=garmin serial` and `port=usb`: Fill the fields *Layer name* and *Output file*. Sometimes it seems to have problems saving in a certain folder, using something like `c:\temp` usually works.

Ubuntu/Mint GNU/Linux

It is first needed an issue about the permissions of the device, as described at https://wiki.openstreetmap.org/wiki/USB_Garmin_on_GNU/Linux. You can try to create a file `/etc/udev/rules.d/51-garmin.rules` containing this rule

```
ATTRS{idVendor}=="091e", ATTRS{idProduct}=="0003", MODE="666"
```

After that is necessary to be sure that the `garmin_gps` kernel module is not loaded

```
rmmod garmin_gps
```

and then you can use the GPS Tools. Unfortunately there seems to be a [bug #7182](#) and usually QGIS freezes several times before the operation work fine.

BTGP-38KM datalogger (only Bluetooth)

MS Windows

The already referred bug does not allow to download the data from within QGIS, so it is needed to use GPSBabel from the command line or using its interface. The working command is

```
gpsbabel -t -i skytraq,baud=9600,initbaud=9600 -f COM9 -o gpx -F C:/GPX/aaa.gpx
```

Ubuntu/Mint GNU/Linux

Use same command (or settings if you use GPSBabel GUI) as in Windows. On Linux it maybe somehow common to get a message like

```
skytraq: Too many read errors on serial port
```

it is just a matter to turn off and on the datalogger and try again.

BlueMax GPS-4044 datalogger (both BT and USB)

MS Windows

注解: It needs to install its drivers before using it on Windows 7. See in the manufacturer site for the proper download.

Downloading with GPSBabel, both with USB and BT returns always an error like

```
gpsbabel -t -i mtk -f COM12 -o gpx -F C:/temp/test.gpx
mtk_logger: Can't create temporary file data.bin
Error running gpsbabel: Process exited unsuccessfully with code 1
```

Ubuntu/Mint GNU/Linux

With USB

After having connected the cable use the `dmesg` command to understand what port is being used, for example `/dev/ttyACM3`. Then as usual use GPSBabel from the CLI or GUI


```
gpsbabel -t -i mtk -f /dev/ttyACM3 -o gpx -F /home/user/bluemax.gpx
```

With Bluetooth




Use Blueteman Device Manager to pair the device and make it available through a system port, then run GPSBabel

```
gpsbabel -t -i mtk -f /dev/rfcomm0 -o gpx -F /home/user/bluemax_bt.gpx
```

20.2 Live GPS tracking

To activate live GPS tracking in QGIS, you need to select *View* ► *Panels*  *GPS Information Panel* or press `Ctrl+0`. You will get a new docked window on the left side of the canvas.


There are four possible screens in this GPS tracking window:

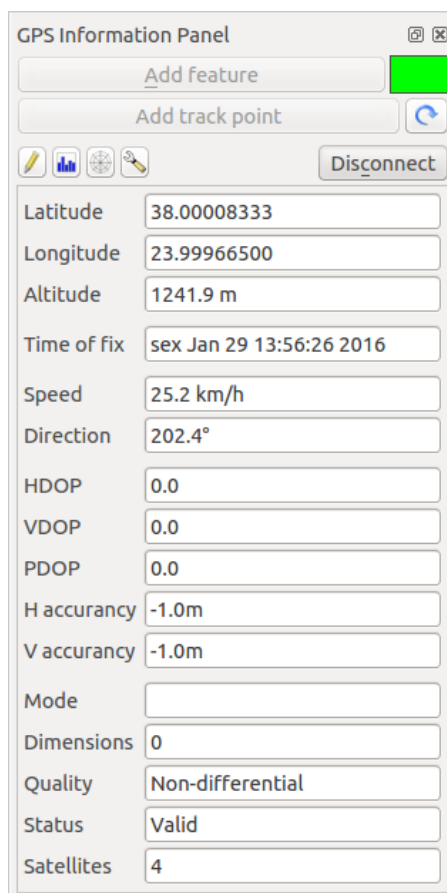
-  GPS position coordinates and an interface for manually entering vertices and features
-  GPS signal strength of satellite connections
-  GPS options screen (see [图 20.5](#))

With a plugged-in GPS receiver (has to be supported by your operating system), a simple click on *Connect* connects the GPS to QGIS. A second click (now on *Disconnect*) disconnects the GPS receiver from your computer. For GNU/Linux, *gpsd* support is integrated to support connection to most GPS receivers. Therefore, you first have to configure *gpsd* properly to connect QGIS to it.

警告: If you want to record your position to the canvas, you have to create a new vector layer first and switch it to editable status to be able to record your track.

20.2.1 Position and additional attributes

 If the GPS is receiving signals from satellites, you will see your position in latitude, longitude and altitude together with additional attributes.









GPS Information Panel	
Add feature	
Add track point	
   	Disconnect
Latitude	38.00008333
Longitude	23.99966500
Altitude	1241.9 m
Time of fix	sex Jan 29 13:56:26 2016
Speed	25.2 km/h
Direction	202.4°
HDOP	0.0
VDOP	0.0
PDOP	0.0
H accuracy	-1.0m
V accuracy	-1.0m
Mode	
Dimensions	0
Quality	Non-differential
Status	Valid
Satellites	4

图 20.3: GPS tracking position and additional attributes

20.2.2 GPS signal strength



Here, you can see the signal strength of the satellites you are receiving signals from.

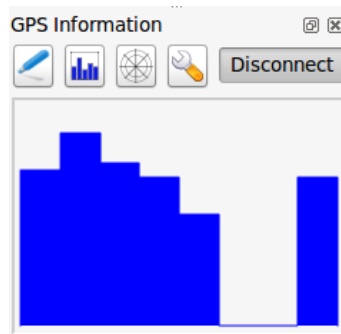


图 20.4: GPS tracking signal strength

20.2.3 GPS options

In case of connection problems, you can switch between:

- *Autodetect*
- *Internal*
- *Serial device*
- *gpsd* (selecting the Host, Port and Device your GPS is connected to)

A click on *Connect* again initiates the connection to the GPS receiver.

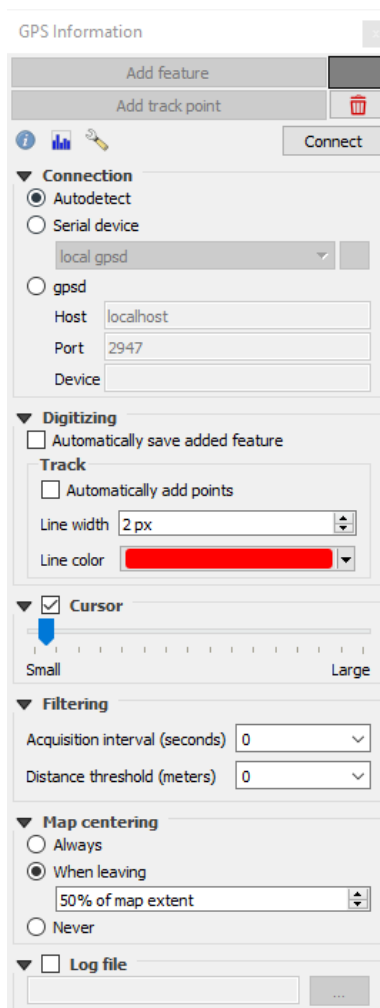


图 20.5: GPS tracking options window


You can activate *Automatically save added features* when you are in editing mode. Or you can activate *Automatically add points* to the map canvas with a certain width and color.

Activating *Cursor*, you can use a slider  to shrink and grow the position cursor on the canvas.

You can also set an *Acquisition interval (seconds)* and a *Distance threshold (meters)* parameters to keep the cursor still active when the receiver is in static conditions.

Activating *Map centering* allows you to decide in which way the canvas will be updated. This includes 'always', 'when leaving', if your recorded coordinates start to move out of the canvas, or 'never', to keep map extent.

Finally, you can activate *Log file* and define a path and a file where log messages about the GPS tracking are logged.

If you want to set a feature manually, you have to go back to  *Position* and click on *Add Point* or *Add Track Point*.

20.2.4 Connect to a Bluetooth GPS for live tracking


With QGIS you can connect a Bluetooth GPS for field data collection. To perform this task you need a GPS Bluetooth device and a Bluetooth receiver on your computer.

At first you must let your GPS device be recognized and paired to the computer. Turn on the GPS, go to the Bluetooth icon on your notification area and search for a New Device.

On the right side of the Device selection mask make sure that all devices are selected so your GPS unit will probably appear among those available. In the next step a serial connection service should be available, select it and click on *Configure* button.

Remember the number of the COM port assigned to the GPS connection as resulting by the Bluetooth properties.

After the GPS has been recognized, make the pairing for the connection. Usually the authorization code is 0000.


Now open *GPS information* panel and switch to  GPS options screen. Select the COM port assigned to the GPS connection and click the *Connect*. After a while a cursor indicating your position should appear.

If QGIS can't receive GPS data, then you should restart your GPS device, wait 5-10 seconds then try to connect again. Usually this solution work. If you receive again a connection error make sure you don't have another Bluetooth receiver near you, paired with the same GPS unit.

20.2.5 Using GPSPMAP 60cs

MS Windows

Easiest way to make it work is to use a middleware (freeware, not open) called [GPSGate](#).

Launch the program, make it scan for GPS devices (works for both USB and BT ones) and then in QGIS just click *Connect* in the Live tracking panel using the  *Autodetect* mode.

Ubuntu/Mint GNU/Linux

As for Windows the easiest way is to use a server in the middle, in this case `gpsd`, so

```
sudo apt install gpsd
```

Then load the `garmin_gps` kernel module

```
sudo modprobe garmin_gps
```

And then connect the unit. Then check with `dmesg` the actual device being used by the unit, for example `/dev/ttyUSB0`. Now you can launch `gpsd`

```
gpsd /dev/ttyUSB0
```


And finally connect with the QGIS live tracking tool.

20.2.6 Using BTGP-38KM datalogger (only Bluetooth)

Using GPSD (under Linux) or GPSTool (under Windows) is effortless.

20.2.7 Using BlueMax GPS-4044 datalogger (both BT and USB)

MS Windows

The live tracking works for both USB and BT modes, by using GPSTool or even without it, just use the  *Autodetect* mode, or point the tool the right port.

Ubuntu/Mint GNU/Linux

For USB

The live tracking works both with GPSD

```
gpsd /dev/ttyACM3
```

or without it, by connecting the QGIS live tracking tool directly to the device (for example `/dev/ttyACM3`).

For Bluetooth

The live tracking works both with GPSD

```
gpsd /dev/rfcomm0
```

or without it, by connecting the QGIS live tracking tool directly to the device (for example `/dev/rfcomm0`).

21.1 Authentication System Overview

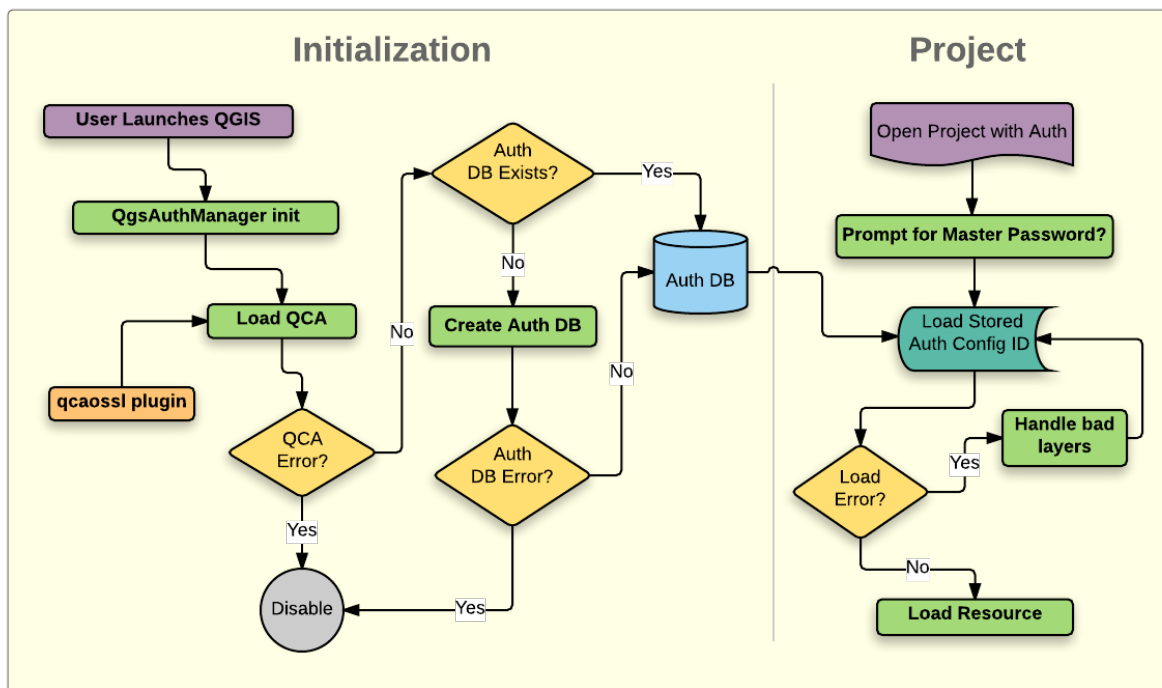


图 21.1: Anatomy of authentication system

21.1.1 Authentication database

The new authentication system stores authentication configurations in an SQLite database file located, by default, at `<profile directory>/qgis-auth.db`.

This authentication database can be moved between QGIS installations without affecting other current QGIS user preferences, as it is completely separate from normal QGIS settings. A configuration ID (a random 7-character

alphanumeric string) is generated when initially storing a configuration to the database. This represents the configuration, thereby allowing the ID to be stored in plain text application components, (such as project, plugin, or settings files) without disclosure of its associated credentials.

注解: The parent directory of the *qgis-auth.db* can be set using the following environment variable, `QGIS_AUTH_DB_DIR_PATH`, or set on the command line during launch with the `--authdbdirectory` option.

21.1.2 Master password

To store or access sensitive information within the database, a user must define a *master password*. A new master password is requested and verified when initially storing any encrypted data to the database. When sensitive information is accessed, the user is prompted for the master password. The password is then cached for the remainder of the session (until application is quit), unless the user manually chooses an action to clear its cached value. Some instances of using the authentication system do not require input of the master password, such as when selecting an existing authentication configuration, or applying a configuration to a server configuration (such as when adding a WMS layer).

You can choose to save the password in the Wallet/Keyring of your computer.

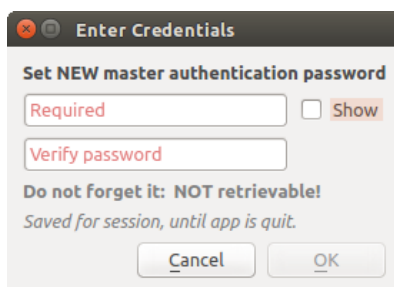


图 21.2: Input new master password

注解: A path to a file containing the master password can be set using the following environment variable, `QGIS_AUTH_PASSWORD_FILE`.

Managing the master password

Once set, the master password can be reset; the current master password will be needed prior to resetting. During this process, there is an option to generate a complete backup of the current database.

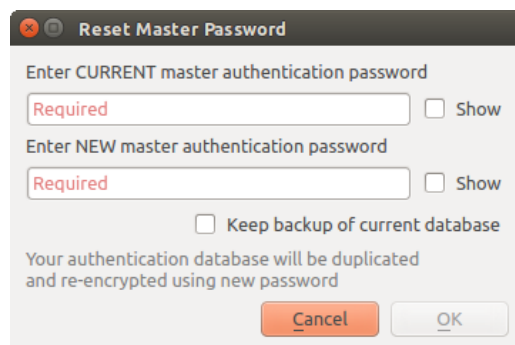


图 21.3: Resetting master password

If the user forgets the master password, there is no way to retrieve or override it. There is also no means of retrieving encrypted information without knowing the master password.

If a user inputs their existing password incorrectly three times, the dialog will offer to erase the database.

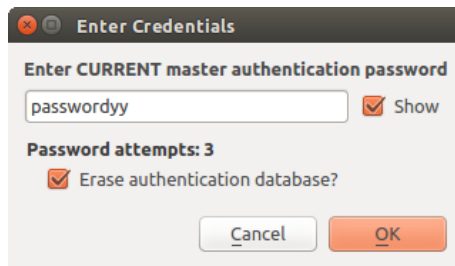


图 21.4: Password prompt after three invalid attempts

21.1.3 Authentication Configurations

You can manage authentication configurations from *Configurations* in the *Authentication* tab of the QGIS Options dialog (*Settings* ► *Options*).

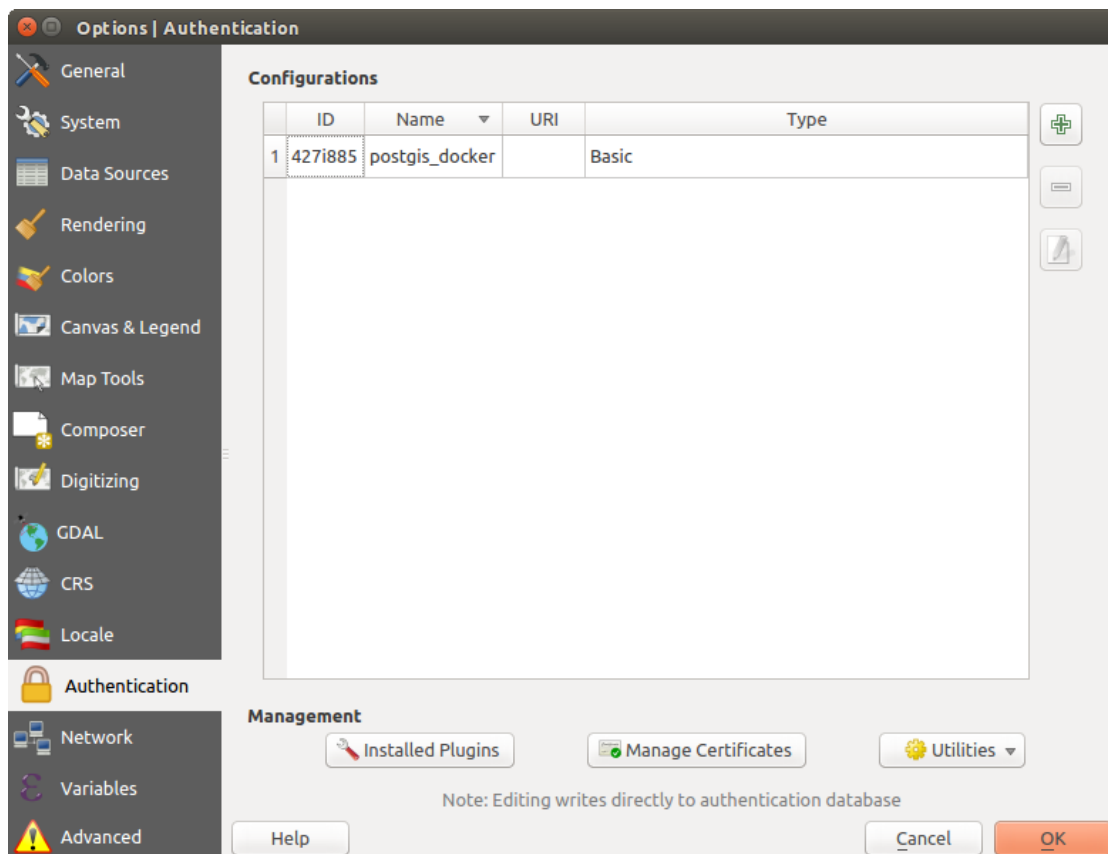





图 21.5: Configurations editor

Use the  button to add a new configuration, the  button to remove configurations, and the  button to modify existing ones.

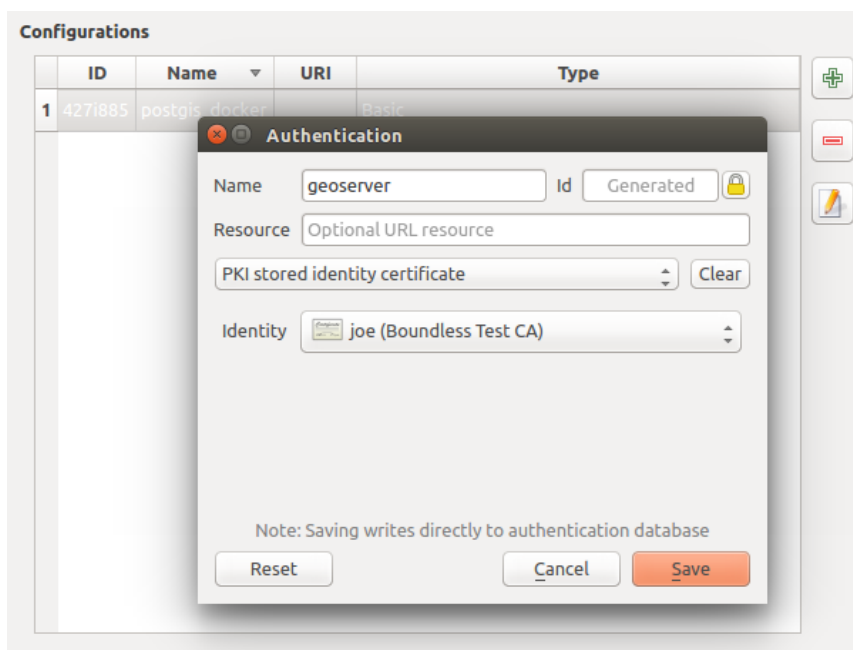


图 21.6: Adding config from within Configuration editor

The same type of operations for authentication configuration management (Add, Edit and Remove) can be done when configuring a given service connection, such as configuring an OWS service connection. For that, there are action buttons within the configuration selector for fully managing configurations found within the authentication database. In this case, there is no need to go to the *configurations* in *Authentication* tab of QGIS options unless you need to do more comprehensive configuration management.

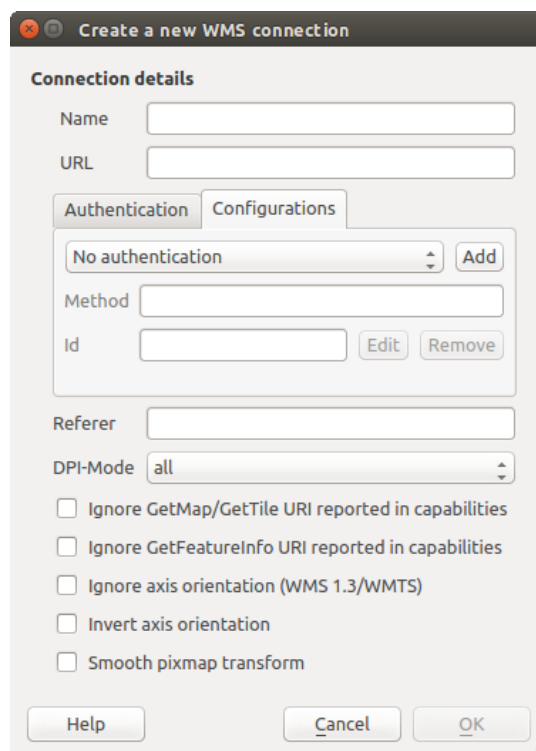



图 21.7: WMS connection dialog showing *Add*, *Edit*, and *Remove* authentication configuration buttons

When creating or editing an authentication configuration, the info required is a name, an authentication method

and any other info that the authentication method requires (see more about the available authentication types in *Authentication Methods*).

21.1.4 Authentication Methods

Available authentications are provided by C++ plugins much in the same way data provider plugins are supported by QGIS. The method of authentication that can be selected is relative to the access needed for the resource/provider, e.g. HTTP(S) or database, and whether there is support in both QGIS code and a plugin. As such, some authentication method plugins may not be applicable everywhere an authentication configuration selector is shown. A list of available authentication method plugins and their compatible resource/providers can be accessed going to *Settings* ► *Options* and, in the *Authentication* tab, click the  *Installed Plugins* button.

Method	Description	Works with
Basic	Basic authentication	postgres, db2, ows, wfs, wcs, wms, ogr, gdal, proxy
EsriToken	ESRI token based authentication	arcgismapservice, arcgisfeatureserver
Identity-Cert	PKI stored identity certificate	ows, wfs, wcs, wms, postgres
OAuth2	OAuth2 authentication	ows, wfs, wcs, wms
PKI-Paths	PKI paths authentication	ows, wfs, wcs, wms, postgres
PKI-PKCS#12	PKI PKCS#12 authentication	ows, wfs, wcs, wms, postgres


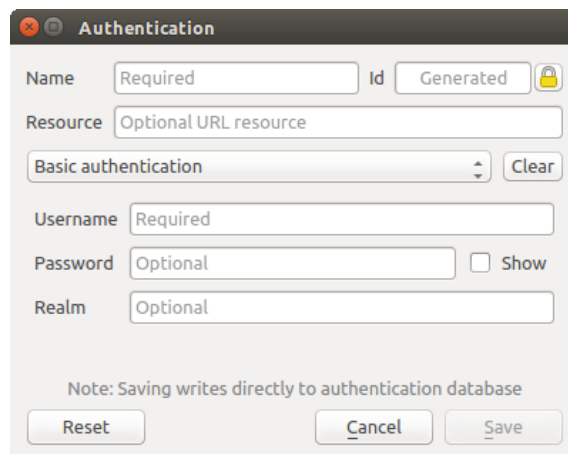
 Close

图 21.8: Available method plugins list

Plugins can be created for new authentication methods that do not require QGIS to be recompiled. Since the support for plugins is currently C++-only, QGIS will need to be restarted for the new dropped-in plugin to become available to the user. Ensure your plugin is compiled against the same target version of QGIS if you intend to add it to an existing target install.



The screenshot shows the 'Authentication' dialog box with the following fields and options:

- Name:** Required (text input)
- Id:** Generated (text input) with a lock icon.
- Resource:** Optional URL resource (text input)
- Authentication Method:** Basic authentication (dropdown menu) with a 'Clear' button.
- Username:** Required (text input)
- Password:** Optional (text input) with a 'Show' checkbox.
- Realm:** Optional (text input)

At the bottom, there is a note: "Note: Saving writes directly to authentication database" and three buttons: "Reset", "Cancel", and "Save".

图 21.9: Basic HTTP authentication configs

The screenshot shows a configuration dialog for ESRI Token authentication. It includes the following elements:

- Name:** A text input field containing the word "Required".
- Id:** A text input field containing the word "Generated", followed by a lock icon.
- Resource:** A text input field containing the text "Optional URL resource".
- Authentication Type:** A dropdown menu currently showing "ESRI token based authentication" with a "Clear" button to its right.
- Token:** A large text area with the label "Token" on the left. The text area is currently empty.
- Footer:** A note that reads "Note: Saving writes directly to authentication database" and three buttons: "Reset", "Cancel", and "Save".

图 21.10: ESRI Token authentication configs

Name Id

Resource

OAuth2 authentication

Configure

Grant Flow

Description

Request URL

Token URL

Refresh Token URL

Redirect URL

Client ID

Client Secret

Scope

API Key

Advanced

Token Session Persist between launches

Access Method

Request Timeout

Extra initial request parameters

Key	Value (unencoded)	
		<input type="button" value="+"/>
		<input type="button" value="-"/>

Note: Saving writes directly to authentication database

图 21.11: OAuth2 authentication configs

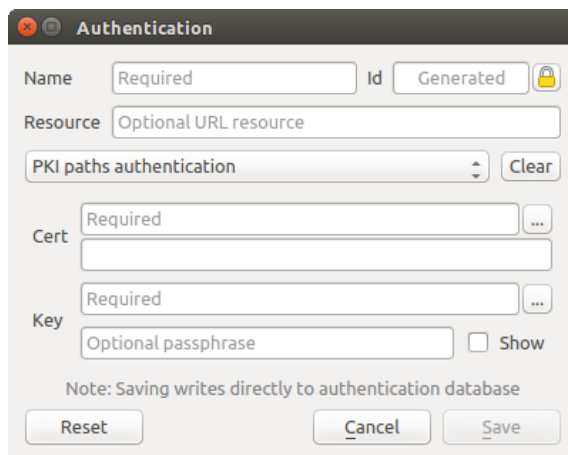


图 21.12: PKI paths authentication configs

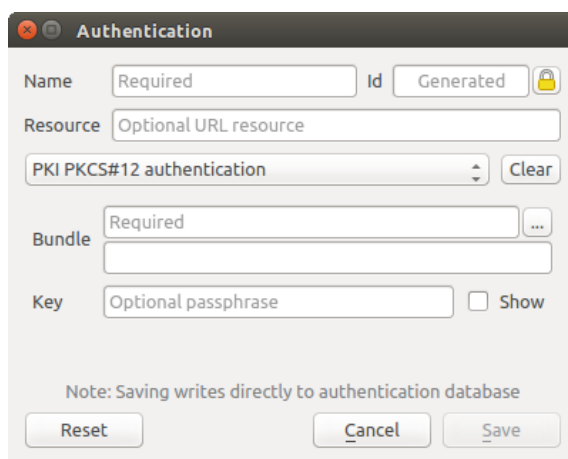


图 21.13: PKI PKCS#12 file paths authentication configs

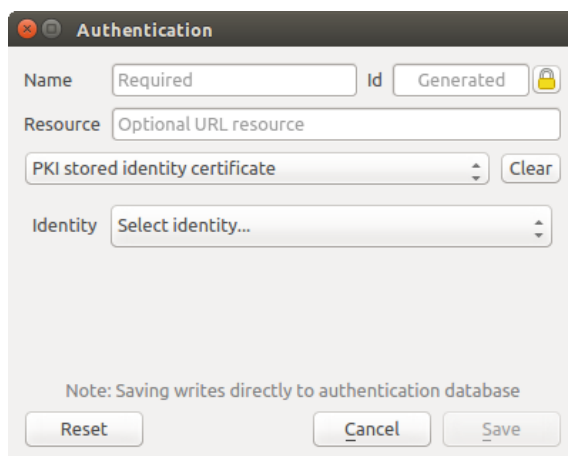


图 21.14: Stored Identity authentication configs

注解: The Resource URL is currently an *unimplemented* feature that will eventually allow a particular configuration to be auto-chosen when connecting to resources at a given URL.

21.1.5 Master Password and Auth Config Utilities

Under the Options menu (*Settings* ► *Options*) in the *Authentication* tab, there are several utility actions to manage the authentication database and configurations:

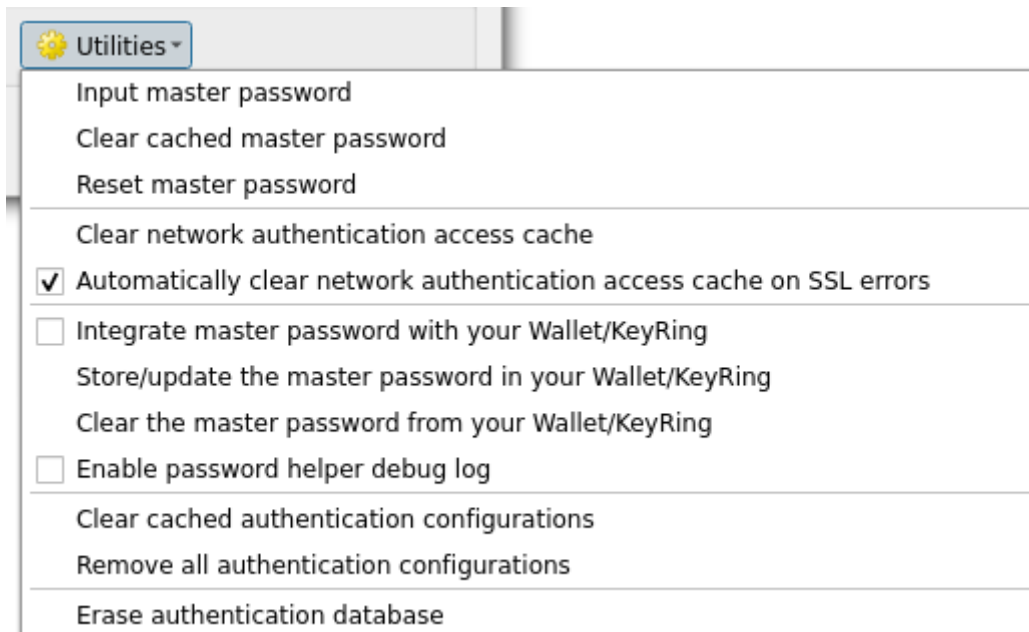


图 21.15: Utilities menu

- **Input master password:** opens the master password input dialog, independent of performing any authentication database command
- **Clear cached master password:** unsets the master password if it has been set
- **Reset master password:** opens a dialog to change the master password (the current password must be known) and optionally back up the current database
- **Clear network authentication access cache:** clears the authentication cache of all connections
- **Automatically clear network authentication access cache on SSL errors:** the connection cache stores all authentication data for connections, also when the connection fails. If you change authentication configurations or certification authorities, you should clear the authentication cache or restart QGIS. When this option is checked, the authentication cache will be automatically cleared every time an SSL error occurs and you choose to abort the connection
- **Integrate master password with your Wallet/Keyring:** adds the master password to your personal Wallet/Keyring
- **Store/update the master password in your Wallet/Keyring:** updates the changed master password in your Wallet/Keyring
- **Clear the master password from your Wallet/Keyring:** deletes the master password from your Wallet/Keyring
- **Enable password helper debug log:** enables a debug tool that will contain all the log information of the authentication methods
- **Clear cached authentication configurations:** clears the internal lookup cache for configurations, used to speed up network connections. This does not clear QGIS' s core network access manager' s cache, which requires a relaunch of QGIS.
- **Remove all authentication configurations:** clears the database of all configuration records, without removing other stored records.

- **Erase authentication database:** schedules a backup of the current database and complete rebuild of the database table structure. The actions are scheduled for a later time, to ensure that other operations, like project loading, do not interrupt the operation or cause errors due to a temporarily missing database.

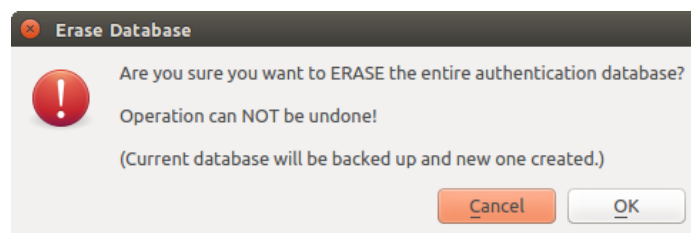


图 21.16: DB erase verification menu

21.1.6 Using authentication configurations

Typically, an authentication configuration is selected in a configuration dialog for a network services (such as WMS). However, the selector widget can be embedded anywhere authentication is needed or in non-core functionality, like in third-party PyQGIS or C++ plugins.

When using the selector, *No authentication* is displayed in the pop-up menu control when nothing is selected, when there are no configurations to choose from, or when a previously assigned configuration can no longer be found in the database. The *Type* and *Id* fields are read-only and provide a description of the authentication method and the configuration ID respectively.

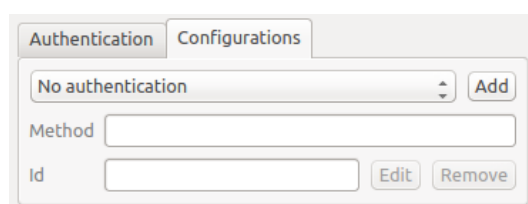


图 21.17: Authentication configuration selector with no authentication

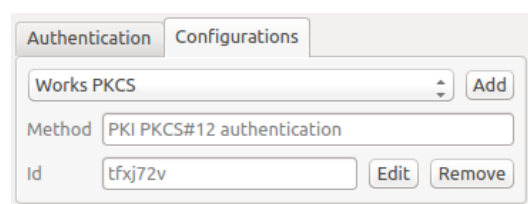


图 21.18: Authentication configuration selector with selected config

21.1.7 Python bindings

All classes and public functions have sip bindings, except `QgsAuthCrypto`, since management of the master password hashing and auth database encryption should be handled by the main app, and not via Python. See *Security Considerations* concerning Python access.

21.2 User Authentication Workflows

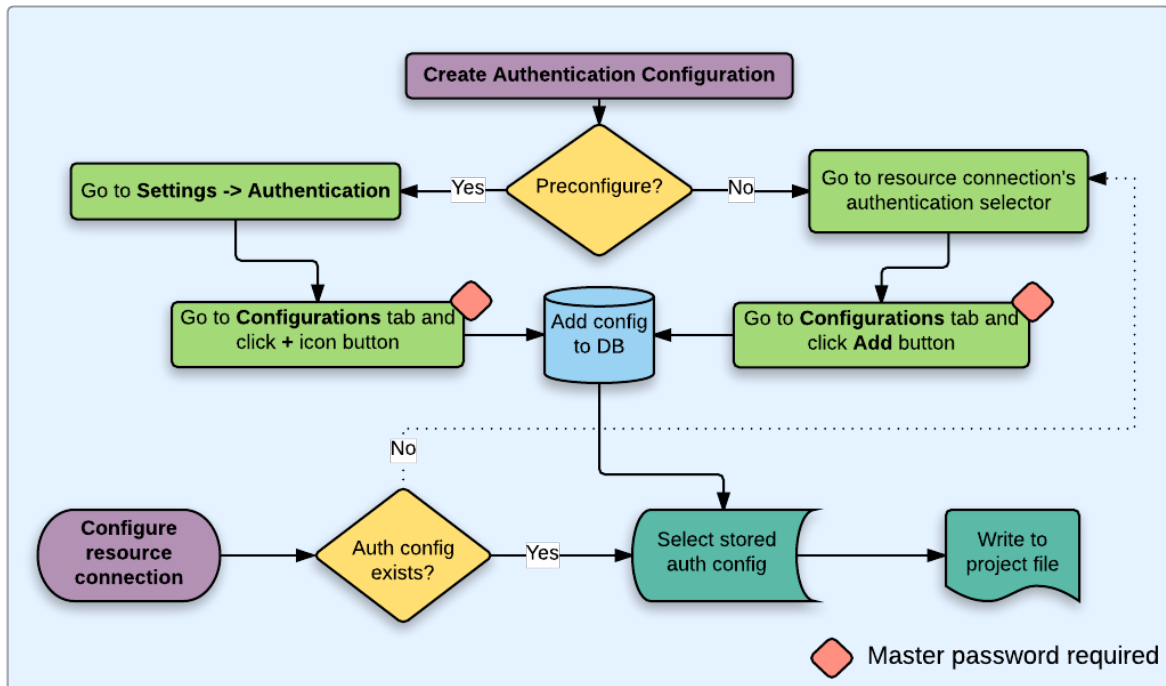


图 21.19: Generic user workflow

21.2.1 HTTP(S) authentication

One of the most common resource connections is via HTTP(S), e.g. web mapping servers, and authentication method plugins often work for these types of connections. Method plugins have access to the HTTP request object and can manipulate both the request as well as its headers. This allows for many forms of internet-based authentication. When connecting via HTTP(S) using the standard username/password authentication method will attempt HTTP BASIC authentication upon connection.

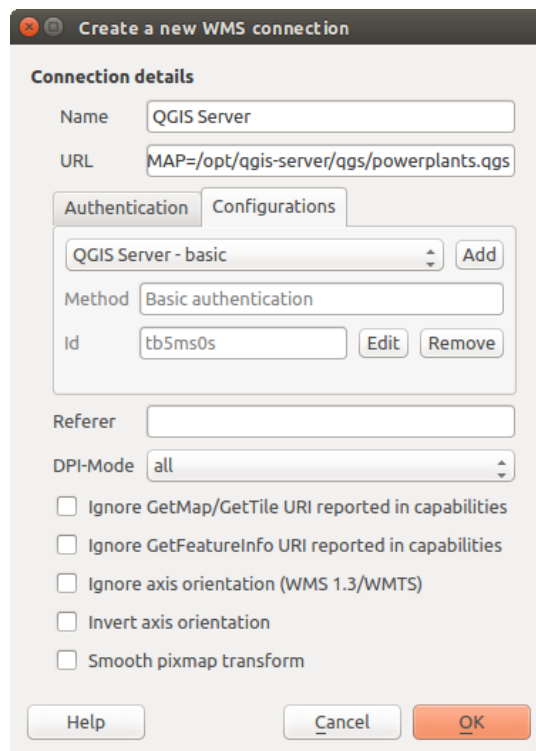


图 21.20: Configuring a WMS connection for HTTP BASIC

21.2.2 Database authentication

Connections to database resources are generally stored as `key=value` pairs, which will expose usernames and (optionally) passwords, if *not* using an authentication configuration. When configuring with the auth system, the `key=value` will be an abstracted representation of the credentials, e.g. `authfg=81t21b9`.

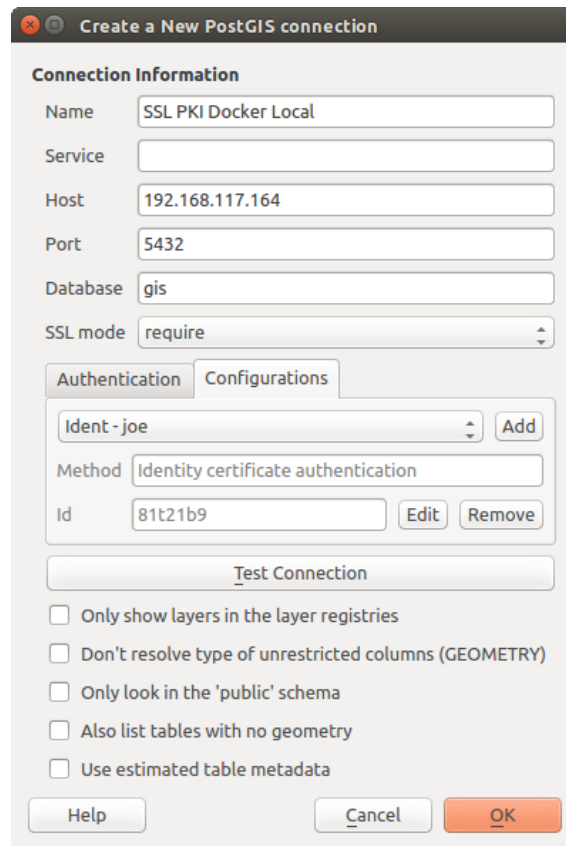


图 21.21: Configuring a Postgres SSL-with-PKI connection

21.2.3 PKI authentication

When configuring PKI components within the authentication system, you have the option of importing components into the database or referencing component files stored on your filesystem. The latter may be useful if such components change frequently, or where the components will be replaced by a system administrator. In either instance you will need to store any passphrase needed to access private keys within the database.

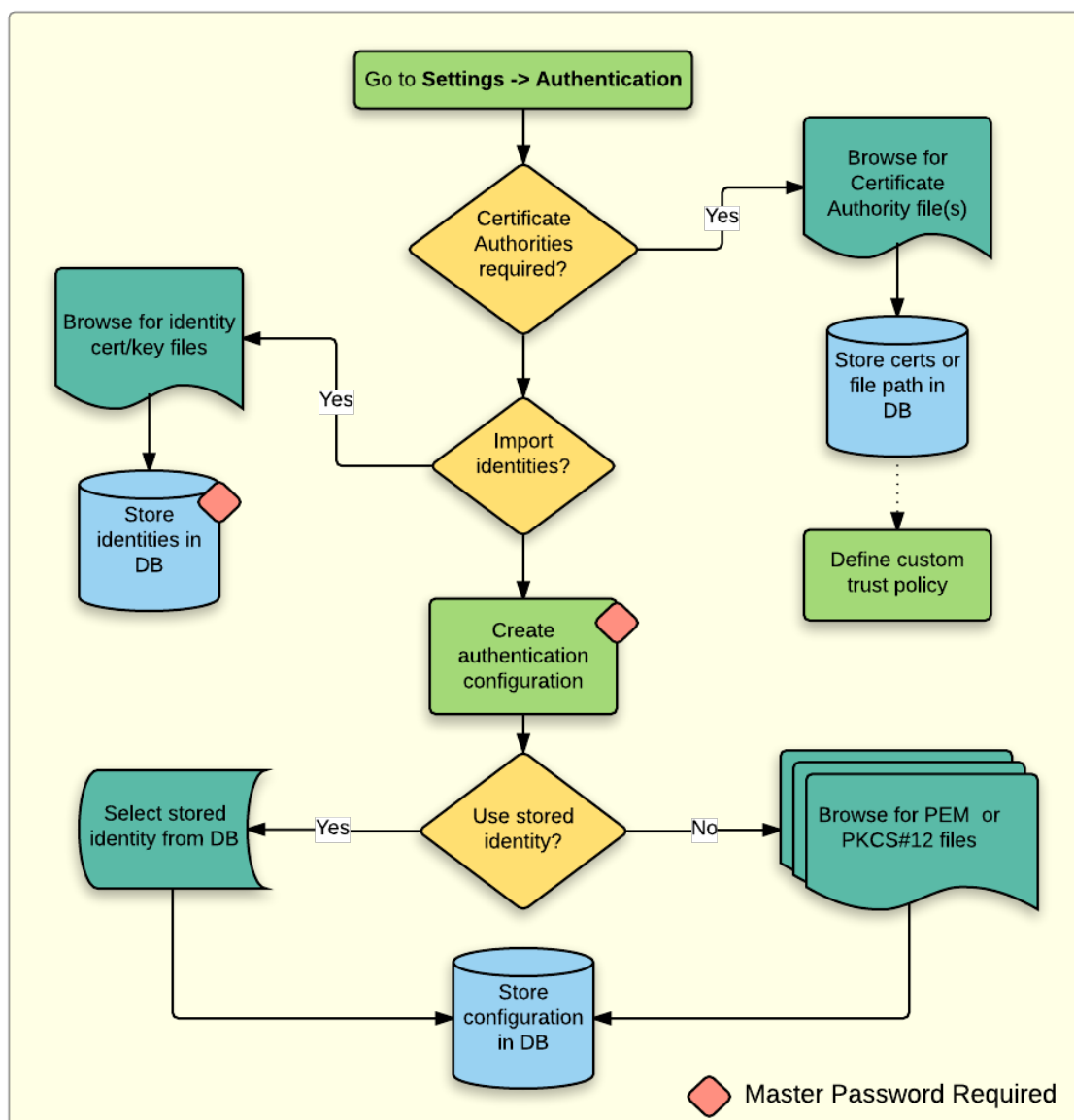


图 21.22: PKI configuration workflow

All PKI components can be managed in separate editors within the **Certificate Manager**, which can be accessed in the *Authentication* tab in *QGIS Options* dialog (*Settings* ► *Options*) by clicking the *Manage Certificates* button.

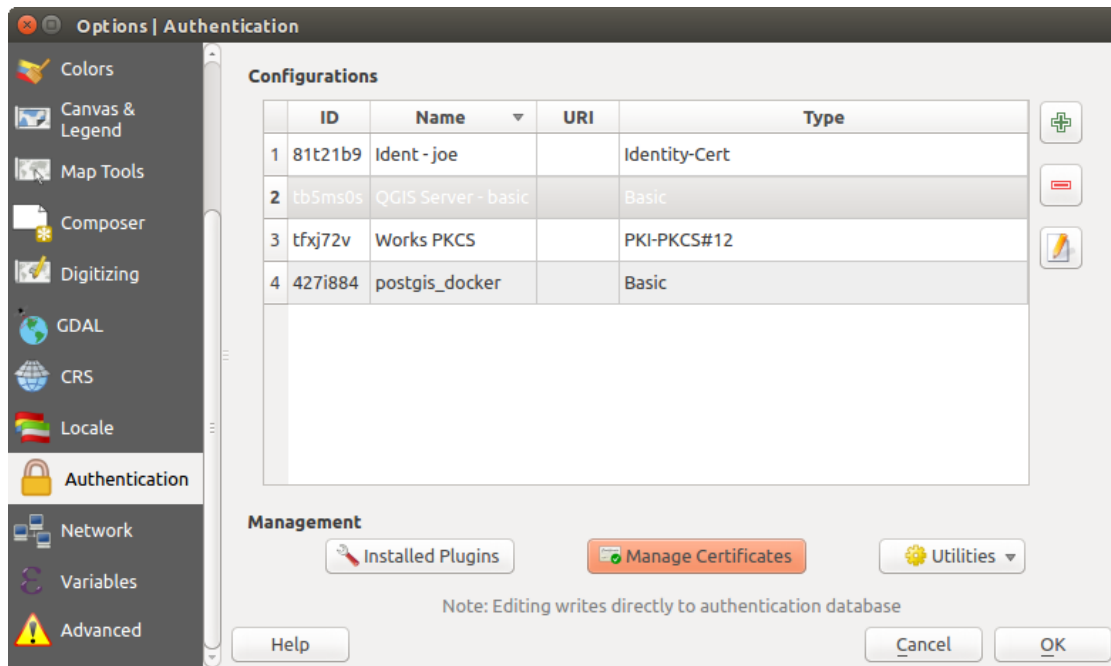


图 21.23: Opening the Certificate Manager

In the *Certificate Manager*, there are editors for **Identities**, **Servers** and **Authorities**. Each of these are contained in their own tabs, and are described below in the order they are encountered in the workflow chart above. The tab order is relative to frequently accessed editors once you are accustomed to the workflow.

注解: Because all authentication system edits write immediately to the authentication database, there is no need to click the *Options* dialog *OK* button for any changes to be saved. This is unlike other settings in the *Options* dialog.

Authorities

You can manage available Certificate Authorities (CAs) from the **Authorities** tab in the **Certificate manager** from the **Authentication** tab of the QGIS **Options** dialog.

As referenced in the workflow chart above, the first step is to import or reference a file of CAs. This step is optional, and may be unnecessary if your PKI trust chain originates from root CAs already installed in your operating system (OS), such as a certificate from a commercial certificate vendor. If your authenticating root CA is not in the OS's trusted root CAs, it will need to be imported or have its file system path referenced. (Contact your system administrator if unsure.)

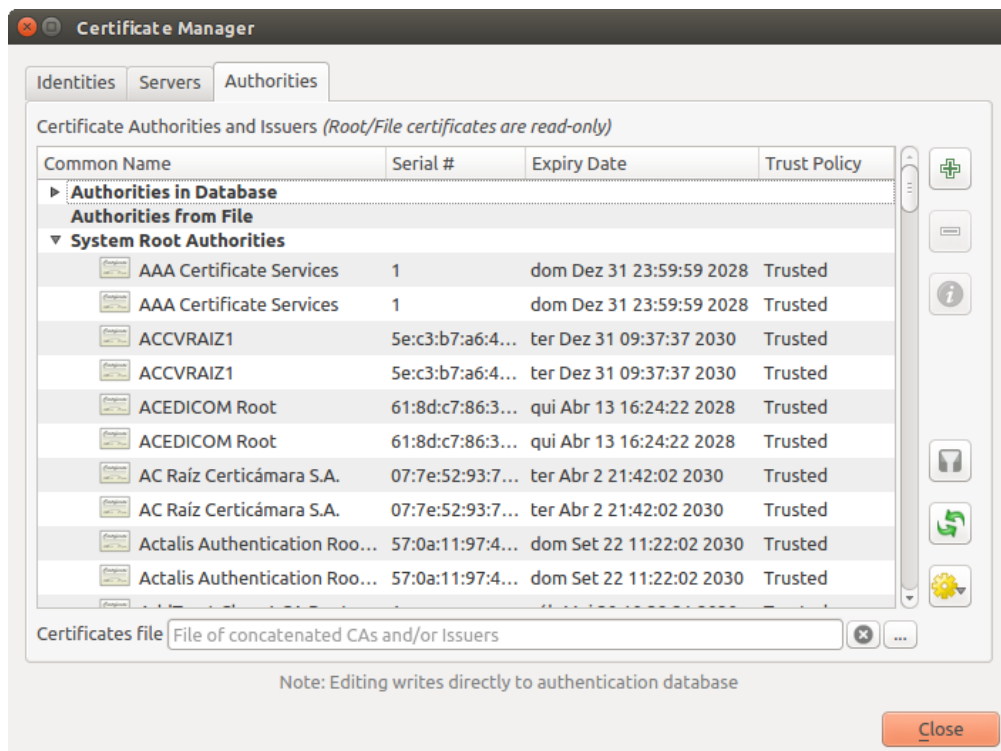



图 21.24: Authorities editor

By default, the root CAs from your OS are available; however, their trust settings are not inherited. You should review the certificate trust policy settings, especially if your OS root CAs have had their policies adjusted. Any certificate that is expired will be set to untrusted and will not be used in secure server connections, unless you specifically override its trust policy. To see the QGIS-discoverable trust chain for any certificate, select it and click the  Show information for certificate.

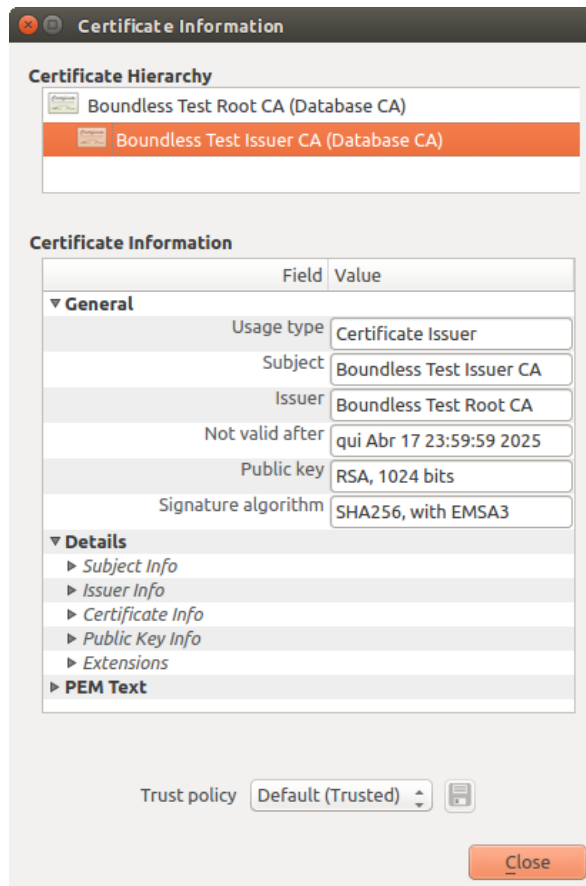


图 21.25: Certificate info dialog




You can edit the *Trust policy*  for any selected certificate within the chain. Any change in trust policy to a selected certificate will not be saved to the database unless the  Save certificate trust policy change to database button is clicked per selected certification. Closing the dialog will **not** apply the policy changes.



图 21.26: Saving the trust policy changes

You can review the filtered CAs, both intermediate and root certificates, that will be trusted for secure connections or change the default trust policy by clicking the  **Options** button.

警告: Changing the default trust policy may result in problems with secure connections.

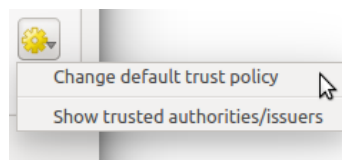


图 21.27: Authorities options menu

You can import CAs or save a file system path from a file that contains multiple CAs, or import individual CAs. The standard PEM format for files that contain multiple CA chain certifications has the root cert at the bottom of the file and all subsequently signed child certificates above, towards the beginning of the file.

The CA certificate import dialog will find all CA certificates within the file, regardless of order, and also offers the option to import certificates that are considered invalid (in case you want to override their trust policy). You can override the trust policy upon import, or do so later within the **Authorities** editor.

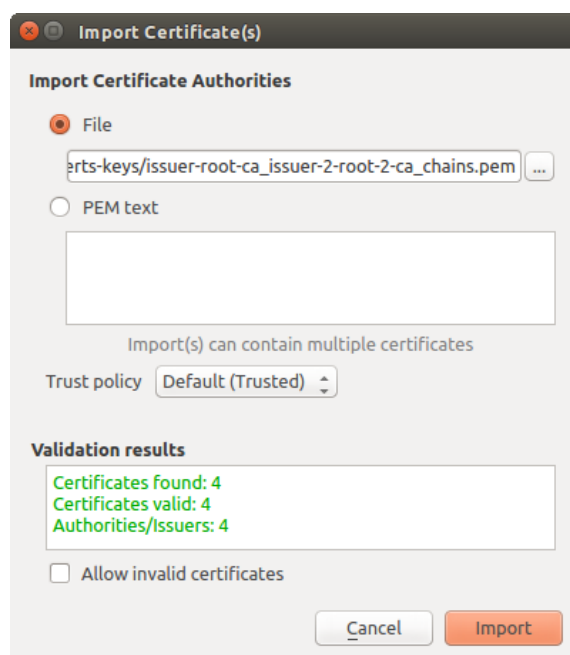


图 21.28: Import certificates dialog

注解: If you are pasting certificate information into the *PEM text* field, note that encrypted certificates are not supported.

Identities

You can manage available client identity bundles from the *Identities* tab in the *Certificate manager* from the **Authentication** tab of the **QGIS Options** dialog. An identity is what authenticates you against a PKI-enabled service and usually consists of a client certificate and private key, either as separate files or combined into a single "bundled" file. The bundle or private key is often passphrase-protected.

Once you have any Certificate Authorities (CAs) imported you can optionally import any identity bundles into the authentication database. If you do not wish to store the identities, you can reference their component file system paths within an individual authentication configuration.

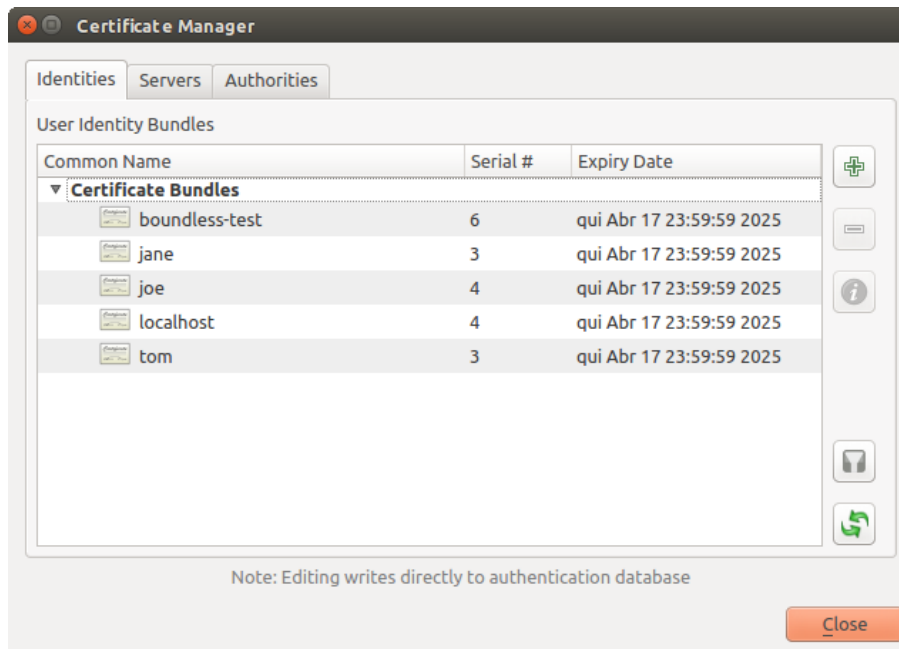


图 21.29: Identities editor

When importing an identity bundle, it can be passphrase-protected or unprotected, and can contain CA certificates forming a trust chain. Trust chain certifications will not be imported here; they can be added separately under the *Authorities* tab.

Upon import the bundle’s certificate and private key will be stored in the database, with the key’s storage encrypted using the QGIS master password. Subsequent usage of the stored bundle from the database will only require input of the master password.

Personal identity bundles consisting of PEM/DER (.pem/.der) and PKCS#12 (.p12/.pfx) components are supported. If a key or bundle is passphrase-protected, the password will be required to validate the component prior to import. Likewise, if the client certificate in the bundle is invalid (for example, its effective date has not yet started or has elapsed) the bundle can not be imported.

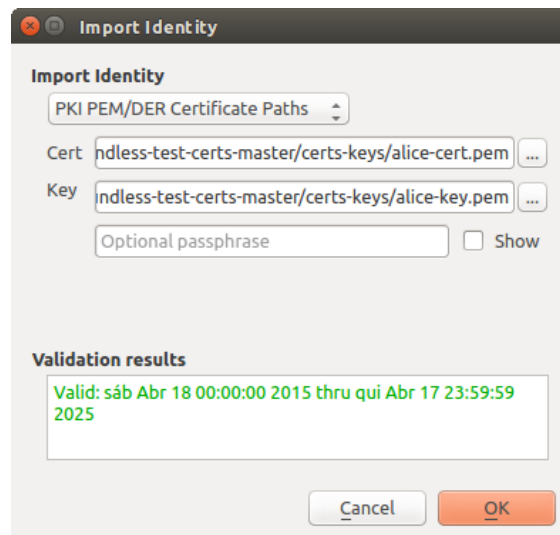


图 21.30: PEM/DER identity import

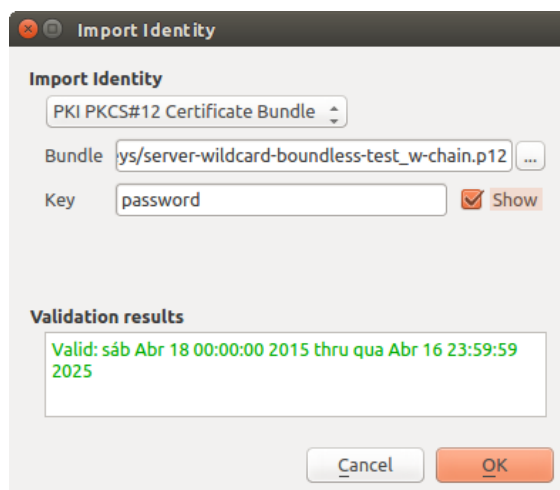


图 21.31: PKCS#12 identity import

21.2.4 Handling bad layers

Occasionally, the authentication configuration ID that is saved with a project file is no longer valid, possibly because the current authentication database is different than when the project was last saved, or due to a credentials mismatch. In such cases the *Handle bad layers* dialog will be presented upon QGIS launch.

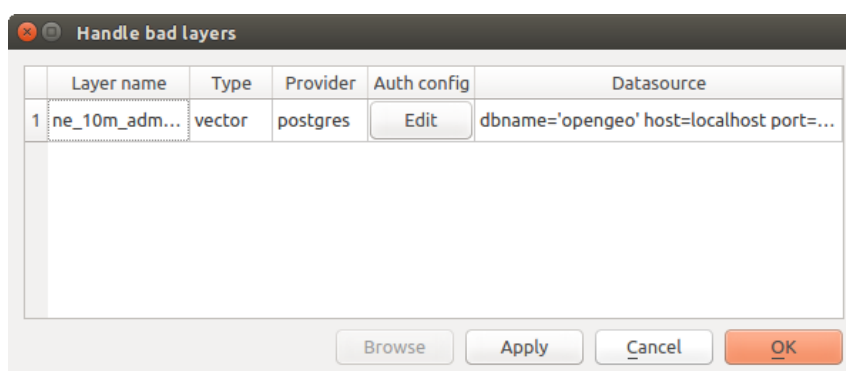


图 21.32: Handle bad layers with authentication

If a data source is found to have an authentication configuration ID associated with it, you will be able to edit it. Doing so will automatically edit the data source string, much in the same way as opening the project file in a text editor and editing the string.

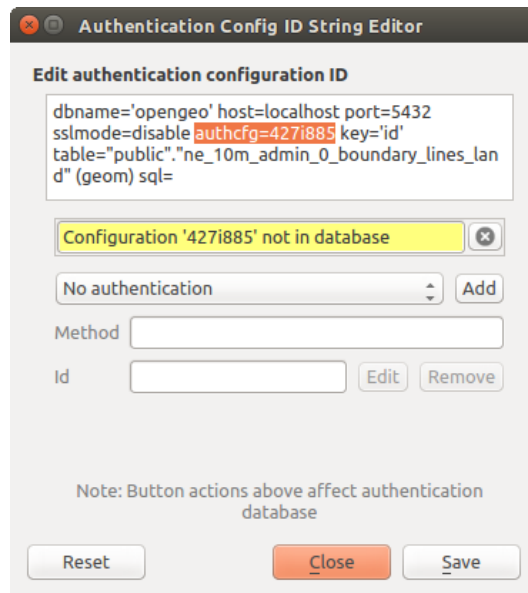


图 21.33: Edit bad layer’s authentication config ID

21.2.5 Changing authentication config ID

Occasionally, you will need to change the authentication configuration ID that is associated with accessing a resource. There are instances where this is useful:

- **Resource auth config ID is no longer valid:** This can occur when you have switched auth databases and need to *align* a new configuration to the ID already associated with a resource.
- **Shared project files:** If you intended to share projects between users, e.g. via a shared file server, you can *predefine* a 7-character (containing **a-z** and/or **0-9**) that is associated with the resource. Then, individual users change the ID of an authentication configuration that is specific to their credentials of the resource. When the project is opened, the ID is found in the authentication database, but the credentials are different per user.

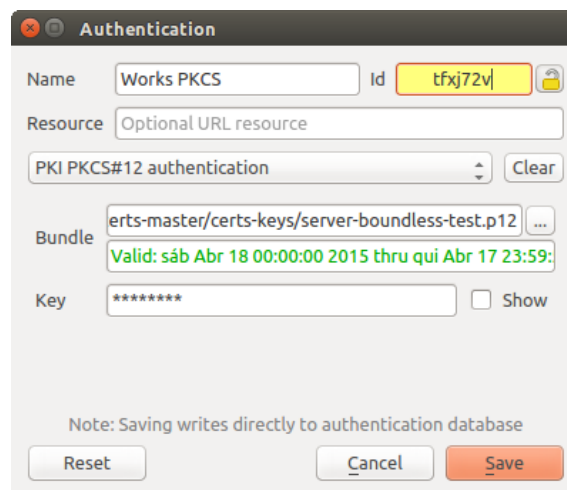


图 21.34: Changing a layer’s authentication config ID (unlocked yellow text field)

警告: Changing the auth config ID is considered an advanced operation and should only be done with full knowledge as to why it is necessary. This is why there is a lock button that needs clicked, to unlock the ID’s text field prior to editing the ID.

21.2.6 QGIS Server support

When using a project file, with layers that have authentication configurations, as a basis for a map in QGIS Server, there are a couple of additional setup steps necessary for QGIS to load the resources:

- Authentication database needs to be available
- Authentication database's master password needs to be available

When instantiating the authentication system, Server will create or use `qgis-auth.db` file in the active *user profile*, or the directory defined by the `QGIS_AUTH_DB_DIR_PATH` environment variable. It may be that the Server's user has no HOME directory, in which case, use the environment variable to define a directory that the Server's user has read/write permissions and is not located within the web-accessible directories.

To pass the master password to Server, write it to the first line of file at a path on the file system readable by the Server processes user and defined using the `QGIS_AUTH_PASSWORD_FILE` environment variable. Ensure to limit the file as only readable by the Server's process user and to not store the file within web-accessible directories.

注解: `QGIS_AUTH_PASSWORD_FILE` variable will be removed from the Server environment immediately after accessing.

21.2.7 SSL server exceptions

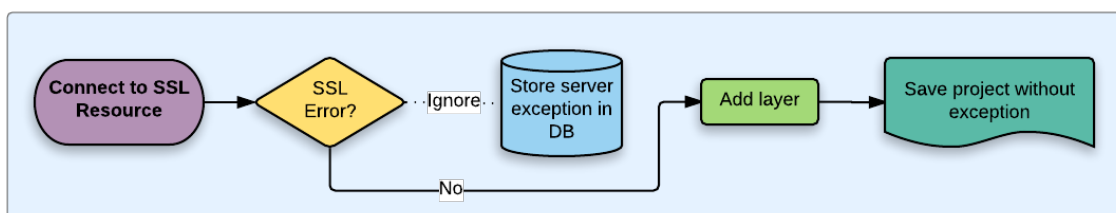



图 21.35: SSL server exception

You can manage SSL server configurations and exceptions from the **Servers** tab in the **Authentication** section of the QGIS **Options** dialog.

Sometimes, when connecting to an SSL server, there are errors with the SSL "handshake" or the server's certificate. You can ignore those errors or create an SSL server configuration as an exception. This is similar to how web browsers allow you to override SSL errors, but with more granular control.

警告: You should not create an SSL server configuration unless you have complete knowledge of the entire SSL setup between the server and client. Instead, report the issue to the server administrator.

注解: Some PKI setups use a completely different CA trust chain to validate client identities than the chain used to validate the SSL server certificate. In such circumstances, any configuration created for the connecting server will not necessarily fix an issue with the validation of your client identity, and only your client identity's issuer or server administrator can fix the issue.

You can pre-configure an SSL server configuration by clicking the  button. Alternatively, you can add a configuration when an SSL error occurs during a connection and you are presented with an **SSL Error** dialog (where the error can be ignored temporarily or saved to the database and ignored):

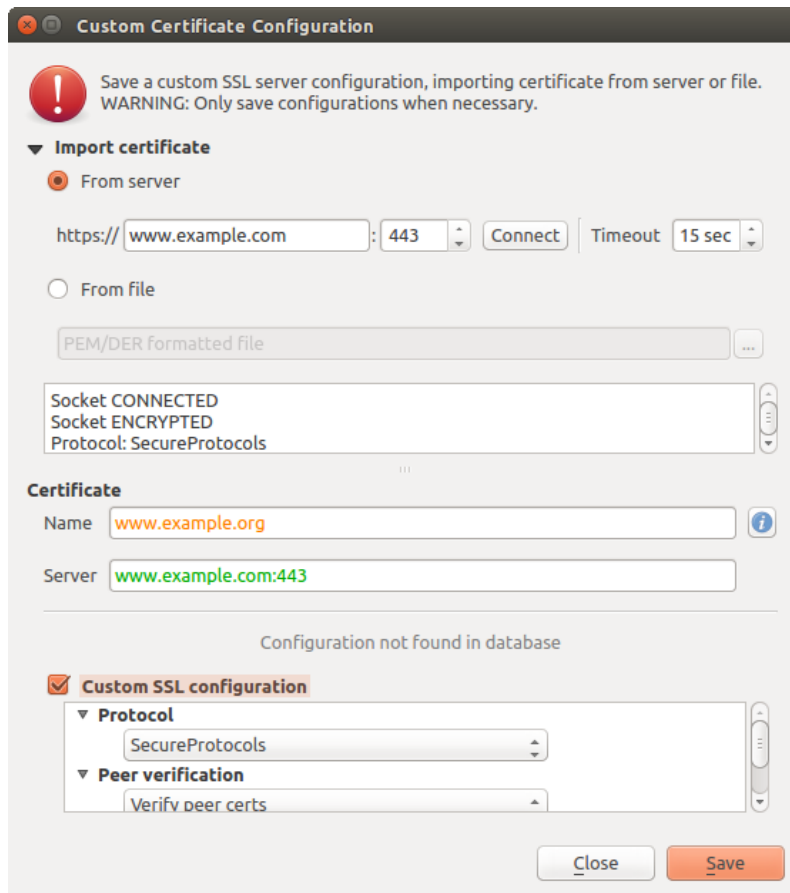


图 21.36: Manually adding configuration

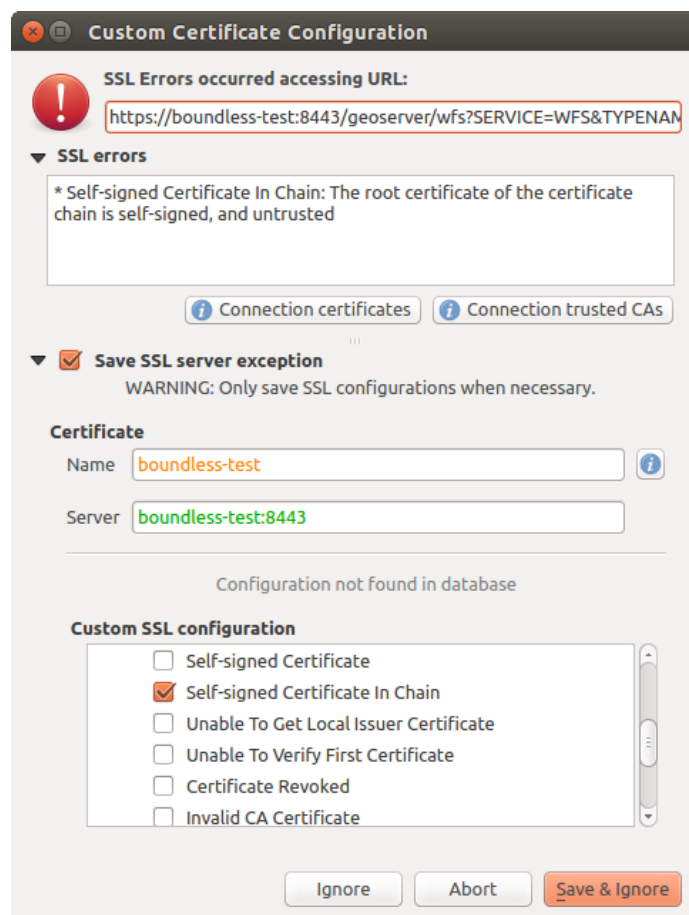


图 21.37: Adding configuration during SSL error

Once an SSL configuration is saved to the database, it can be edited or deleted.

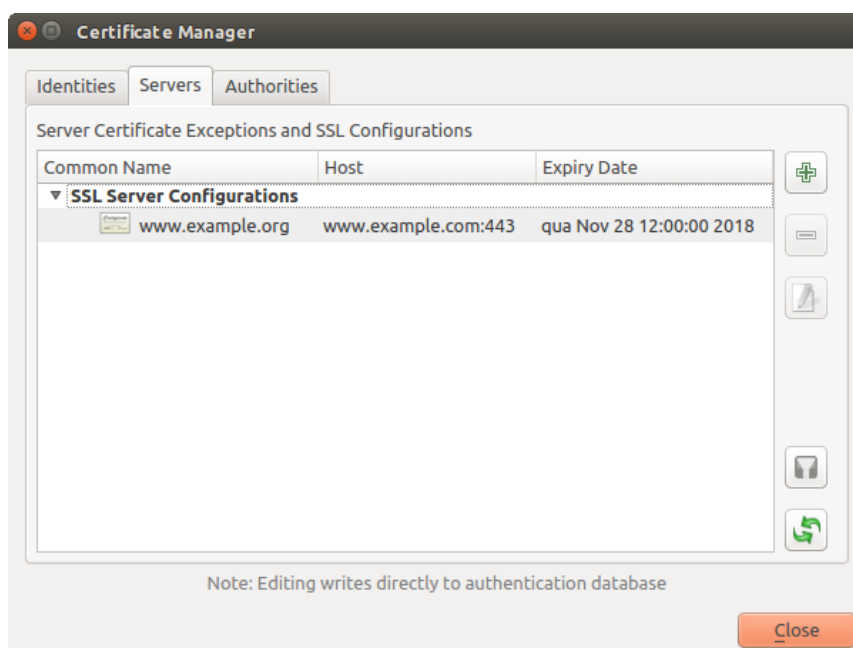


图 21.38: Existing SSL configuration

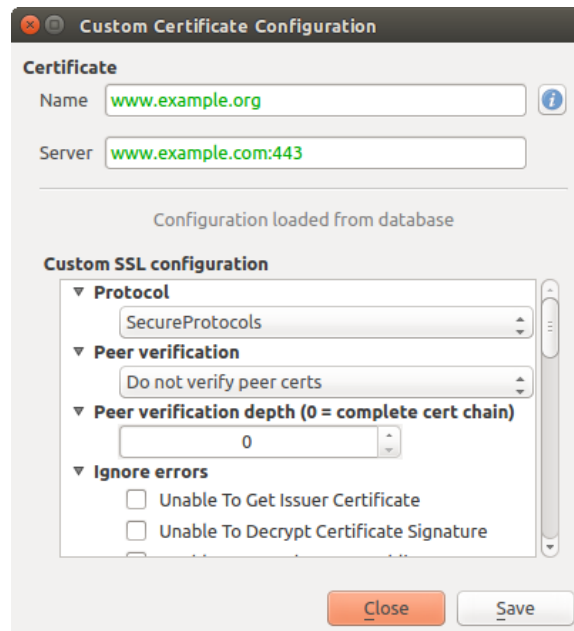


图 21.39: Editing an existing SSL configuration

If you want to pre-configure an SSL configuration and the import dialog is not working for your server's connection, you can manually trigger a connection via the **Python Console** by running the following code (replace `https://bugreports.qt-project.org` with the URL of your server):

```
from qgis.PyQt.QtNetwork import QNetworkRequest
from qgis.PyQt.QtCore import QUrl
from qgis.core import QgsNetworkAccessManager

req = QNetworkRequest(QUrl('https://bugreports.qt-project.org'))
reply = QgsNetworkAccessManager.instance().get(req)
```

This will open an SSL error dialog if any errors occur, where you can choose to save the configuration to the database.

21.3 Security Considerations

Once the master password is entered, the API is open to access authentication configs in the authentication database, similar to how Firefox works. However, in the initial implementation, no wall against PyQGIS access has been defined. This may lead to issues where a user downloads/installs a malicious PyQGIS plugin or standalone app that gains access to authentication credentials.

The quick solution for initial release of feature is to just not include most PyQGIS bindings for the authentication system.

Another simple, though not robust, fix is to add a combobox in *Settings ► Options ► Authentication* (defaults to "never"):

```
"Allow Python access to authentication system"
Choices: [ confirm once per session | always confirm | always allow | never]
```

Such an option's setting would need to be saved in a location non-accessible to Python, e.g. the authentication database, and encrypted with the master password.

- Another option may be to track which plugins the user has specifically
- allowed to access the authentication system, though it may be tricky to deduce which plugin is actually making the call.

- Sandboxing plugins, possibly in their own virtual environments, would reduce 'cross-plugin' hacking of authentication configs from another plugin that is authorized. This might mean limiting cross-plugin communication as well, but maybe only between third-party plugins.
- Another good solution is to issue code-signing certificates to vetted plugin authors. Then validate the plugin's certificate upon loading. If need be the user can also directly set an untrusted policy for the certificate associated with the plugin using existing certificate management dialogs.
- Alternatively, access to sensitive authentication system data from Python
- could never be allowed, and only the use of QGIS core widgets, or duplicating authentication system integrations, would allow the plugin to work with resources that have an authentication configuration, while keeping master password and authentication config loading in the realm of the main app.

The same security concerns apply to C++ plugins, though it will be harder to restrict access, since there is no function binding to simply be removed as with Python.

21.3.1 Restrictions

The confusing [licensing and exporting](#) issues associated with OpenSSL apply. In order for Qt to work with SSL certificates, it needs access to the OpenSSL libraries. Depending upon how Qt was compiled, the default is to dynamically link to the OpenSSL libs at run-time (to avoid the export limitations).

QCA follows a similar tactic, whereby linking to QCA incurs no restrictions, because the qca-openssl (OpenSSL) plugin is loaded at run-time. The qca-openssl plugin is directly linked to the OpenSSL libs. Packagers would be the ones needing to ensure any OpenSSL-linking restrictions are met, if they ship the plugin. Maybe. I don't really know. I'm not a lawyer.

The authentication system safely disables itself when `qca-openssl` is not found at run-time.

GRASS GIS Integration

GRASS integration provides access to GRASS GIS databases and functionalities (see GRASS-PROJECT in *Literature and Web References*). The integration consists of two parts: provider and plugin. The provider allows to browse, manage and visualize GRASS raster and vector layers. The plugin can be used to create new GRASS locations and mapsets, change GRASS region, create and edit vector layers and analyze GRASS 2-D and 3-D data with more than 400 GRASS modules. In this section, we'll introduce the provider and plugin functionalities and give some examples of managing and working with GRASS data.


The provider supports GRASS version 6 and 7, the plugin supports GRASS 6 and 7 (starting from QGIS 2.12). QGIS distribution may contain provider/plugin for either GRASS 6 or GRASS 7 or for both versions at the same time (binaries have different file names). Only one version of the provider/plugin may be loaded on runtime however.

22.1 Demo dataset

As an example, we will use the QGIS Alaska dataset (see section 下载示例数据). It includes a small sample GRASS LOCATION with three vector layers and one raster elevation map. Create a new folder called `grassdata`, download the QGIS 'Alaska' dataset `qgis_sample_data.zip` from <https://qgis.org/downloads/data/> and unzip the file into `grassdata`.

More sample GRASS LOCATIONS are available at the GRASS website at <https://grass.osgeo.org/download/sample-data/>.

22.2 Loading GRASS raster and vector layers

If the provider is loaded in QGIS, the location item with GRASS  icon is added in the browser tree under each folder item which contains GRASS location. Go to the folder `grassdata` and expand location `alaska` and mapset `demo`.


You can load GRASS raster and vector layers like any other layer from the browser either by double click on layer item or by dragging and dropping to map canvas or legend.

小技巧: GRASS Data Loading



If you don't see GRASS location item, verify in *Help* ► *About* ► *Providers* if GRASS vector provider is loaded.

22.3 Importing data into a GRASS LOCATION via drag and drop

This section gives an example of how to import raster and vector data into a GRASS mapset.

1. In QGIS browser navigate to the mapset you want to import data into.
2. In QGIS browser find a layer you want to import to GRASS, note that you can open another instance of the browser (*Browser Panel (2)*) if source data are too far from the mapset in the tree.
3. Drag a layer and drop it on the target mapset. The import may take some time for larger layers, you will see animated icon  in front of new layer item until the import finishes.

When raster data are in different CRS, they can be reprojected using an *Approximate* (fast) or *Exact* (precise) transformation. If a link to the source raster is created (using `r.external`), the source data are in the same CRS and the format is known to GDAL, the source data CRS will be used. You can set these options in the *Browser* tab in *GRASS Options*.

If a source raster has more bands, a new GRASS map is created for each layer with `.<band number>` suffix and group of all maps with  icon is created. External rasters have a different icon .


22.4 Managing GRASS data in QGIS Browser

- Copying maps: GRASS maps may be copied between mapsets within the same location using drag and drop.
- Deleting maps: Right click on a GRASS map and select *Delete* from context menu.
- Renaming maps: Right click on a GRASS map and select *Rename* from context menu.







22.5 GRASS Options

GRASS options may be set in *GRASS Options* dialog, which can be opened by right clicking on the location or mapset item in the browser and then choosing *GRASS Options*.

22.6 Starting the GRASS plugin

To use GRASS functionalities in QGIS, you must select and load the GRASS plugin using the Plugin Manager. To do this, go to the menu *Plugins* ►  *Manage and Install Plugins...*, select *GRASS* and click *OK*.

The following main features are provided with the GRASS menu (*Plugins* ► *GRASS*) when you start the GRASS plugin:

-  Open Mapset
-  New Mapset
-  Close Mapset
-  Open GRASS Tools
-  Display Current GRASS Region
-  GRASS Options

22.7 Opening GRASS mapset

A GRASS mapset must be opened to get access to GRASS Tools in the plugin (the tools are disabled if no mapset is open). You can open a mapset from the browser: right click on mapset item and then choose *Open mapset* from context menu.

22.8 GRASS LOCATION and MAPSET

GRASS data are stored in a directory referred to as GISDBASE. This directory, often called `grassdata`, must be created before you start working with the GRASS plugin in QGIS. Within this directory, the GRASS GIS data are organized by projects stored in subdirectories called `LOCATIONS`. Each `LOCATION` is defined by its coordinate system, map projection and geographical boundaries. Each `LOCATION` can have several `MAPSETs` (subdirectories of the `LOCATION`) that are used to subdivide the project into different topics or subregions, or as workspaces for individual team members (see Neteler & Mitasova 2008 in *Literature and Web References*). In order to analyse vector and raster layers with GRASS modules, you generally have to import them into a GRASS `LOCATION`. (This is not strictly true -- with the GRASS modules `r.external` and `v.external` you can create read-only links to external GDAL/OGR-supported datasets without importing them. This is not the usual way for beginners to work with GRASS, therefore this functionality will not be described here.)

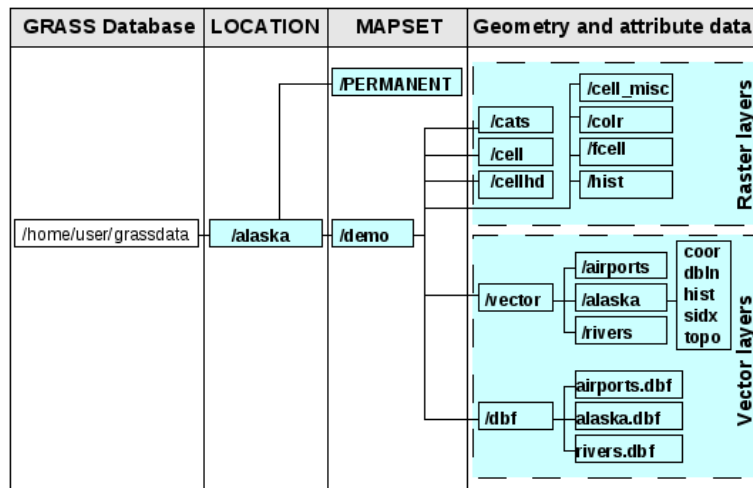




图 22.1: GRASS data in the alaska LOCATION

22.9 Importing data into a GRASS LOCATION

See section *Importing data into a GRASS LOCATION via drag and drop* to find how data can be easily imported by dragging and dropping in the browser.



This section gives an example of how to import raster and vector data into the 'alaska' GRASS `LOCATION` provided by the QGIS 'Alaska' dataset in traditional way, using standard GRASS modules. Therefore, we use the landcover raster map `landcover.img` and the vector GML file `lakes.gml` from the QGIS 'Alaska' dataset (see 下载示例数据).

1. Start QGIS and make sure the GRASS plugin is loaded.
2. In the GRASS toolbar, click the  Open MAPSET icon to bring up the *MAPSET* wizard.
3. Select as GRASS database the folder `grassdata` in the QGIS Alaska dataset, as `LOCATION` 'alaska', as `MAPSET` 'demo' and click *OK*.
4. Now click the  Open GRASS tools icon. The GRASS Toolbox (see section *The GRASS Toolbox*) dialog appears.

5. To import the raster map `landcover.img`, click the module `r.in.gdal` in the *Modules Tree* tab. This GRASS module allows you to import GDAL-supported raster files into a GRASS LOCATION. The module dialog for `r.in.gdal` appears.
6. Browse to the folder `raster` in the QGIS 'Alaska' dataset and select the file `landcover.img`.
7. As raster output name, define `landcover_grass` and click *Run*. In the *Output* tab, you see the currently running GRASS command `r.in.gdal -o input=/path/to/landcover.img output=landcover_grass`.
8. When it says **Successfully finished**, click *View Output*. The `landcover_grass` raster layer is now imported into GRASS and will be visualized in the QGIS canvas.
9. To import the vector GML file `lakes.gml`, click the module `v.in.ogr` in the *Modules Tree* tab. This GRASS module allows you to import OGR-supported vector files into a GRASS LOCATION. The module dialog for `v.in.ogr` appears.
10. Browse to the folder `gml` in the QGIS 'Alaska' dataset and select the file `lakes.gml` as OGR file.
11. As vector output name, define `lakes_grass` and click *Run*. You don't have to care about the other options in this example. In the *Output* tab you see the currently running GRASS command `v.in.ogr -o dsn=/path/to/lakes.gml output=lakes_grass`.
12. When it says **Successfully finished**, click *View Output*. The `lakes_grass` vector layer is now imported into GRASS and will be visualized in the QGIS canvas.

22.9.1 Creating a new GRASS LOCATION

As an example, here is the sample GRASS LOCATION `alaska`, which is projected in the Albers Equal Area projection using feet as units. This sample GRASS LOCATION `alaska` will be used for all examples and exercises in the following GRASS-related sections. It is useful to download and install the dataset on your computer (see [下载示例数据](#)).

1. Start QGIS and make sure the GRASS plugin is loaded.
2. Visualize the `alaska.shp` shapefile (see section *Loading a layer from a file*) from the QGIS Alaska dataset (see [下载示例数据](#)).
3. In the GRASS toolbar, click on the  `New mapset` icon to bring up the *MAPSET* wizard.
4. Select an existing GRASS database (GISDBASE) folder `grassdata`, or create one for the new LOCATION using a file manager on your computer. Then click *Next*.
5. We can use this wizard to create a new MAPSET within an existing LOCATION (see section *Adding a new MAPSET*) or to create a new LOCATION altogether. Select *Create new location* (see [图 22.2](#)).
6. Enter a name for the LOCATION -- we used 'alaska' -- and click *Next*.
7. Define the projection by clicking on the radio button *Projection* to enable the projection list.
8. We are using Albers Equal Area Alaska (feet) projection. Since we happen to know that it is represented by the EPSG ID 2964, we enter it in the search box. (Note: If you want to repeat this process for another LOCATION and projection and haven't memorized the EPSG ID, click on the  `CRS Status` icon in the lower right-hand corner of the status bar (see section *Working with Projections*)).
9. In *Filter*, insert 2964 to select the projection.
10. Click *Next*.
11. To define the default region, we have to enter the LOCATION bounds in the north, south, east, and west directions. Here, we simply click on the button *Set Current QGIS Extent*, to apply the extent of the loaded layer `alaska.shp` as the GRASS default region extent.
12. Click *Next*.

13. We also need to define a MAPSET within our new LOCATION (this is necessary when creating a new LOCATION). You can name it whatever you like - we used 'demo'. GRASS automatically creates a special MAPSET called PERMANENT, designed to store the core data for the project, its default spatial extent and coordinate system definitions (see Neteler & Mitasova 2008 in *Literature and Web References*).
14. Check out the summary to make sure it's correct and click *Finish*.
15. The new LOCATION, 'alaska', and two MAPSETS, 'demo' and 'PERMANENT', are created. The currently opened working set is 'demo', as you defined.
16. Notice that some of the tools in the GRASS toolbar that were disabled are now enabled.

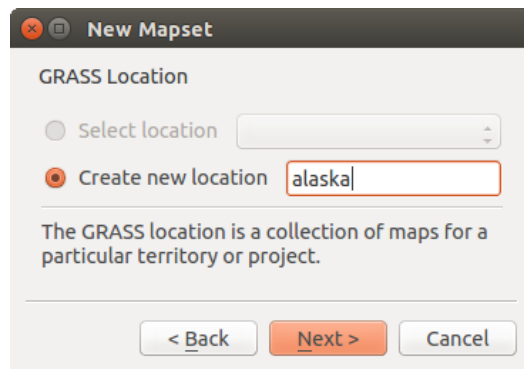




图 22.2: Creating a new GRASS LOCATION or a new MAPSET in QGIS

If that seemed like a lot of steps, it's really not all that bad and a very quick way to create a LOCATION. The LOCATION 'alaska' is now ready for data import (see section *Importing data into a GRASS LOCATION*). You can also use the already-existing vector and raster data in the sample GRASS LOCATION 'alaska', included in the QGIS 'Alaska' dataset 下载示例数据, and move on to section *The GRASS vector data model*.

22.9.2 Adding a new MAPSET

A user has write access only to a GRASS MAPSET which he or she created. This means that besides access to your own MAPSET, you can read maps in other users' MAPSETS (and they can read yours), but you can modify or remove only the maps in your own MAPSET.

All MAPSETS include a WIND file that stores the current boundary coordinate values and the currently selected raster resolution (see Neteler & Mitasova 2008 in *Literature and Web References*, and section *The GRASS region tool*).

1. Start QGIS and make sure the GRASS plugin is loaded.
2. In the GRASS toolbar, click on the  icon to bring up the MAPSET wizard.
3. Select the GRASS database (GISDBASE) folder `grassdata` with the LOCATION 'alaska', where we want to add a further MAPSET called 'test'.
4. Click *Next*.
5. We can use this wizard to create a new MAPSET within an existing LOCATION or to create a new LOCATION altogether. Click on the radio button  *Select location* (see 图 22.2) and click *Next*.
6. Enter the name `test` for the new MAPSET. Below in the wizard, you see a list of existing MAPSETS and corresponding owners.
7. Click *Next*, check out the summary to make sure it's all correct and click *Finish*.

22.10 The GRASS vector data model

It is important to understand the GRASS vector data model prior to digitizing. In general, GRASS uses a topological vector model. This means that areas are not represented as closed polygons, but by one or more boundaries. A boundary between two adjacent areas is digitized only once, and it is shared by both areas. Boundaries must be connected and closed without gaps. An area is identified (and labelled) by the **centroid** of the area.

Besides boundaries and centroids, a vector map can also contain points and lines. All these geometry elements can be mixed in one vector and will be represented in different so-called 'layers' inside one GRASS vector map. So in GRASS, a layer is not a vector or raster map but a level inside a vector layer. This is important to distinguish carefully. (Although it is possible to mix geometry elements, it is unusual and, even in GRASS, only used in special cases such as vector network analysis. Normally, you should prefer to store different geometry elements in different layers.)

It is possible to store several 'layers' in one vector dataset. For example, fields, forests and lakes can be stored in one vector. An adjacent forest and lake can share the same boundary, but they have separate attribute tables. It is also possible to attach attributes to boundaries. An example might be the case where the boundary between a lake and a forest is a road, so it can have a different attribute table.

The 'layer' of the feature is defined by the 'layer' inside GRASS. 'Layer' is the number which defines if there is more than one layer inside the dataset (e.g., if the geometry is forest or lake). For now, it can be only a number. In the future, GRASS will also support names as fields in the user interface.

Attributes can be stored inside the GRASS LOCATION as dBase, SQLite3 or in external database tables, for example, PostgreSQL, MySQL, Oracle, etc.

Attributes in database tables are linked to geometry elements using a 'category' value.

'Category' (key, ID) is an integer attached to geometry primitives, and it is used as the link to one key column in the database table.

小技巧: Learning the GRASS Vector Model

The best way to learn the GRASS vector model and its capabilities is to download one of the many GRASS tutorials where the vector model is described more deeply. See <https://grass.osgeo.org/documentation/manuals/> for more information, books and tutorials in several languages.

22.11 Creating a new GRASS vector layer

To create a new GRASS vector layer, select one of following items from mapset context menu in the browser:

- New Point Layer
- New Line Layer
- New Polygon Layer

and enter a name in the dialog. A new vector map will be created and layer will be added to canvas and editing started. Selecting type of the layer does not restrict geometry types which can be digitized in the vector map. In GRASS, it is possible to organize all sorts of geometry types (point, line and polygon) in one vector map. The type is only used to add the layer to the canvas, because QGIS requires a layer to have a specific type.

It is also possible to add layers to existing vector maps selecting one of the items described above from context menu of existing vector map.

In GRASS, it is possible to organize all sorts of geometry types (point, line and area) in one layer, because GRASS uses a topological vector model, so you don't need to select the geometry type when creating a new GRASS vector. This is different from shapefile creation with QGIS, because shapefiles use the Simple Feature vector model (see section *Creating new vector layers*).

22.12 Digitizing and editing a GRASS vector layer

GRASS vector layers can be digitized using the standard QGIS digitizing tools. There are however some particularities, which you should know about, due to

- GRASS topological model versus QGIS simple feature
- complexity of GRASS model
 - multiple layers in single maps
 - multiple geometry types in single map
 - geometry sharing by multiple features from multiple layers

The particularities are discussed in the following sections.

Save, discard changes, undo, redo

警告: All the changes done during editing are immediately written to vector map and related attribute tables.

Changes are written after each operation, it is however, possible to do undo/redo or discard all changes when closing editing. If undo or discard changes is used, original state is rewritten in vector map and attribute tables.

There are two main reasons for this behaviour:

- It is the nature of GRASS vectors coming from conviction that user wants to do what he is doing and it is better to have data saved when the work is suddenly interrupted (for example, blackout)
- Necessity for effective editing of topological data is visualized information about topological correctness, such information can only be acquired from GRASS vector map if changes are written to the map.

Toolbar

The 'Digitizing Toolbar' has some specific tools when a GRASS layer is edited:






Icon	Tool	目标
	New Point	Digitize new point
	New Line	Digitize new line
	New Boundary	Digitize new boundary
	New Centroid	Digitize new centroid (label existing area)
	New Closed Boundary	Digitize new closed boundary

Table GRASS Digitizing: GRASS Digitizing Tools

小技巧: Digitizing polygons in GRASS

If you want to create a polygon in GRASS, you first digitize the boundary of the polygon. Then you add a centroid (label point) into the closed boundary. The reason for this is that a topological vector model links the attribute information of a polygon always to the centroid and not to the boundary.

Category

Category, often called cat, is sort of ID. The name comes from times when GRASS vectors had only singly attribute "category". Category is used as a link between geometry and attributes. A single geometry may have multiple categories and thus represent multiple features in different layers. Currently it is possible to assign only one category per layer using QGIS editing tools. New features have automatically assigned new unique category, except boundaries. Boundaries usually only form areas and do not represent linear features, it is however possible to define attributes for a boundary later, for example in different layer.

New categories are always created only in currently being edited layer.

It is not possible to assign more categories to geometry using QGIS editing, such data are properly represented as multiple features, and individual features, even from different layers, may be deleted.

Attributes

Attributes of currently edited layer can only be modified. If the vector map contains more layers, features of other layers will have all attributes set to '<not editable (layer #)>' to warn you that such attribute is not editable. The reason is, that other layers may have and usually have different set of fields while QGIS only supports one fixed set of fields per layer.

If a geometry primitive does not have a category assigned, a new unique category is automatically assigned and new record in attribute table is created when an attribute of that geometry is changed.

小技巧: If you want to do bulk update of attributes in table, for example using 'Field Calculator' (*Using the Field Calculator*), and there are features without category which you don't want to update (typically boundaries), you can filter them out by setting 'Advanced Filter' to `cat is not null`.

Editing style

The topological symbology is essential for effective editing of topological data. When editing starts, a specialized 'GRASS Edit' renderer is set on the layer automatically and original renderer is restored when editing is closed. The style may be customized in layer properties 'Style' tab. The style can also be stored in project file or in separate file as any other style. If you customize the style, do not change its name, because it is used to reset the style when editing is started again.

小技巧: Do not save project file when the layer is edited, the layer would be stored with 'Edit Style' which has no meaning if layer is not edited.

The style is based on topological information which is temporarily added to attribute table as field 'topo_symbol'. The field is automatically removed when editing is closed.

小技巧: Do not remove 'topo_symbol' field from attribute table, that would make features invisible because the renderer is based on that column.


Snapping

To form an area, vertices of connected boundaries must have **exactly** the same coordinates. This can be achieved using snapping tool only if canvas and vector map have the same CRS. Otherwise, due conversion from map coordinates to canvas and back, the coordinate may become slightly different due to representation error and CRS transformations.

小技巧: Use layer's CRS also for canvas when editing.

Limitations

Simultaneous editing of multiple layers within the same vector at the same time is not supported. This is mainly due to the impossibility of handling multiple undo stacks for a single data source.


 On Linux and macOS only one GRASS layer can be edited at time. This is due to a bug in GRASS which does not allow to close database drivers in random order. This is being solved with GRASS developers.

小技巧: GRASS Edit Permissions

You must be the owner of the GRASS MAPSET you want to edit. It is impossible to edit data layers in a MAPSET that is not yours, even if you have write permission.

22.13 The GRASS region tool


The region definition (setting a spatial working window) in GRASS is important for working with raster layers. Vector analysis is by default not limited to any defined region definitions. But all newly created rasters will have the spatial extension and resolution of the currently defined GRASS region, regardless of their original extension and resolution. The current GRASS region is stored in the `$LOCATION/$MAPSET/WIND` file, and it defines north, south, east and west bounds, number of columns and rows, horizontal and vertical spatial resolution.

It is possible to switch on and off the visualization of the GRASS region in the QGIS canvas using the  `Display current GRASS region` button.

The region can be modified in 'Region' tab in 'GRASS Tolls' dock widget. Type in the new region bounds and resolution, and click *Apply*. If you click on *Select the extent by dragging on canvas* you can select a new region interactively with your mouse on the QGIS canvas dragging a rectangle.

The GRASS module `g.region` provides a lot more parameters to define an appropriate region extent and resolution for your raster analysis. You can use these parameters with the GRASS Toolbox, described in section *The GRASS Toolbox*.

22.14 The GRASS Toolbox

The  `Open GRASS Tools` box provides GRASS module functionalities to work with data inside a selected GRASS `LOCATION` and `MAPSET`. To use the GRASS Toolbox you need to open a `LOCATION` and `MAPSET` that you have write permission for (usually granted, if you created the `MAPSET`). This is necessary, because new raster or vector layers created during analysis need to be written to the currently selected `LOCATION` and `MAPSET`.

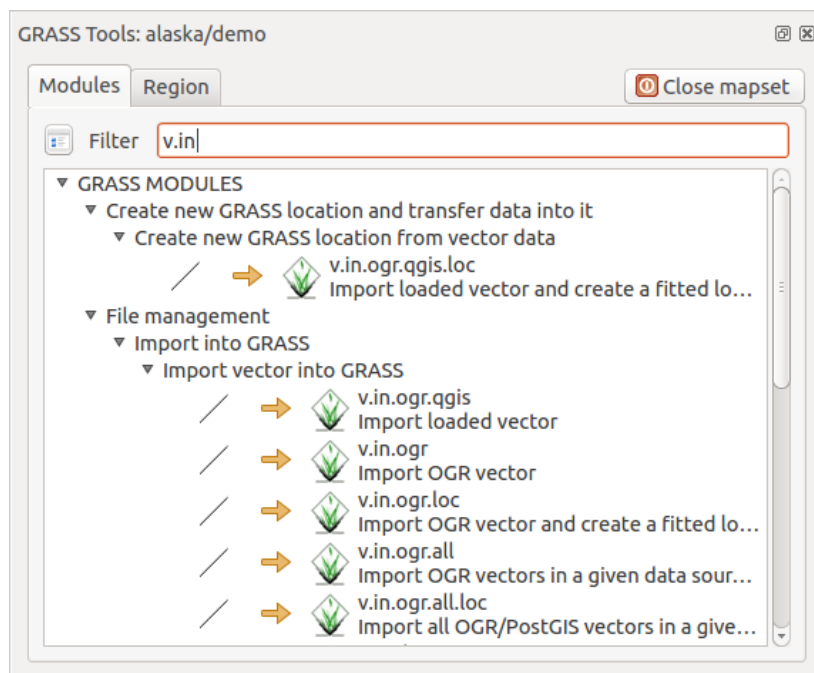


图 22.3: GRASS Toolbox and Module Tree

22.14.1 Working with GRASS modules

The GRASS shell inside the GRASS Toolbox provides access to almost all (more than 300) GRASS modules in a command line interface. To offer a more user-friendly working environment, about 200 of the available GRASS modules and functionalities are also provided by graphical dialogs within the GRASS plugin Toolbox.

A complete list of GRASS modules available in the graphical Toolbox in QGIS version 3.16 is available in the GRASS wiki at https://grasswiki.osgeo.org/wiki/GRASS-QGIS_relevant_module_list.

It is also possible to customize the GRASS Toolbox content. This procedure is described in section *Customizing the GRASS Toolbox*.

As shown in [图 22.3](#), you can look for the appropriate GRASS module using the thematically grouped *Modules Tree* or the searchable *Modules List* tab.

By clicking on a graphical module icon, a new tab will be added to the Toolbox dialog, providing three new sub-tabs: *Options*, *Output* and *Manual*.

Options

The *Options* tab provides a simplified module dialog where you can usually select a raster or vector layer visualized in the QGIS canvas and enter further module-specific parameters to run the module.

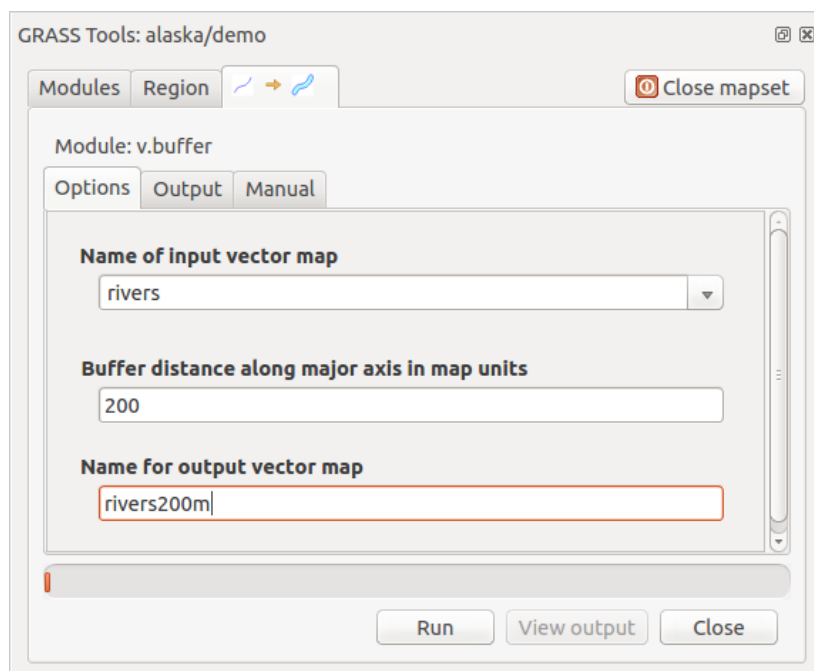


图 22.4: GRASS Toolbox Module Options

The provided module parameters are often not complete to keep the dialog simple. If you want to use further module parameters and flags, you need to start the GRASS shell and run the module in the command line.

A new feature since QGIS 1.8 is the support for a *Show Advanced Options* button below the simplified module dialog in the *Options* tab. At the moment, it is only added to the module `v.in.ascii` as an example of use, but it will probably be part of more or all modules in the GRASS Toolbox in future versions of QGIS. This allows you to use the complete GRASS module options without the need to switch to the GRASS shell.

Output

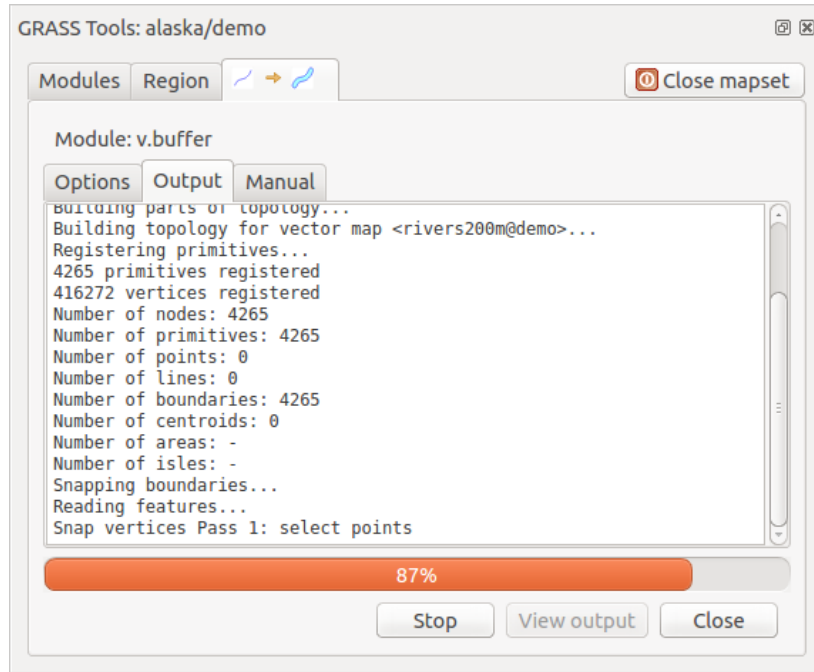


图 22.5: GRASS Toolbox Module Output

The *Output* tab provides information about the output status of the module. When you click the *Run* button, the module switches to the *Output* tab and you see information about the analysis process. If all works well, you will finally see a *Successfully finished* message.

Manual

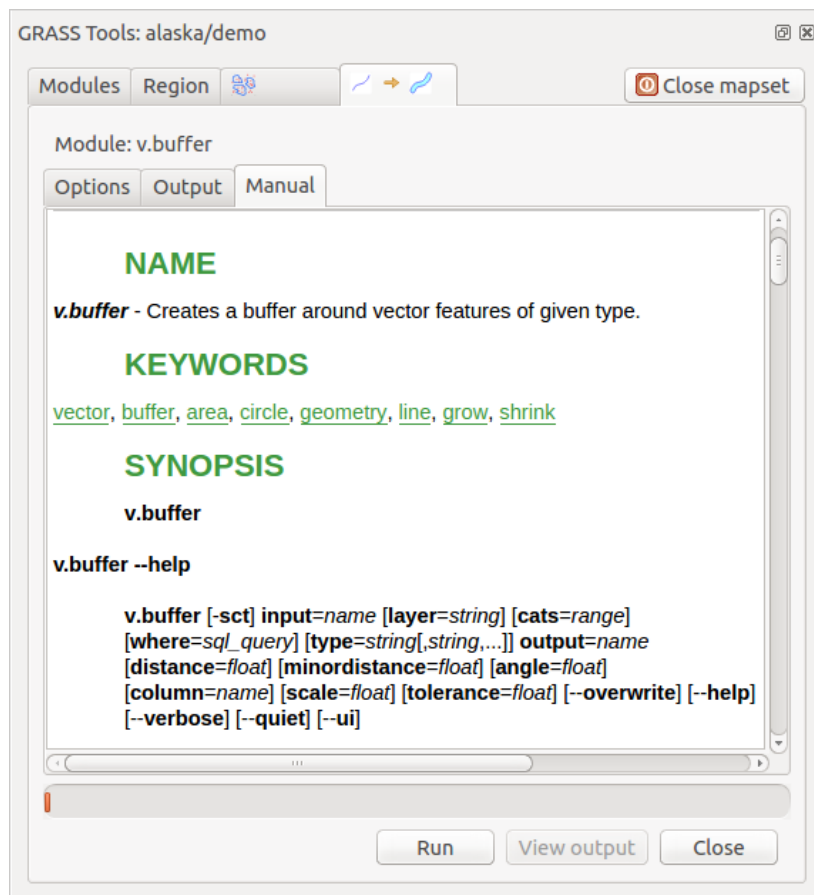


图 22.6: GRASS Toolbox Module Manual

The *Manual* tab shows the HTML help page of the GRASS module. You can use it to check further module parameters and flags or to get a deeper knowledge about the purpose of the module. At the end of each module manual page, you see further links to the `Main Help index`, the `Thematic index` and the `Full index`. These links provide the same information as the module `g.manual`.

小技巧: Display results immediately



If you want to display your calculation results immediately in your map canvas, you can use the 'View Output' button at the bottom of the module tab.

22.14.2 GRASS module examples

The following examples will demonstrate the power of some of the GRASS modules.

Creating contour lines

The first example creates a vector contour map from an elevation raster (DEM). Here, it is assumed that you have the Alaska LOCATION set up as explained in section *Importing data into a GRASS LOCATION*.

- First, open the location by clicking the  Open mapset button and choosing the Alaska location.
- Now open the Toolbox with the  Open GRASS tools button.
- In the list of tool categories, double-click *Raster* ► *Surface Management* ► *Generate vector contour lines*.
- Now a single click on the tool **r.contour** will open the tool dialog as explained above (see *Working with GRASS modules*).
- In the *Name of input raster map* enter `gtopo30`.
- Type into the *Increment between Contour levels* the value 100. (This will create contour lines at intervals of 100 meters.)
- Type into the *Name for output vector map* the name `ctour_100`.
- Click *Run* to start the process. Wait for several moments until the message `Successfully finished` appears in the output window. Then click *View Output* and *Close*.

Since this is a large region, it will take a while to display. After it finishes rendering, you can open the layer properties window to change the line color so that the contours appear clearly over the elevation raster, as in *The Vector Properties Dialog*.

Next, zoom in to a small, mountainous area in the center of Alaska. Zooming in close, you will notice that the contours have sharp corners. GRASS offers the **v.generalize** tool to slightly alter vector maps while keeping their overall shape. The tool uses several different algorithms with different purposes. Some of the algorithms (i.e., Douglas Peuker and Vertex Reduction) simplify the line by removing some of the vertices. The resulting vector will load faster. This process is useful when you have a highly detailed vector, but you are creating a very small-scale map, so the detail is unnecessary.

小技巧: The simplify tool

Note that QGIS has a *Vector* ► *Geometry Tools* ► *Simplify geometries* tool that works just like the GRASS **v.generalize** Douglas-Peuker algorithm.

However, the purpose of this example is different. The contour lines created by **r.contour** have sharp angles that should be smoothed. Among the **v.generalize** algorithms, there is Chaiken's, which does just that (also Hermite splines). Be aware that these algorithms can **add** additional vertices to the vector, causing it to load even more slowly.

- Open the GRASS Toolbox and double-click the categories *Vector* ► *Develop map* ► *Generalization*, then click on the **v.generalize** module to open its options window.
- Check that the `'ctour_100'` vector appears as the *Name of input vector*.
- From the list of algorithms, choose Chaiken's. Leave all other options at their default, and scroll down to the last row to enter in the field *Name for output vector map* `'ctour_100_smooth'`, and click *Run*.
- The process takes several moments. Once `Successfully finished` appears in the output windows, click *View Output* and then *Close*.
- You may change the color of the vector to display it clearly on the raster background and to contrast with the original contour lines. You will notice that the new contour lines have smoother corners than the original while staying faithful to the original overall shape.

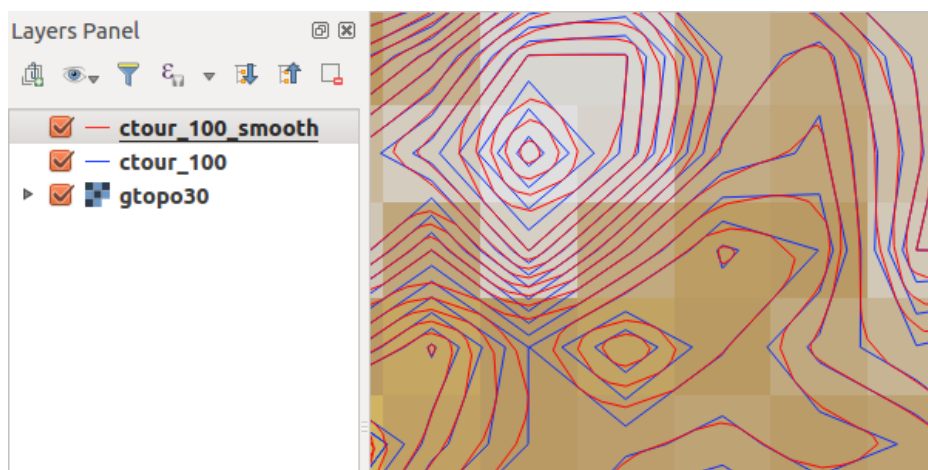


图 22.7: GRASS module v.generalize to smooth a vector map

小技巧: Other uses for r.contour

The procedure described above can be used in other equivalent situations. If you have a raster map of precipitation data, for example, then the same method will be used to create a vector map of isohyetal (constant rainfall) lines.

Creating a Hillshade 3-D effect

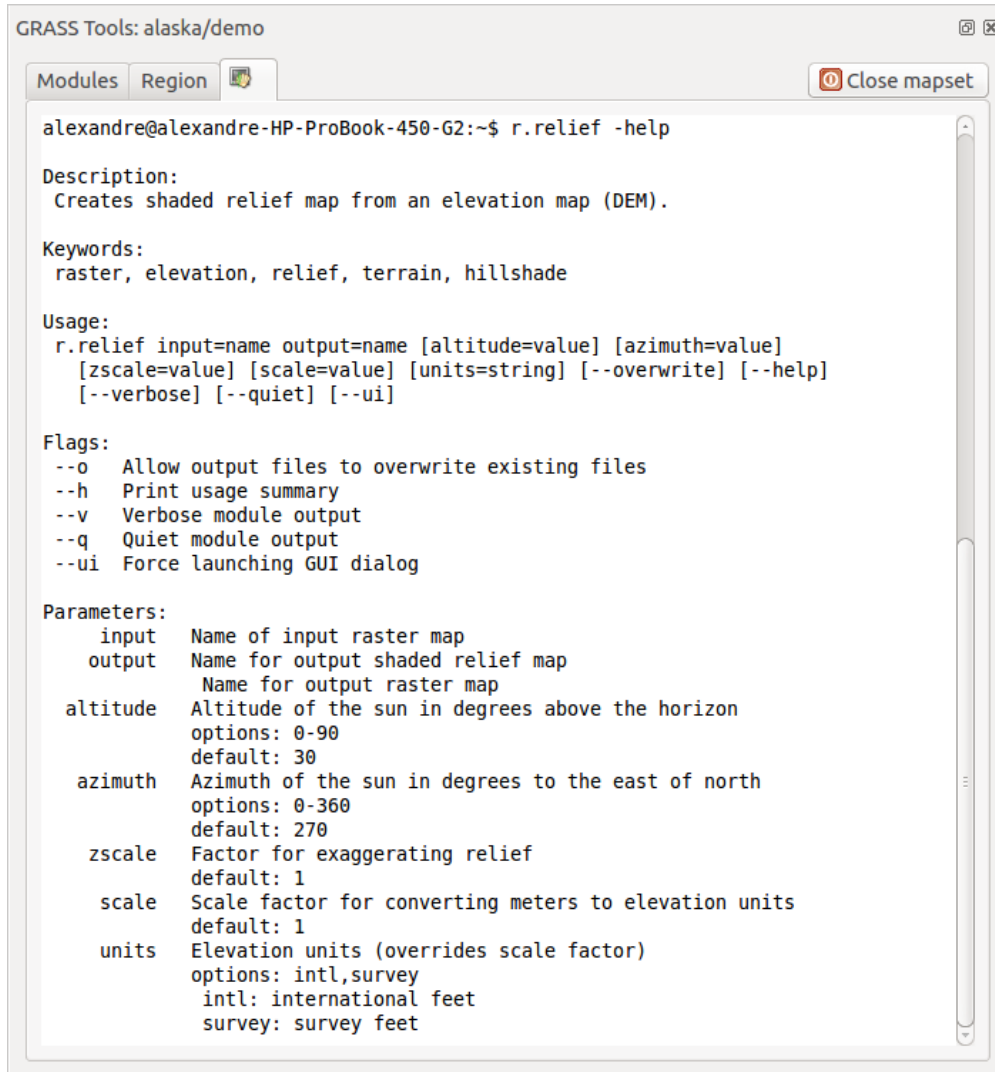
Several methods are used to display elevation layers and give a 3-D effect to maps. The use of contour lines, as shown above, is one popular method often chosen to produce topographic maps. Another way to display a 3-D effect is by hillshading. The hillshade effect is created from a DEM (elevation) raster by first calculating the slope and aspect of each cell, then simulating the sun's position in the sky and giving a reflectance value to each cell. Thus, you get sun-facing slopes lighted; the slopes facing away from the sun (in shadow) are darkened.

- Begin this example by loading the `gtopo30` elevation raster. Start the GRASS Toolbox, and under the Raster category, double-click to open *Spatial analysis* ► *Terrain analysis*.
- Then click **r.shaded.relief** to open the module.
- Change the *azimuth angle* 270 to 315.
- Enter `gtopo30_shade` for the new hillshade raster, and click *Run*.
- When the process completes, add the hillshade raster to the map. You should see it displayed in grayscale.
- To view both the hillshading and the colors of the `gtopo30` together, move the hillshade map below the `gtopo30` map in the table of contents, then open the *Properties* window of `gtopo30`, switch to the *Transparency* tab and set its transparency level to about 25%.

You should now have the `gtopo30` elevation with its colormap and transparency setting displayed **above** the grayscale hillshade map. In order to see the visual effects of the hillshading, turn off the `gtopo30_shade` map, then turn it back on.

Using the GRASS shell

The GRASS plugin in QGIS is designed for users who are new to GRASS and not familiar with all the modules and options. As such, some modules in the Toolbox do not show all the options available, and some modules do not appear at all. The GRASS shell (or console) gives the user access to those additional GRASS modules that do not appear in the Toolbox tree, and also to some additional options to the modules that are in the Toolbox with the simplest default parameters. This example demonstrates the use of an additional option in the **r.shaded.relief** module that was shown above.



```

GRASS Tools: alaska/demo
Modules Region
Close mapset

alexandre@alexandre-HP-ProBook-450-G2:~$ r.shaded.relief -help

Description:
  Creates shaded relief map from an elevation map (DEM).

Keywords:
  raster, elevation, relief, terrain, hillshade

Usage:
  r.shaded.relief input=name output=name [altitude=value] [azimuth=value]
  [zscale=value] [scale=value] [units=string] [--overwrite] [--help]
  [--verbose] [--quiet] [--ui]

Flags:
  --o Allow output files to overwrite existing files
  --h Print usage summary
  --v Verbose module output
  --q Quiet module output
  --ui Force launching GUI dialog

Parameters:
  input      Name of input raster map
  output     Name for output shaded relief map
             Name for output raster map
  altitude   Altitude of the sun in degrees above the horizon
             options: 0-90
             default: 30
  azimuth    Azimuth of the sun in degrees to the east of north
             options: 0-360
             default: 270
  zscale     Factor for exaggerating relief
             default: 1
  scale      Scale factor for converting meters to elevation units
             default: 1
  units      Elevation units (overrides scale factor)
             options: intl,survey
             intl: international feet
             survey: survey feet

```

图 22.8: The GRASS shell, r.shaded.relief module

The module **r.shaded.relief** can take a parameter `zmult`, which multiplies the elevation values relative to the X-Y coordinate units so that the hillshade effect is even more pronounced.

- Load the `gtopo30` elevation raster as above, then start the GRASS Toolbox and click on the GRASS shell. In the shell window, type the command `r.shaded.relief map=gtopo30 shade=gtopo30_shade2 azimuth=315 zmult=3` and press Enter.
- After the process finishes, shift to the *Browse* tab and double-click on the new `gtopo30_shade2` raster to display it in QGIS.
- As explained above, move the shaded relief raster below the `gtopo30` raster in the table of contents, then check the transparency of the colored `gtopo30` layer. You should see that the 3-D effect stands out more strongly compared with the first shaded relief map.

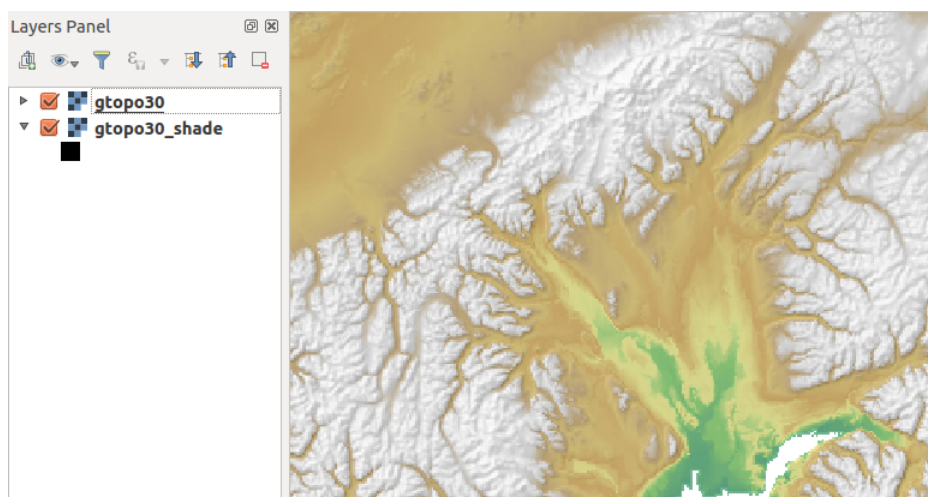


图 22.9: Displaying shaded relief created with the GRASS module `r.shaded.relief`

Raster statistics in a vector map

The next example shows how a GRASS module can aggregate raster data and add columns of statistics for each polygon in a vector map.

- Again using the Alaska data, refer to *Importing data into a GRASS LOCATION* to import the shapefiles/ `trees.shp` file into GRASS.
- Now an intermediate step is required: centroids must be added to the imported trees map to make it a complete GRASS area vector (including both boundaries and centroids).
- From the Toolbox, choose *Vector ► Manage features*, and open the module `v.centroids`.
- Enter as the *output vector map* 'forest_areas' and run the module.
- Now load the `forest_areas` vector and display the types of forests - deciduous, evergreen, mixed - in different colors: In the layer *Properties* window, *Symbology* tab, choose from *Legend type* 'Unique value' and set the *Classification field* to 'VEGDESC'. (Refer to the explanation of the symbology tab in *Symbology Properties* of the vector section.)
- Next, reopen the GRASS Toolbox and open *Vector ► Vector update by other maps*.
- Click on the `v.rast.stats` module. Enter `gtopo30` and `forest_areas`.
- Only one additional parameter is needed: Enter *column prefix* `elev`, and click *Run*. This is a computationally heavy operation, which will run for a long time (probably up to two hours).
- Finally, open the `forest_areas` attribute table, and verify that several new columns have been added, including `elev_min`, `elev_max`, `elev_mean`, etc., for each forest polygon.

22.14.3 Customizing the GRASS Toolbox

Nearly all GRASS modules can be added to the GRASS Toolbox. An XML interface is provided to parse the pretty simple XML files that configure the modules' appearance and parameters inside the Toolbox.

A sample XML file for generating the module `v.buffer` (`v.buffer.qgm`) looks like this:

```
<?xml version="1.0" encoding="UTF-8"?>
<!DOCTYPE qgisgrassmodule SYSTEM "http://mrcc.com/qgisgrassmodule.dtd">

<qgisgrassmodule label="Vector buffer" module="v.buffer">
  <option key="input" typeoption="type" layeroption="layer" />
</qgisgrassmodule>
```

(下页继续)

(续上页)


```
<option key="buffer"/>
<option key="output" />
</qgisgrassmodule>
```

The parser reads this definition and creates a new tab inside the Toolbox when you select the module. A more detailed description for adding new modules, changing a module's group, etc., can be found at <https://qgis.org/en/site/getinvolved/development/addinggrasstools.html>.

23.1 简介

This chapter introduces the QGIS processing framework, a geoprocessing environment that can be used to call native and third-party algorithms from QGIS, making your spatial analysis tasks more productive and easy to accomplish.

As a *Core plugin*, Processing is installed by default but you need to activate it:

1. Go to *Plugins ► Manage and install plugins...*
2. Click on the *Installed* tab at the left
3. Check the box next to the  *Processing* entry
4. Close the dialog.

A *Processing* menu is now available in the top menu bar. From there you can reach the main components of this framework.

In the following sections, we will review how to use the graphical elements of this framework and make the most out of each one of them.

There are four basic elements in the framework GUI, which are used to run algorithms for different purposes. Choosing one tool or another will depend on the kind of analysis that is to be performed and the particular characteristics of each user and project. All of them (except for the batch processing interface, which is called from the toolbox or the algorithm execution dialog, as we will see) can be accessed from the *Processing* menu item (you will see more entries; the remaining ones are not used to execute algorithms and will be explained later in this chapter).

- The *Toolbox*: The main element of the GUI, it is used to execute a single algorithm or run a batch process based on that algorithm.

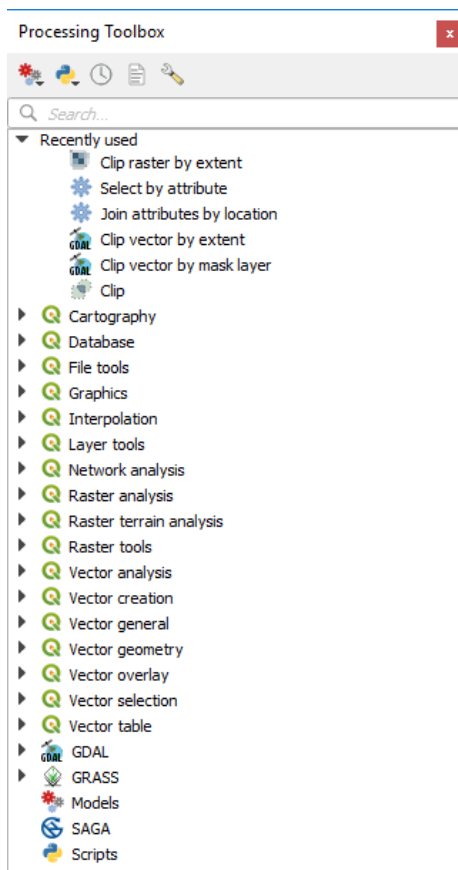


图 23.1: 处理工具箱

- The *Graphical Modeler*: Several algorithms can be combined graphically using the modeler to define a workflow, creating a single process that involves several subprocesses.

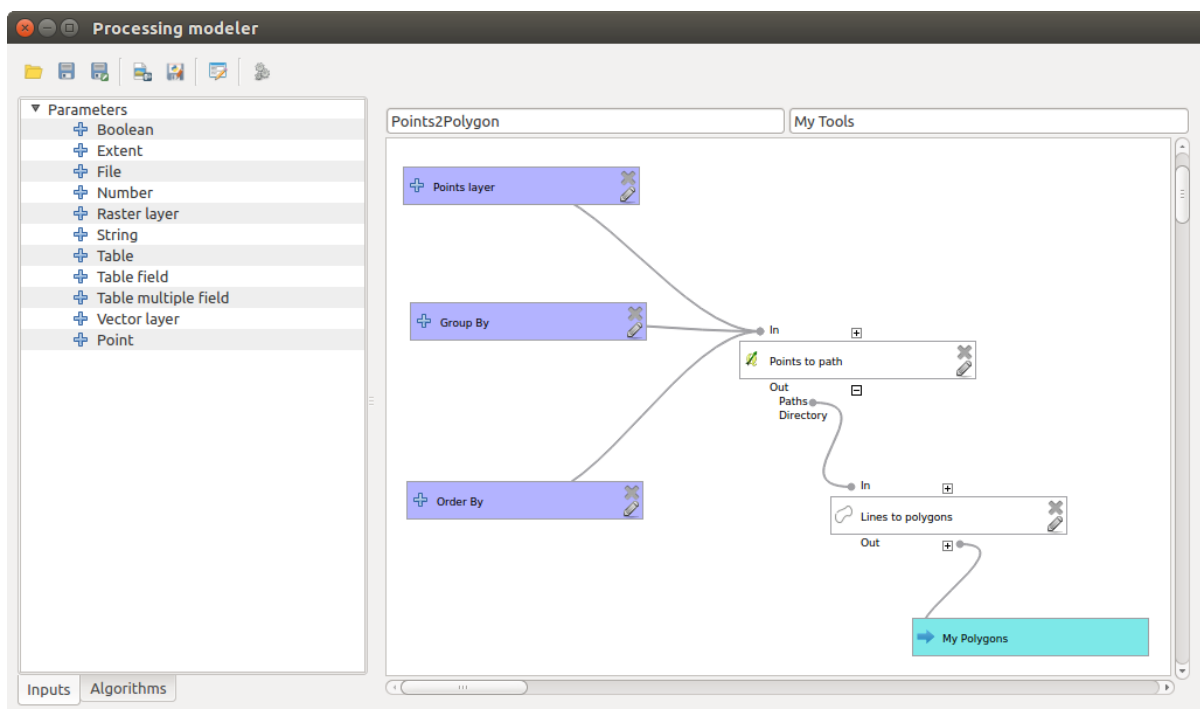


图 23.2: 处理模型

- The *History* manager: All actions performed using any of the aforementioned elements are stored in a history file and can be later easily reproduced using the history manager.

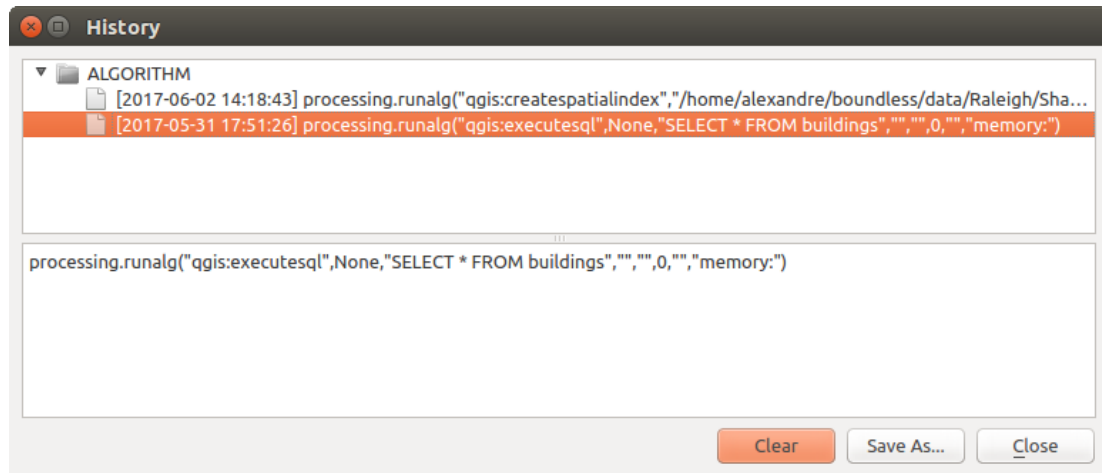


图 23.3: Processing History

- The *Batch Processing* interface: This interface allows you to execute batch processes and automate the execution of a single algorithm on multiple datasets.

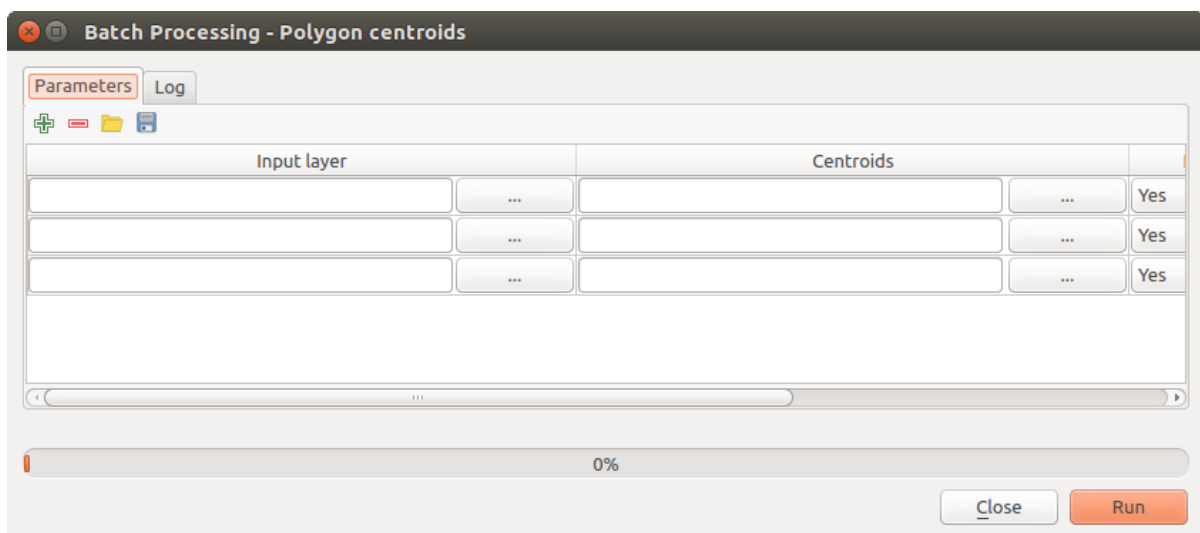


图 23.4: Batch Processing interface

In the following sections, we will review each one of these elements in detail.

23.2 Configuring the Processing Framework

The Processing Options menu (*Settings* ► *Options* ► *Processing* tab) allows you to configure how algorithms work. Configuration parameters are structured in separate blocks that you can select on the left-hand side of the dialog.

The *General* block contains a number of interesting parameters.

- *Default output raster layer extension* is by default `tif`
- *Default output vector layer extension* is by default `gpkg`
- *Invalid features filtering*

- *Keep dialog open after running algorithm.* Once an algorithm has finished execution and its output layers are loaded into the QGIS project, the algorithm dialog is closed. If you want to keep it open (to run the algorithm again with different parameters, or to better check the output that is written to the log tab), check this option.
- *Max Threads*
- *Output folder*
- *Pre-execution script* and *Post-execution script.* These parameters point to files that contain scripts written using the processing scripting functionality, explained in the section covering scripting and the console.
- *Prefer output filename for layer names.* The name of each resulting layer created by an algorithm is defined by the algorithm itself. In some cases, a fixed name might be used, meaning that the same output name will be used, no matter which input layer is used. In other cases, the name might depend on the name of the input layer or some of the parameters used to run the algorithm. If this checkbox is checked, the name will be taken from the output filename instead. Notice that, if the output is saved to a temporary file, the filename of this temporary file is usually a long and meaningless one intended to avoid collision with other already existing filenames.
- *Results group name.* If you want to obtain all processing result layers in a group in the *Layers* panel, set a group name for this parameter. The group may exist already or not. QGIS will add all output layers to such a group. By default, this parameter is empty, so all output layers are added to different places in the *Layers* panel, depending on the item that is active when running an algorithm. Note that output layers will be loaded to the *Layers* panel only if *Open output file after running algorithm* is checked in the algorithm dialog.
- *Show algorithms with known issues*
- *Show layer CRS definition in selection boxes*
- *Show tooltip when there are disabled providers*
- *Style for line layers, Style for point layers, Style for polygons layers* and *Style for raster layers* are used for setting the default rendering style for output layers (that is, layers generated by processing algorithms). Just create the style you want using QGIS, save it to a file, and then enter the path to that file in the settings so the algorithms can use it. Whenever a layer is loaded by Processing and added to the QGIS canvas, it will be rendered with that style.

Rendering styles can be configured individually for each algorithm and each one of its outputs. Just right-click on the name of the algorithm in the toolbox and select *Edit rendering styles for outputs*. You will see a dialog like the one shown next.

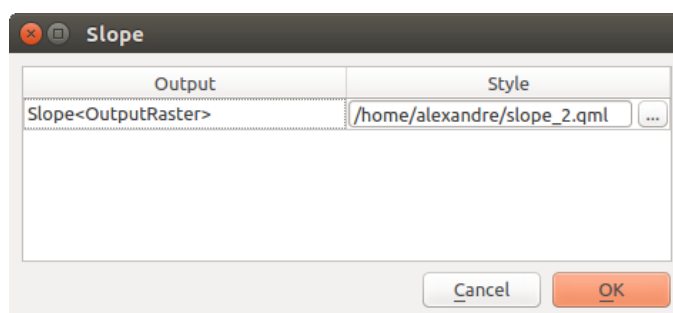


图 23.5: Rendering Styles

Select the style file (.qml) that you want for each output and press *OK*.

- *Temporary output folder path*
- *Warn before executing if parameter CRS's do not match*

You will also find a block for algorithm *Providers*. Each entry in this block contains an *Activate* item that you can use to make algorithms appear or not in the toolbox. Some algorithm providers have their own configuration items, which will be explained when covering particular algorithm providers.

23.3 The Toolbox

The *Processing Toolbox* is the main element of the processing GUI, and the one that you are more likely to use in your daily work. It shows the list of all available **algorithms** grouped in different blocks called *Providers*, and custom **models** and **scripts** you can add to extend the set of tools. Hence the toolbox is the access point to run them, whether as a single process or as a batch process involving several executions of the same algorithm on different sets of inputs.

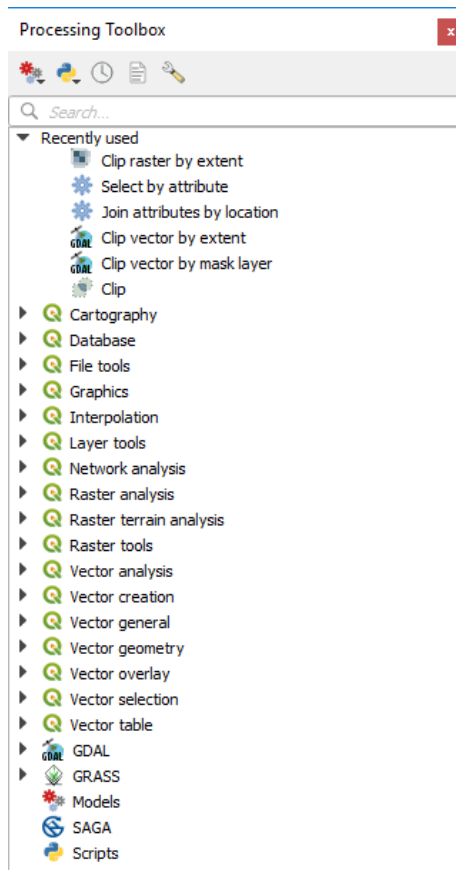









图 23.6: 处理工具箱

Providers can be (de)activated in the *Processing settings dialog*. By default, only providers that do not rely on third-party applications (that is, those that only require QGIS elements to be run) are active. Algorithms requiring external applications might need additional configuration. Configuring providers is explained in a *later chapter* in this manual.

In the upper part of the toolbox dialog, you will find a set of tools to:

- work with  **Models**: *Create New Model...*, *Open Existing Model...* and *Add Model to Toolbox...*;
- work with  **Scripts**: *Create New Script...*, *Create New Script from Template...*, *Open Existing Script...* and *Add Script to Toolbox...*;
- open the  **History** panel;
- open the  **Results Viewer** panel;
- toggle the toolbox to the *in-place modification mode* using the  **Edit Features In-Place** button: only the algorithms that are suitable to be executed on the active layer without outputting a new layer are displayed;
- open the  **Options** dialog.

Below this toolbar is a  *Search...* box to help you easily find the tools you need. You can enter any word or phrase on the text box. Notice that, as you type, the number of algorithms, models or scripts in the toolbox is reduced to just those that contain the text you have entered in their names or keywords.

注解: At the top of the list of algorithms are displayed the most recent used tools; handy if you want to reexecute any.

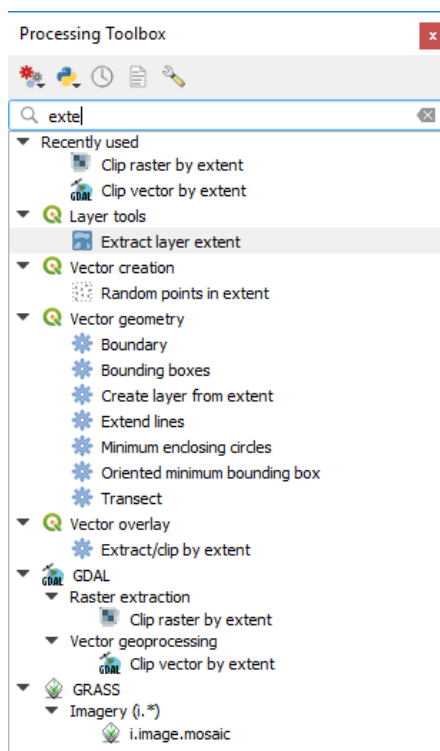


图 23.7: Processing Toolbox showing search results

To execute a tool, just double-click on its name in the toolbox.

23.3.1 The algorithm dialog

Once you double-click on the name of the algorithm that you want to execute, a dialog similar to that in the [图 23.8](#) below is shown (in this case, the dialog corresponds to the `Centroids` algorithm).

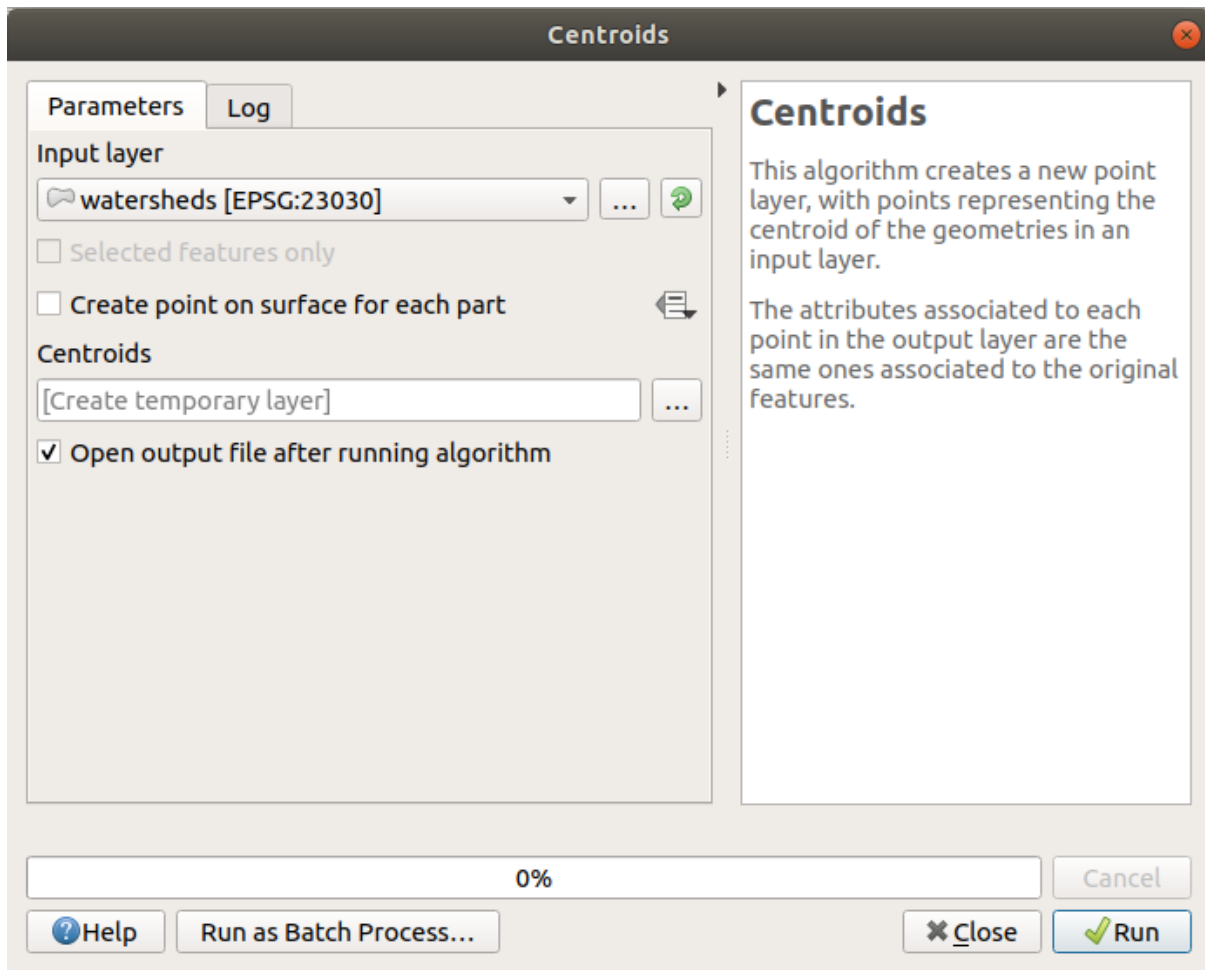


图 23.8: Algorithm Dialog - Parameters

The dialog shows two tabs (*Parameters* and *Log*) on the left part, the algorithm description on the right, and a set of buttons at the bottom.

The *Parameters* tab is used to set the input values that the algorithm needs to be executed. It shows a list of input values and configuration parameters to be set. It of course has a different content, depending on the requirements of the algorithm to be executed, and is created automatically based on those requirements.

Although the number and type of parameters depend on the characteristics of the algorithm, the structure is similar for all of them. The parameters found in the table can be of one of the following types.

- A **raster layer**, to select from a list of all such layers available (currently opened) in QGIS. The selector contains as well a button on its right-hand side, to let you select filenames that represent layers currently not loaded in QGIS.
- A **vector layer**, to select from a list of all vector layers available in QGIS. Layers not loaded in QGIS can be selected as well, as in the case of raster layers, but only if the algorithm does not require a table field selected from the attributes table of the layer. In that case, only opened layers can be selected, since they need to be open so as to retrieve the list of field names available.

You will see an iterator button by each vector layer selector, as shown in the figure below.

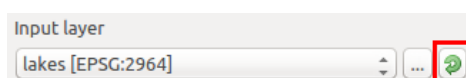



图 23.9: Vector iterator button

If the algorithm contains several of them, you will be able to toggle just one of them. If the button corresponding to a vector input is toggled, the algorithm will be executed iteratively on each one of its features, instead of just once for the whole layer, producing as many outputs as times the algorithm is executed. This allows for automating the process when all features in a layer have to be processed separately.

注解: By default, the parameters dialog will show a description of the CRS of each layer along with its name. If you do not want to see this additional information, you can disable this functionality in the Processing Settings dialog, unchecking the *General* ► *Show layer CRS definition in selection boxes* option.

- A **table**, to select from a list of all available in QGIS. Non-spatial tables are loaded into QGIS like vector layers, and in fact they are treated as such by the program. Currently, the list of available tables that you will see when executing an algorithm that needs one of them is restricted to tables coming from files in dBase (.dbf) or Comma-Separated Values (.csv) formats.
- An **option**, to choose from a selection list of possible options.
- A **numerical value**, to be introduced in a spin box. In some contexts (when the parameter applies at the feature level and not at the layer's), you will find a  button by its side, allowing you to open the *expression builder* and enter a mathematical expression to generate variable values for the parameter. Some useful variables related to data loaded into QGIS can be added to your expression, so you can select a value derived from any of these variables, such as the cell size of a layer or the northernmost coordinate of another one.

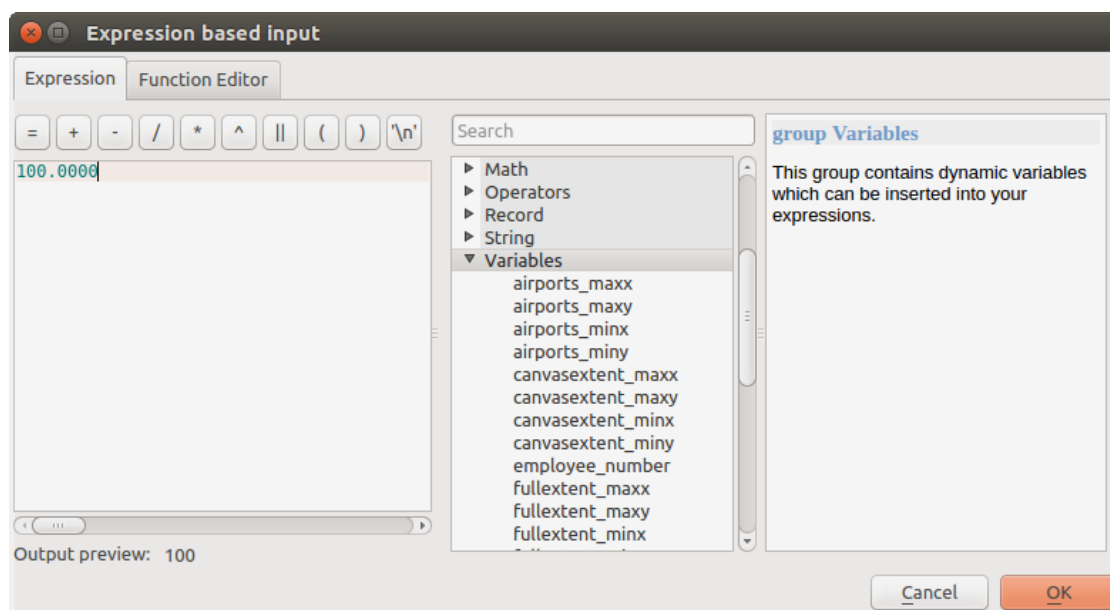


图 23.10: 基于表达式的输入

- A **range**, with min and max values to be introduced in two text boxes.
- A **text string**, to be introduced in a text box.
- A **field**, to choose from the attributes table of a vector layer or a single table selected in another parameter.
- A **coordinate reference system**. You can select it among the recently used ones from the drop-down list or from the *CRS selection* dialog that appears when you click on the button on the right-hand side.
- An **extent**, a text box defining a rectangle through its corners coordinate in the format *xmin, xmax, ymin, ymax*. Clicking on the button on the right-hand side of the value selector, a pop-up menu will appear, giving you options to:
 - *Calculate from layer*: fills the text box with the coordinates of the bounding box of a layer to select among the loaded ones

- *Use map canvas extent*
- *Draw on canvas*: the parameters window will hide itself, so you can click and drag onto the canvas. Once you have defined the extent rectangle, the dialog will reappear, containing the values in the extent text box.

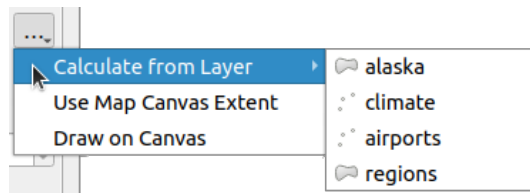


图 23.11: Extent selector

- A **list of elements** (whether raster or vector layers, tables, fields) to select from. Click on the ... button at the left of the option to see a dialog like the following one. Multiple selection is allowed and when the dialog is closed, number of selected items is displayed in the parameter text box widget.

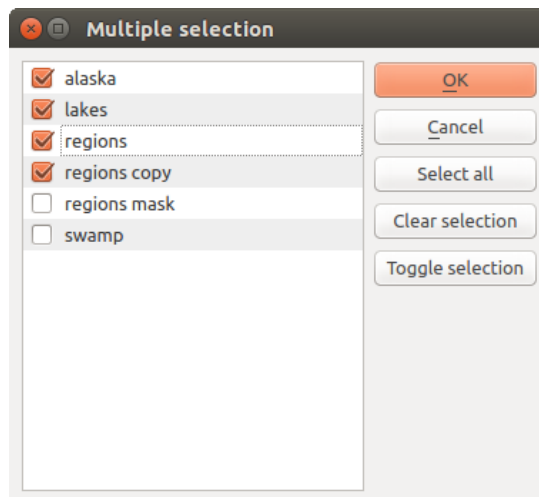


图 23.12: Multiple Selection

- A **small table** to be edited by the user. These are used to define parameters like lookup tables or convolution kernels, among others.

Click on the button on the right side to see the table and edit its values.

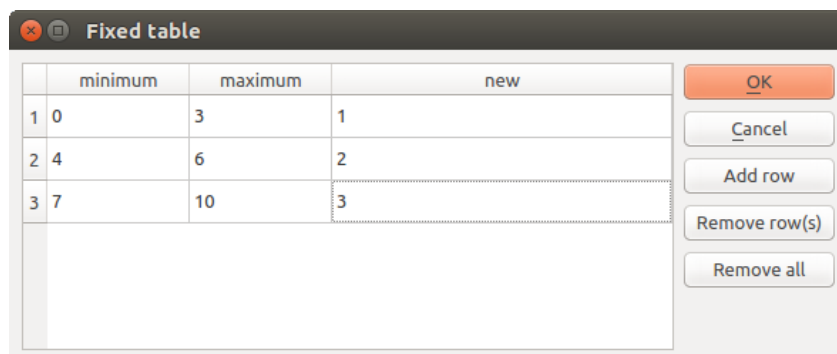


图 23.13: Fixed Table

Depending on the algorithm, the number of rows can be modified or not by using the buttons on the right side of the window.

注解: Some algorithms require many parameters to run, e.g. in the *Raster calculator* you have to specify manually the cell size, the extent and the CRS. You can avoid to choose all the parameters manually when the algorithm has the `Reference layers` parameter. With this parameter you can choose the reference layer and all its properties (cell size, extent, CRS) will be used.

Along with the *Parameters* tab, there is another tab named *Log* (see 图 23.14 below). Information provided by the algorithm during its execution is written in this tab, and allow you to track the execution and be aware and have more details about the algorithm as it runs. Information on algorithm execution is also output in the *View* ► *Panels* ► *Log Messages Panel*.

Notice that not all algorithms write information to the *Log* tab, and many of them might run silently without producing any output other than the final files. Check the *Log Messages Panel* in that case.

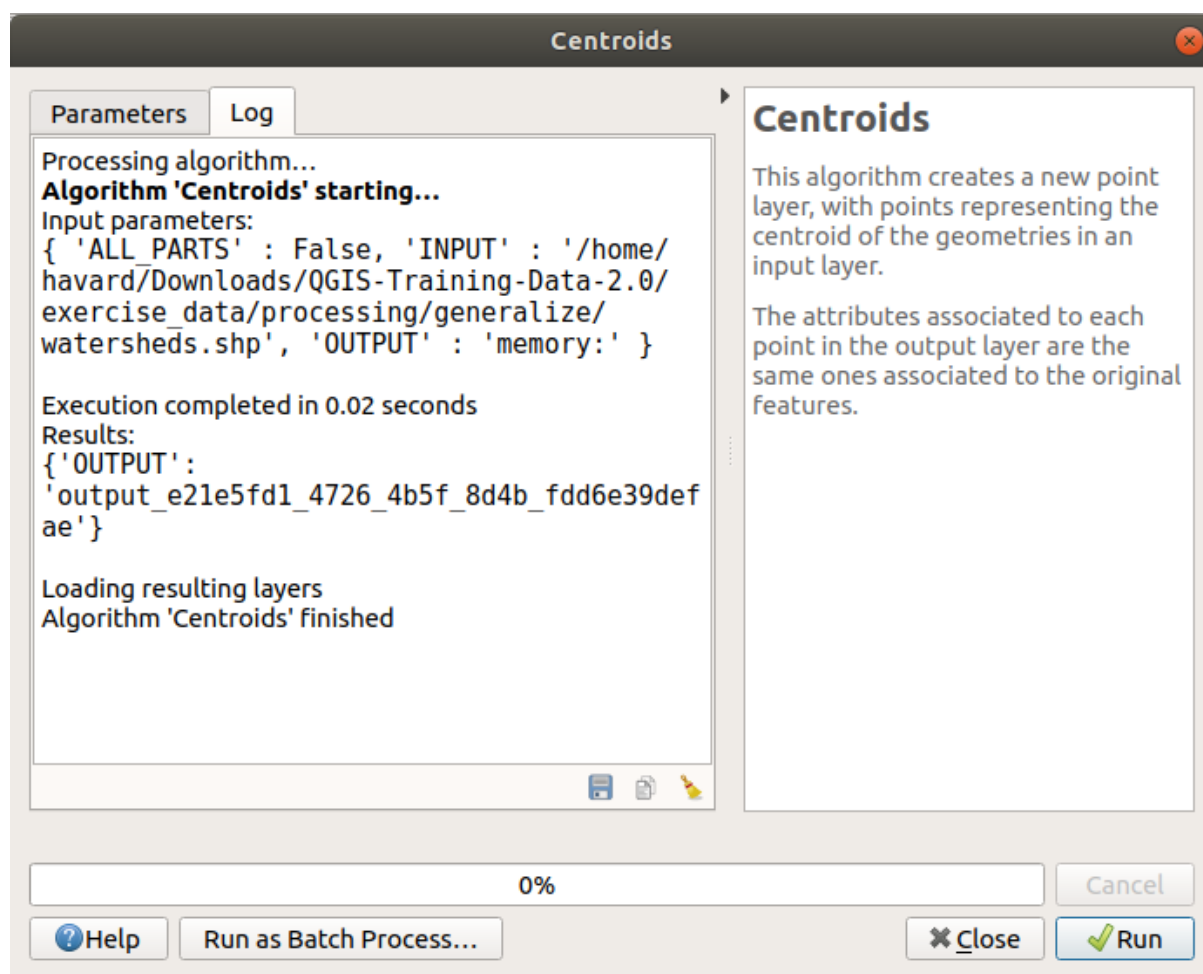


图 23.14: Algorithm Dialog - Log

At the bottom of the *Log* tab you will find buttons to  *Save Log to File*,  *Copy Log to Clipboard* and  *Clear Log*. These are particularly handy when you have checked the *Keep dialog open after running algorithm* in the *General* part of the Processing options.

On the right hand side of the dialog you will find a short description of the algorithm, which will help you understand its purpose and its basic ideas. If such a description is not available, the description panel will not be shown.

For a more detailed help file, which might include description of every parameter it uses, or examples, you will find a *Help* button at the bottom of the dialog bringing you to the *Processing algorithms documentation* or to the provider documentation (for some third-party providers).

The *Run as batch process* button triggers the *batch processing mode* allowing to configure and run multiple instances of the algorithm with a variety of parameters.

A note on projections

Processing algorithm execution are always performed in the input layer coordinate reference system (CRS). Due to QGIS's on-the-fly reprojecting capabilities, although two layers might seem to overlap and match, that might not be true if their original coordinates are used without reprojecting them onto a common coordinate system. Whenever you use more than one layer as input to a *QGIS native algorithm*, whether vector or raster, the layers will all be reprojected to match the coordinate reference system of the first input layer.

This is however less true for most of the external applications whose algorithms are exposed through the processing framework as they assume that all of the layers are already in a common coordinate system and ready to be analyzed.

By default, the parameters dialog will show a description of the CRS of each layer along with its name, making it easy to select layers that share the same CRS to be used as input layers. If you do not want to see this additional information, you can disable this functionality in the Processing settings dialog, unchecking the *Show layer CRS definition in selection boxes* option.

If you try to execute an algorithm using as input two or more layers with unmatching CRSs, a warning dialog will be shown. This occurs thanks to the *Warn before executing if layer CRS's do not match* option.

You still can execute the algorithm, but be aware that in most cases that will produce wrong results, such as empty layers due to input layers not overlapping.

小技巧: Use Processing algorithms to do intermediate reprojection

When an algorithm can not successfully perform on multiple input layers due to unmatching CRSs, use QGIS internal algorithm such as *Reproject layer* to perform layers' reprojection to the same CRS before executing the algorithm using these outputs.

23.3.2 Data objects generated by algorithms

Data objects generated by an algorithm can be of any of the following types:

- A raster layer
- A vector layer
- A table
- An HTML file (used for text and graphical outputs)

These are all saved to disk, and the parameters table will contain a text box corresponding to each one of these outputs, where you can type the output channel to use for saving it. An output channel contains the information needed to save the resulting object somewhere. In the most usual case, you will save it to a file, but in the case of vector layers, and when they are generated by native algorithms (algorithms not using external applications) you can also save to a PostGIS, GeoPackage or SpatiaLite database, or a memory layer.

To select an output channel, just click on the button on the right side of the text box, and you will see a small context menu with the available options.

In the most usual case, you will select saving to a file. If you select that option, you will be prompted with a save file dialog, where you can select the desired file path. Supported file extensions are shown in the file format selector of the dialog, depending on the kind of output and the algorithm.

The format of the output is defined by the filename extension. The supported formats depend on what is supported by the algorithm itself. To select a format, just select the corresponding file extension (or add it, if you are directly typing the file path instead). If the extension of the file path you entered does not match any of the supported formats, a default extension will be appended to the file path, and the file format corresponding to that extension will be used to

save the layer or table. Default extensions are `.dbf` for tables, `.tif` for raster layers and `.gpkg` for vector layers. These can be modified in the setting dialog, selecting any other of the formats supported by QGIS.

If you do not enter any filename in the output text box (or select the corresponding option in the context menu), the result will be saved as a *temporary file* in the corresponding default file format, and it will be deleted once you exit QGIS (take care with that, in case you save your project and it contains temporary layers).

You can set a default folder for output data objects. Go to the settings dialog (you can open it from the *Settings* ► *Options* ► *Processing* menu), and in the *General* group, you will find a parameter named *Output folder*. This output folder is used as the default path in case you type just a filename with no path (i.e., `myfile.shp`) when executing an algorithm.

When running an algorithm that uses a vector layer in iterative mode, the entered file path is used as the base path for all generated files, which are named using the base name and appending a number representing the index of the iteration. The file extension (and format) is used for all such generated files.

Apart from raster layers and tables, algorithms also generate graphics and text as HTML files. These results are shown at the end of the algorithm execution in a new dialog. This dialog will keep the results produced by any algorithm during the current session, and can be shown at any time by selecting *Processing* ► *Results Viewer* from the QGIS main menu.

Some external applications might have files (with no particular extension restrictions) as output, but they do not belong to any of the categories above. Those output files will not be processed by QGIS (opened or included into the current QGIS project), since most of the time they correspond to file formats or elements not supported by QGIS. This is, for instance, the case with LAS files used for LiDAR data. The files get created, but you won't see anything new in your QGIS working session.

For all the other types of output, you will find a checkbox that you can use to tell the algorithm whether to load the file once it is generated by the algorithm or not. By default, all files are opened.

Optional outputs are not supported. That is, all outputs are created. However, you can uncheck the corresponding checkbox if you are not interested in a given output, which essentially makes it behave like an optional output (in other words, the layer is created anyway, but if you leave the text box empty, it will be saved to a temporary file and deleted once you exit QGIS).

23.4 The history manager

23.4.1 The processing history

Every time you execute an algorithm, information about the process is stored in the history manager. The date and time of the execution are saved, along with the parameters used, making it is easy to track and control all the work that has been developed using the Processing framework, and to reproduce it.

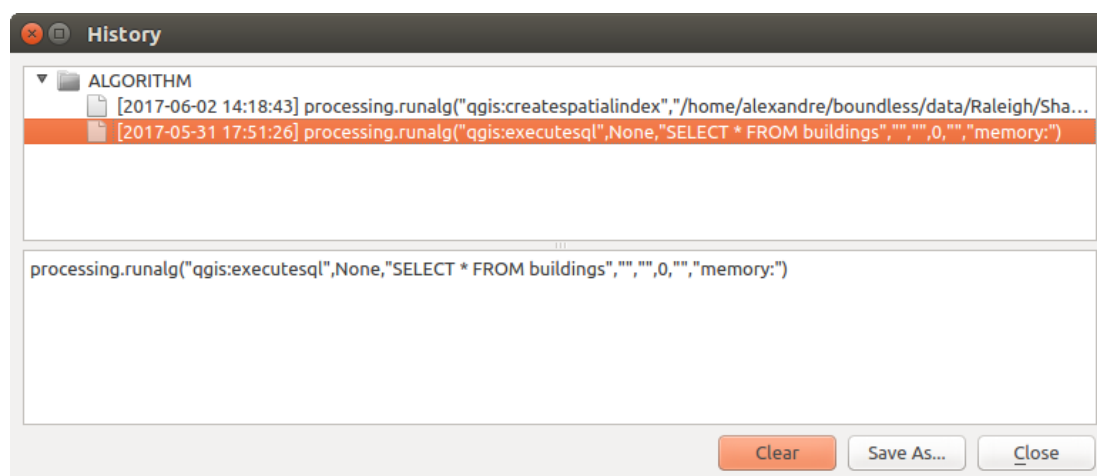


图 23.15: 历史

Process information is kept as a command-line expression, even if the algorithm was launched from the toolbox. This makes it useful for those learning how to use the command-line interface, since they can call an algorithm using the toolbox and then check the history manager to see how it could be called from the command line.

Apart from browsing the entries in the registry, you can also re-execute processes by simply double-clicking on the entry. The algorithm dialog then opens with parameters already set, and you can change any of them to fit your needs and re-run the algorithm.

The *History* dialog also provides a convenient way to contribute to the consolidation of the testing infrastructure of QGIS Processing algorithms and scripts. When you right-click on an entry, you can *Create Test...* using the concerned algorithm and parameters, following instructions at https://github.com/qgis/QGIS/blob/release-3_16/python/plugins/processing/tests/README.md.

23.4.2 The processing log

The history dialog only contains the execution calls, but not the information produced by the algorithm when executed. That information is written to the QGIS log (*View ► Panels ► Log Messages Panel*).

Third-party algorithms are usually executed by using their command-line interfaces, which communicate with the user via the console. Although that console is not shown, usually a full dump of it is written to the log each time you run one of those algorithms. To avoid cluttering the log with that information, you can disable it for each provider in the settings dialog.

Some algorithms, even if they can produce a result with the given input data, output comments or additional information to log when they detect potential problems with the data, in order to warn you. Make sure you check those messages in the log if you get unexpected results.

23.5 The graphical modeler

The *graphical modeler* allows you to create complex models using a simple and easy-to-use interface. When working with a GIS, most analysis operations are not isolated, rather part of a chain of operations. Using the graphical modeler, that chain of operations can be wrapped into a single process, making it convenient to execute later with a different set of inputs. No matter how many steps and different algorithms it involves, a model is executed as a single algorithm, saving time and effort.

The graphical modeler can be opened from the Processing menu (*Processing ► Graphical Modeler*).

The modeler has a working canvas where the structure of the model and the workflow it represents are shown. The left part of the window is a section with five panels that can be used to add new elements to the model:

1. **Model Properties:** you can specify the name of the model and the group that will contain it
2. **Inputs:** all the inputs that will shape your model
3. **Algorithms:** the Processing algorithms available
4. **Variables:** you can also define variables that will only be available in the Processing Modeler
5. **Undo History:** this panel will register everything that happens in the modeler, making it easy to cancel things you did wrong.

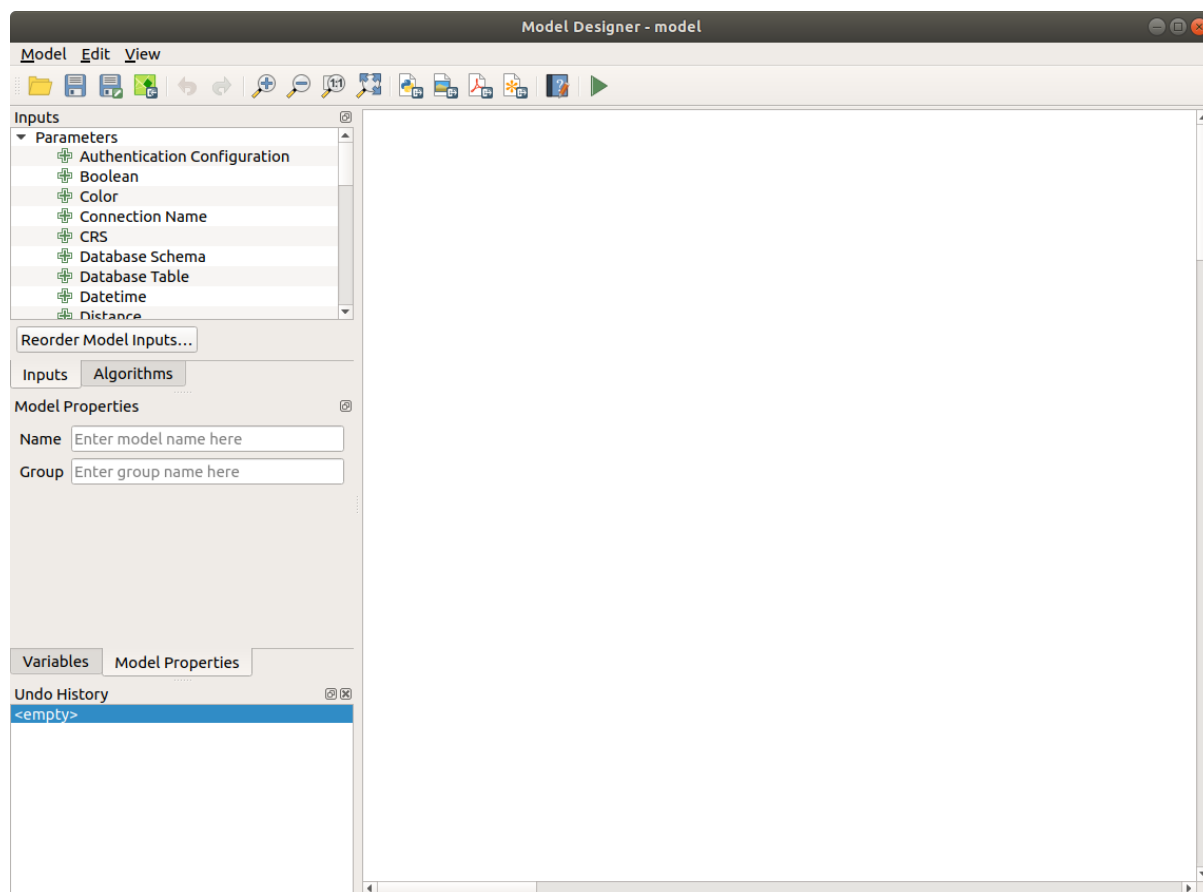


图 23.16: Modeler

Creating a model involves two basic steps:

1. *Definition of necessary inputs.* These inputs will be added to the parameters window, so the user can set their values when executing the model. The model itself is an algorithm, so the parameters window is generated automatically as for all algorithms available in the Processing framework.
2. *Definition of the workflow.* Using the input data of the model, the workflow is defined by adding algorithms and selecting how they use the defined inputs or the outputs generated by other algorithms in the model.

23.5.1 Definition of inputs

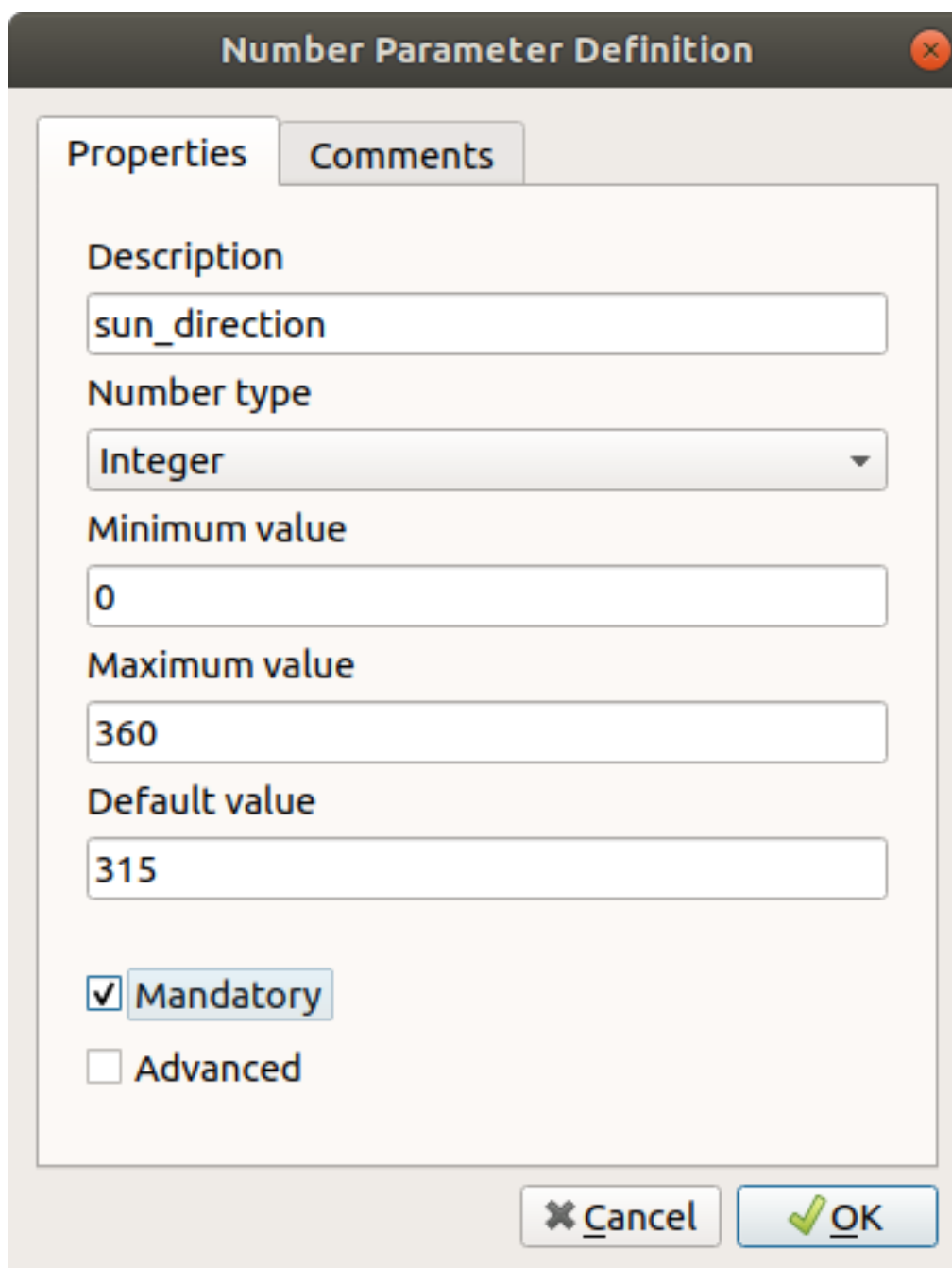
The first step is to define the inputs for the model. The following elements are found in the *Inputs* panel on the left side of the modeler window:

- Authentication Configuration
- Boolean
- Color
- Connection Name
- Coordinate Operation
- CRS
- Database Schema
- Database Table
- Datetime

- Distance
- Enum
- Expression
- Extent
- Field Aggregates
- Fields Mapper
- File/Folder
- Geometry
- Map Layer
- Map Theme
- Matrix
- Mesh Layer
- Multiple Input
- Number
- Point
- Print Layout
- Print Layout Item
- 范围（最大值和最小值之差）
- Raster Band
- Raster Layer
- Scale
- String
- TIN Creation Layers
- Vector Features
- Vector Field
- Vector Layer
- Vector Tile Writer Layers

注解: Hovering with the mouse over the inputs will show a tooltip with additional information.

When double-clicking on an element, a dialog is shown that lets you define its characteristics. Depending on the parameter, the dialog will contain at least one element (the description, which is what the user will see when executing the model). For example, when adding a numerical value, as can be seen in the next figure, in addition to the description of the parameter, you have to set a default value and the range of valid values.



The screenshot shows a dialog box titled "Number Parameter Definition" with a close button in the top right corner. It features two tabs: "Properties" and "Comments". The "Properties" tab is active and contains the following fields and options:

- Description:** A text input field containing "sun_direction".
- Number type:** A dropdown menu set to "Integer".
- Minimum value:** A text input field containing "0".
- Maximum value:** A text input field containing "360".
- Default value:** A text input field containing "315".
- Mandatory:** A checked checkbox.
- Advanced:** An unchecked checkbox.

At the bottom of the dialog are two buttons: "Cancel" and "OK".

图 23.17: Model Parameters Definition

You can define your input as mandatory for your model by checking the **Mandatory** option and by checking the **Advanced** checkbox you can set the input to be within the **Advanced** section. This is particularly useful when the model has many parameters and some of them are not trivial, but you still want to choose them.

The **Comments** tab allows you to tag the input with more information, to better describe the parameter. Comments are visible only in the modeler canvas and not in the final algorithm dialog.

For each added input, a new element is added to the modeler canvas.



图 23.18: Model Parameters

You can also add inputs by dragging the input type from the list and dropping it at the position where you want it in the modeler canvas. If you want to change a parameter of an existing input, just double click on it, and the same dialog will pop up.

23.5.2 Definition of the workflow

In the following example we will add two inputs and two algorithms. The aim of the model is to copy the elevation values from a DEM raster layer to a line layer using the `Drape` algorithm, and then calculate the total ascent of the line layer using the `Climb Along Line` algorithm.

In the *Inputs* tab, choose the two inputs as `Vector Layer` for the line and `Raster Layer` for the DEM. We are now ready to add the algorithms to the workflow.

Algorithms can be found in the *Algorithms* panel, grouped much in the same way as they are in the Processing toolbox.

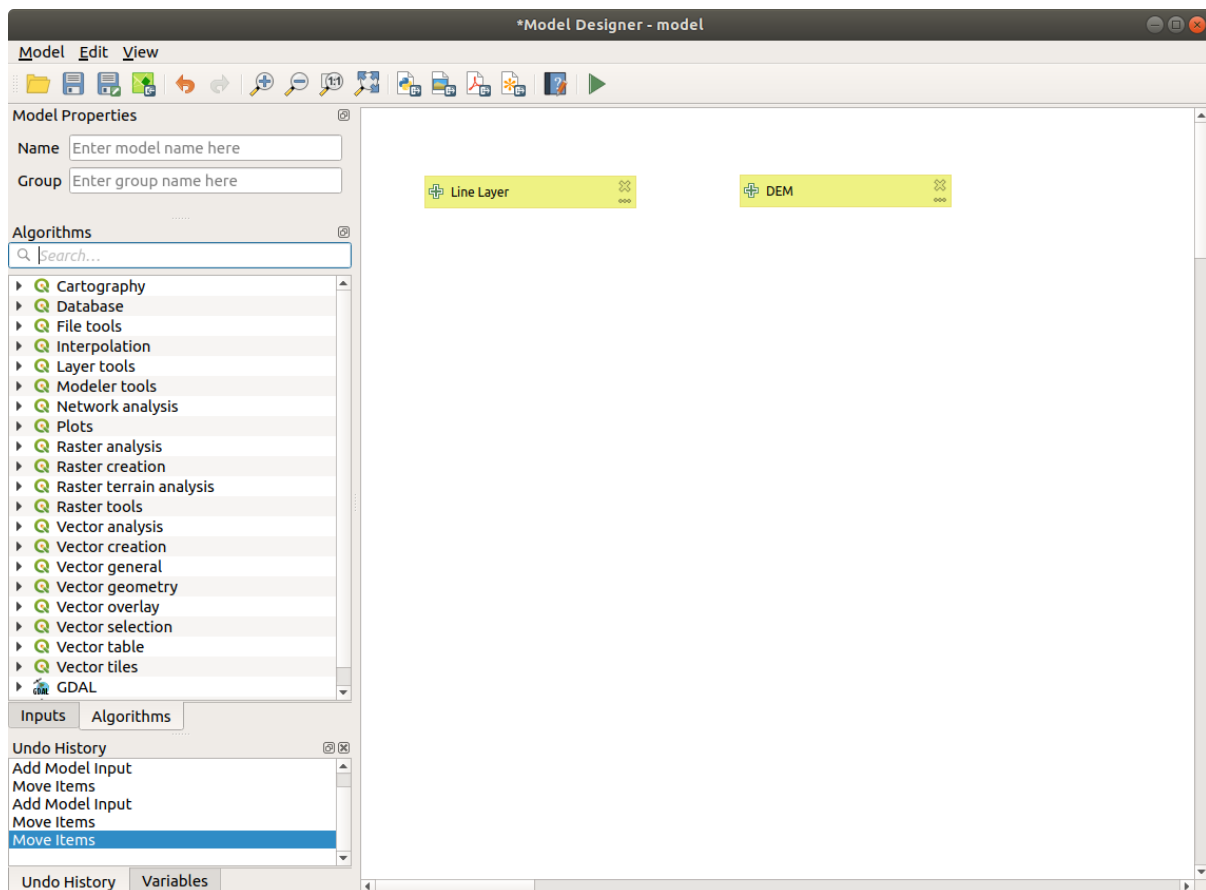


图 23.19: Model Inputs

To add an algorithm to a model, double-click on its name or drag and drop it, just like for inputs. As for the inputs you can change the description of the algorithm and add a comment. When adding an algorithm, an execution dialog will appear, with a content similar to the one found in the execution panel that is shown when executing the algorithm from the toolbox. The following picture shows both the `Drape` (set Z value from raster) and the `Climb along line` algorithm dialogs.

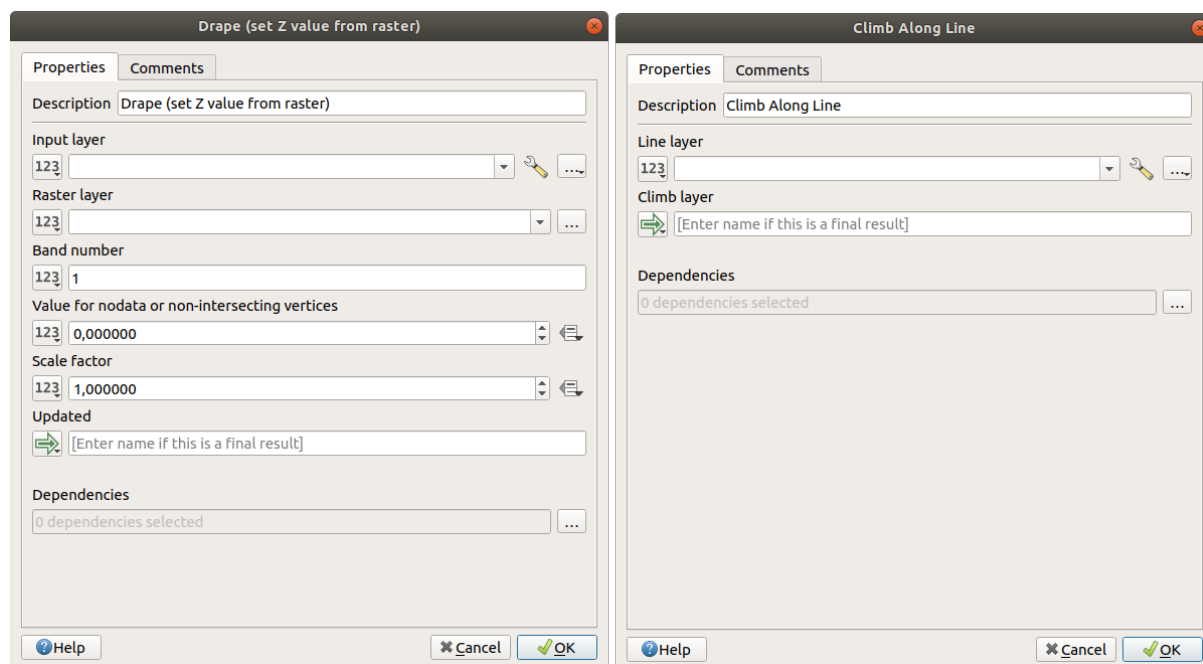







图 23.20: Model Algorithm parameters

As you can see there are some differences.

You have four choices to define the algorithm **inputs**:

-  Value: allows you to set the parameter from a loaded layer in the QGIS project or to browse a layer from a folder
-  Pre-calculated Value: with this option you can open the Expression Builder and define your own expression to fill the parameter. Model inputs together with some other layer statistics are available as **variables** and are listed at the top of the Search dialog of the Expression Builder
-  Model Input: choose this option if the parameter comes from an input of the model you have defined. Once clicked, this option will list all the suitable inputs for the parameter
-  Algorithm Output: is useful when the input parameter of an algorithm is an output of another algorithm

Algorithm **outputs** have the additional  Model Output option that makes the output of the algorithm available in the model.

If a layer generated by the algorithm is only to be used as input to another algorithm, don't edit that text box.

In the following picture you can see the two input parameters defined as Model Input and the temporary output layer:

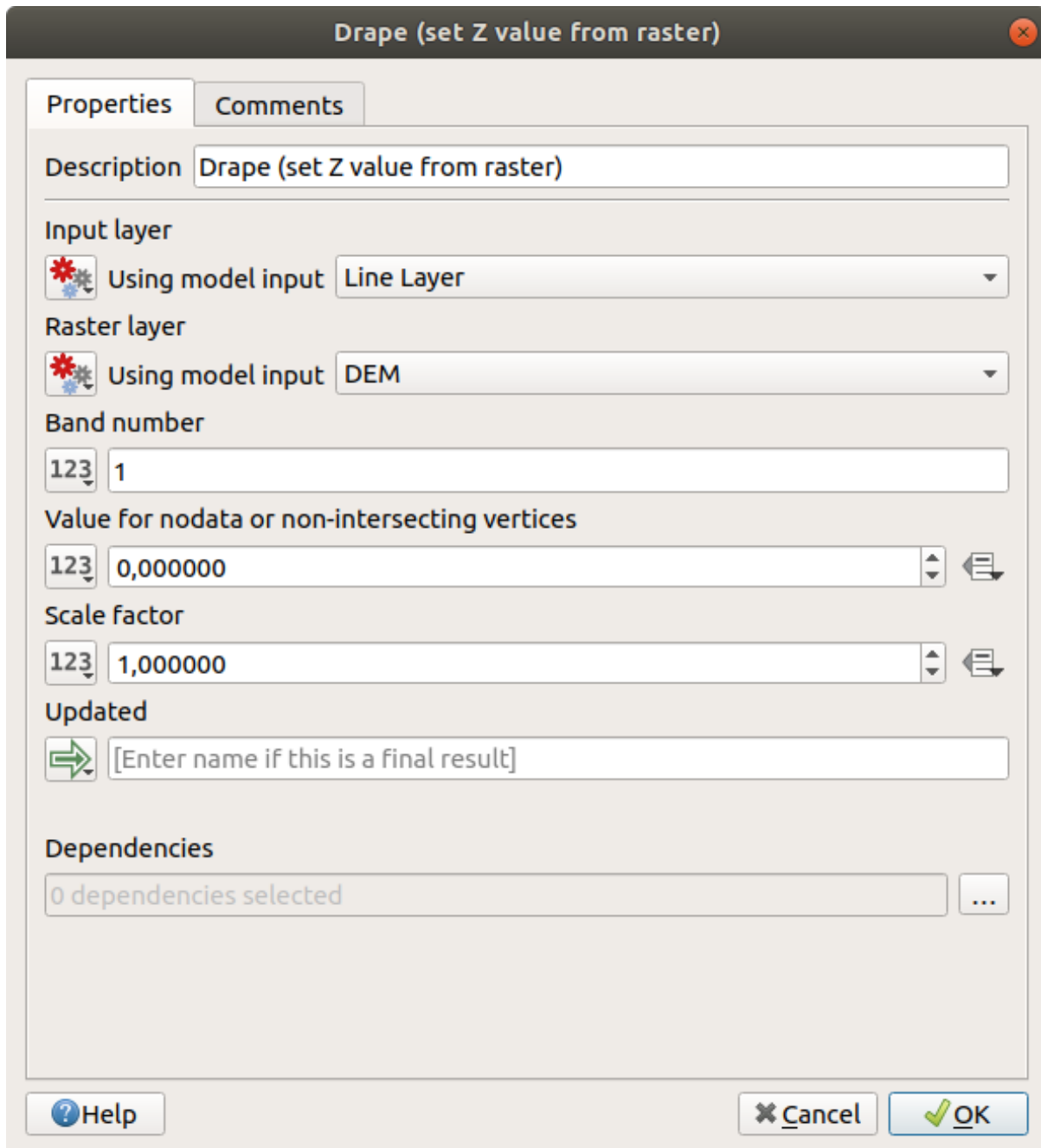


图 23.21: Algorithm Input and Output parameters

In all cases, you will find an additional parameter named *Dependencies* that is not available when calling the algorithm from the toolbox. This parameter allows you to define the order in which algorithms are executed, by explicitly defining one algorithm as a *parent* of the current one. This will force the *parent* algorithm to be executed before the current one.

When you use the output of a previous algorithm as the input of your algorithm, that implicitly sets the previous algorithm as parent of the current one (and places the corresponding arrow in the modeler canvas). However, in some cases an algorithm might depend on another one even if it does not use any output object from it (for instance, an algorithm that executes a SQL sentence on a PostGIS database and another one that imports a layer into that same database). In that case, just select the previous algorithm in the *Dependencies* parameter and they will be executed in the correct order.

Once all the parameters have been assigned valid values, click on *OK* and the algorithm will be added to the canvas. It will be linked to the elements in the canvas (algorithms or inputs) that provide objects that are used as inputs for

the algorithm.

Elements can be dragged to a different position on the canvas. This is useful to make the structure of the model more clear and intuitive. You can also resize elements. This is particularly useful if the description of the input or algorithm is long.

Links between elements are updated automatically and you can see a plus button at the top and at the bottom of each algorithm. Clicking the button will list all the inputs and outputs of the algorithm so you can have a quick overview.

You can zoom in and out by using the mouse wheel.

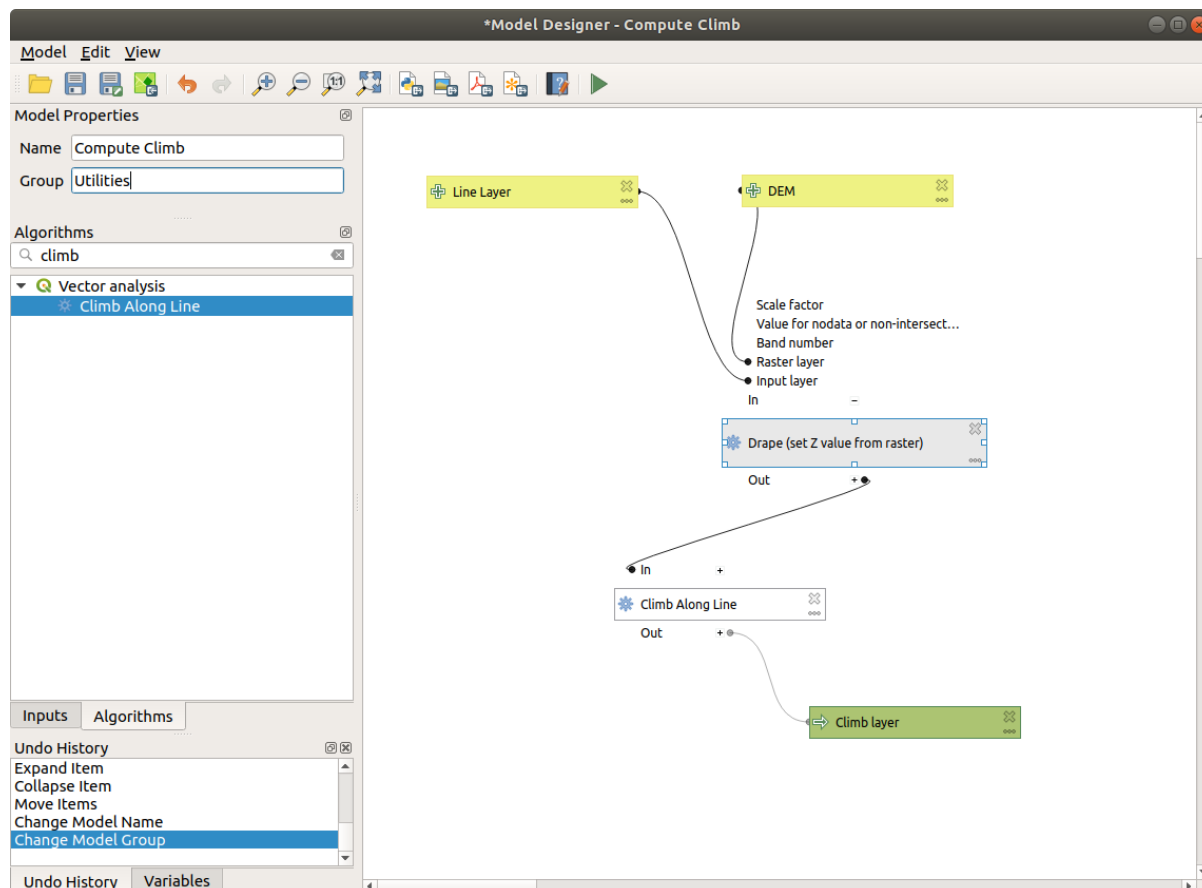







图 23.22: A complete model

You can run your algorithm any time by clicking on the  button. In order to use the algorithm from the toolbox, it has to be saved and the modeler dialog closed, to allow the toolbox to refresh its contents.

23.5.3 Interacting with the canvas and elements








You can use the , ,  and  buttons to zoom the modeler canvas. The behavior of the buttons is basically the same of the main QGIS toolbar.

The `Undo History` panel together with the  and  buttons are extremely useful to quickly rollback to a previous situation. The `Undo History` panel lists everything you have done when creating the workflow.

You can move or resize many elements at the same time by first selecting them, dragging the mouse.

If you want to snap the elements while moving them in the canvas you can choose `View ► Enable Snapping`.

The `Edit` menu contains some very useful options to interact with your model elements:

-  Select All: select all elements of the model
- Snap Selected Components to Grid: snap and align the elements into a grid
-  Undo: undo the last action
-  Redo: redo the last action
-  Cut: cut the selected elements
-  Copy: copy the selected elements
-  Paste: paste the elements
-  Delete Selected Components: delete all the selected elements from the model
- Add Group Box: add a draggable *box* to the canvas. This feature is very useful in big models to group elements in the modeler canvas and to keep the workflow clean. For example we might group together all the inputs of the example:

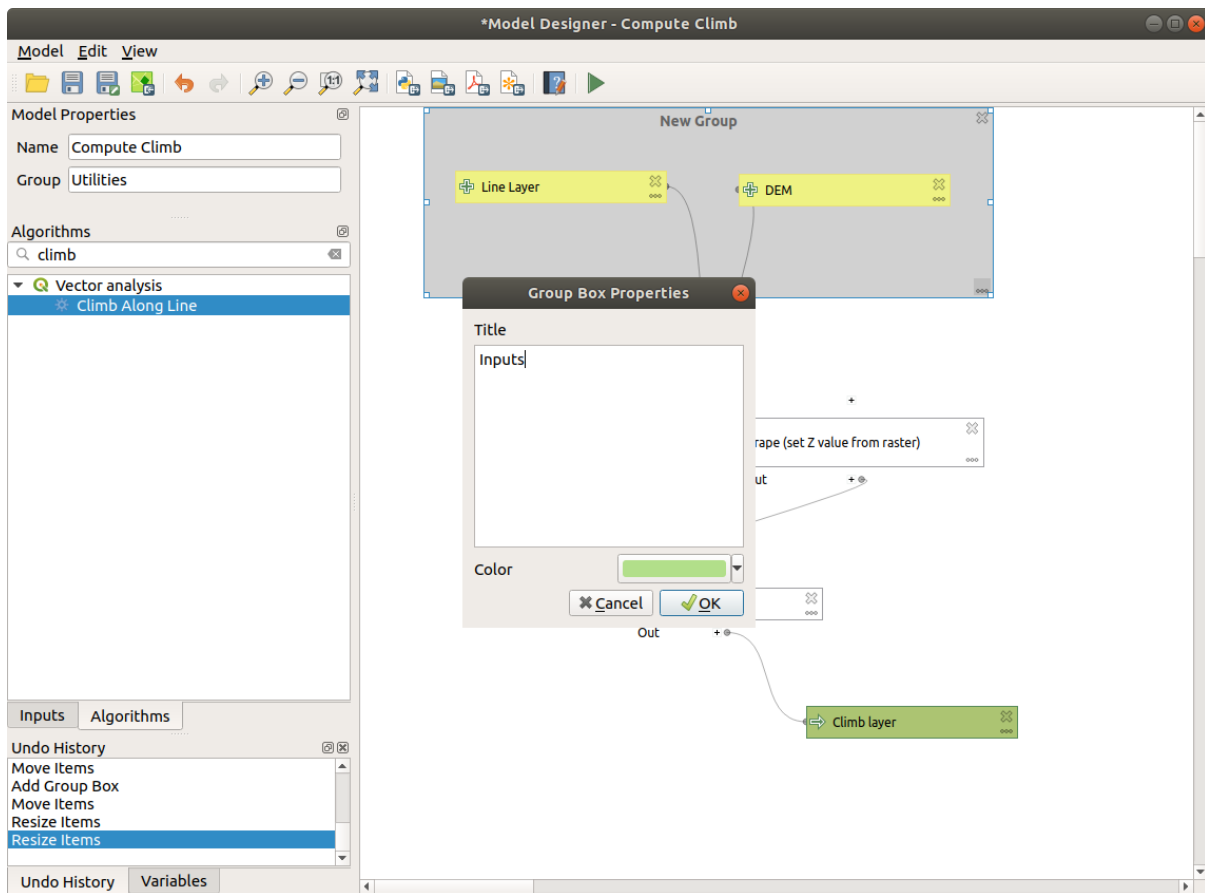


图 23.23: Model Group Box

You can change the name and the color of the boxes. Group boxes are very useful when used together with *View ► Zoom To*. This allows you to zoom to a specific part of the model.

You might want to change the order of the inputs and how they are listed in the main model dialog. At the bottom of the Input panel you will find the *Reorder Model Inputs...* button and by clicking on it a new dialog pops up allowing you to change the order of the inputs:

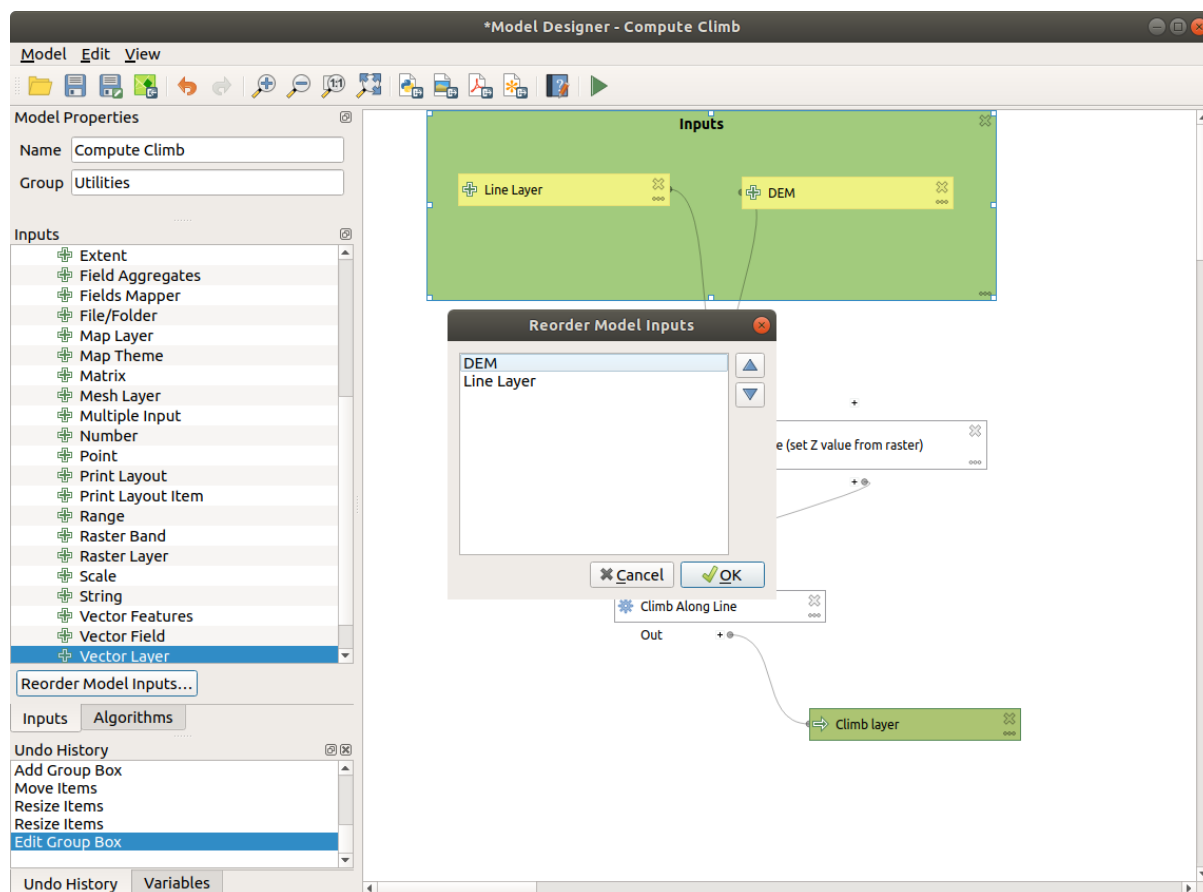




图 23.24: Reorder Model Inputs

23.5.4 Saving and loading models

Use the  Save model button to save the current model and the  Open Model button to open a previously saved model. Models are saved with the `.model3` extension. If the model has already been saved from the modeler window, you will not be prompted for a filename. Since there is already a file associated with the model, that file will be used for subsequent saves.

Before saving a model, you have to enter a name and a group for it in the text boxes in the upper part of the window.

Models saved in the `models` folder (the default folder when you are prompted for a filename to save the model) will appear in the toolbox in the corresponding branch. When the toolbox is invoked, it searches the `models` folder for files with the `.model3` extension and loads the models they contain. Since a model is itself an algorithm, it can be added to the toolbox just like any other algorithm.

Models can also be saved within the project file using the  Save model in project button. Models saved using this method won't be written as `.model3` files on the disk but will be embedded in the project file.




Project models are available in the  Project models menu of the toolbox.

The models folder can be set from the Processing configuration dialog, under the *Modeler* group.

Models loaded from the `models` folder appear not only in the toolbox, but also in the algorithms tree in the *Algorithms* tab of the modeler window. That means that you can incorporate a model as a part of a bigger model, just like other algorithms.

Models will show up in the *Browser* panel and can be run from there.

Exporting a model as an image, PDF or SVG

A model can also be exported as an image, SVG or PDF (for illustration purposes) by clicking  Export as image,  Export as PDF or  Export as SVG.

23.5.5 Editing a model

You can edit the model you are currently creating, redefining the workflow and the relationships between the algorithms and inputs that define the model.

If you right-click on an algorithm in the canvas, you will see a context menu like the one shown next:

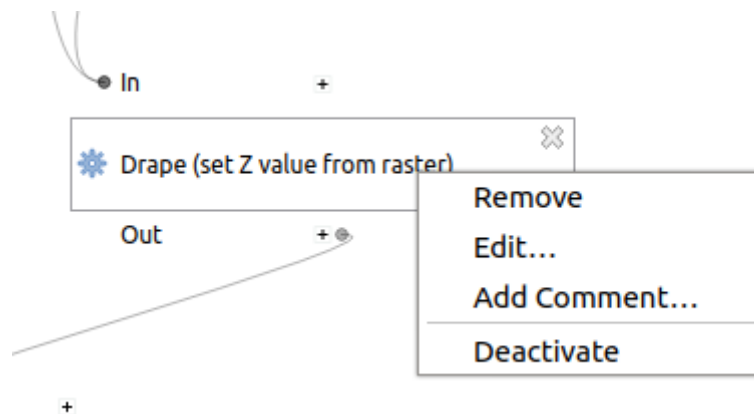


图 23.25: Modeler Right Click

Selecting the *Remove* option will cause the selected algorithm to be removed. An algorithm can be removed only if there are no other algorithms depending on it. That is, if no output from the algorithm is used in a different one as input. If you try to remove an algorithm that has others depending on it, a warning message like the one you can see below will be shown:

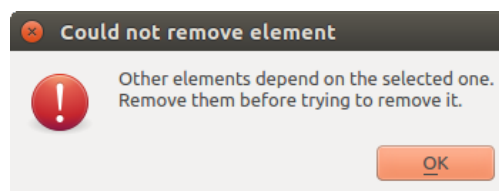


图 23.26: Cannot Delete Algorithm

Selecting the *Edit...* option will show the parameter dialog of the algorithm, so you can change the inputs and parameter values. Not all input elements available in the model will appear as available inputs. Layers or values generated at a more advanced step in the workflow defined by the model will not be available if they cause circular dependencies.

Select the new values and click on the *OK* button as usual. The connections between the model elements will change in the modeler canvas accordingly.

The *Add comment...* allows you to add a comment to the algorithm to better describe the behavior.

A model can be run partially by deactivating some of its algorithms. To do it, select the *Deactivate* option in the context menu that appears when right-clicking on an algorithm element. The selected algorithm, and all the ones in the model that depend on it will be displayed in grey and will not be executed as part of the model.

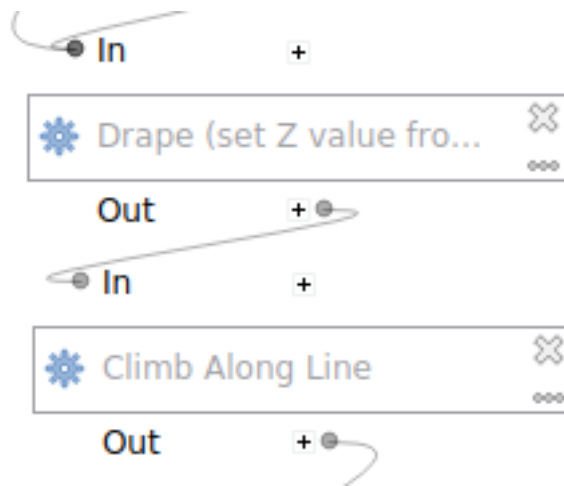



图 23.27: Model With Deactivated Algorithms

When right-clicking on an algorithm that is not active, you will see a *Activate* menu option that you can use to reactivate it.

23.5.6 Editing model help files and meta-information

You can document your models from the modeler itself. Click on the  Edit model help button, and a dialog like the one shown next will appear.

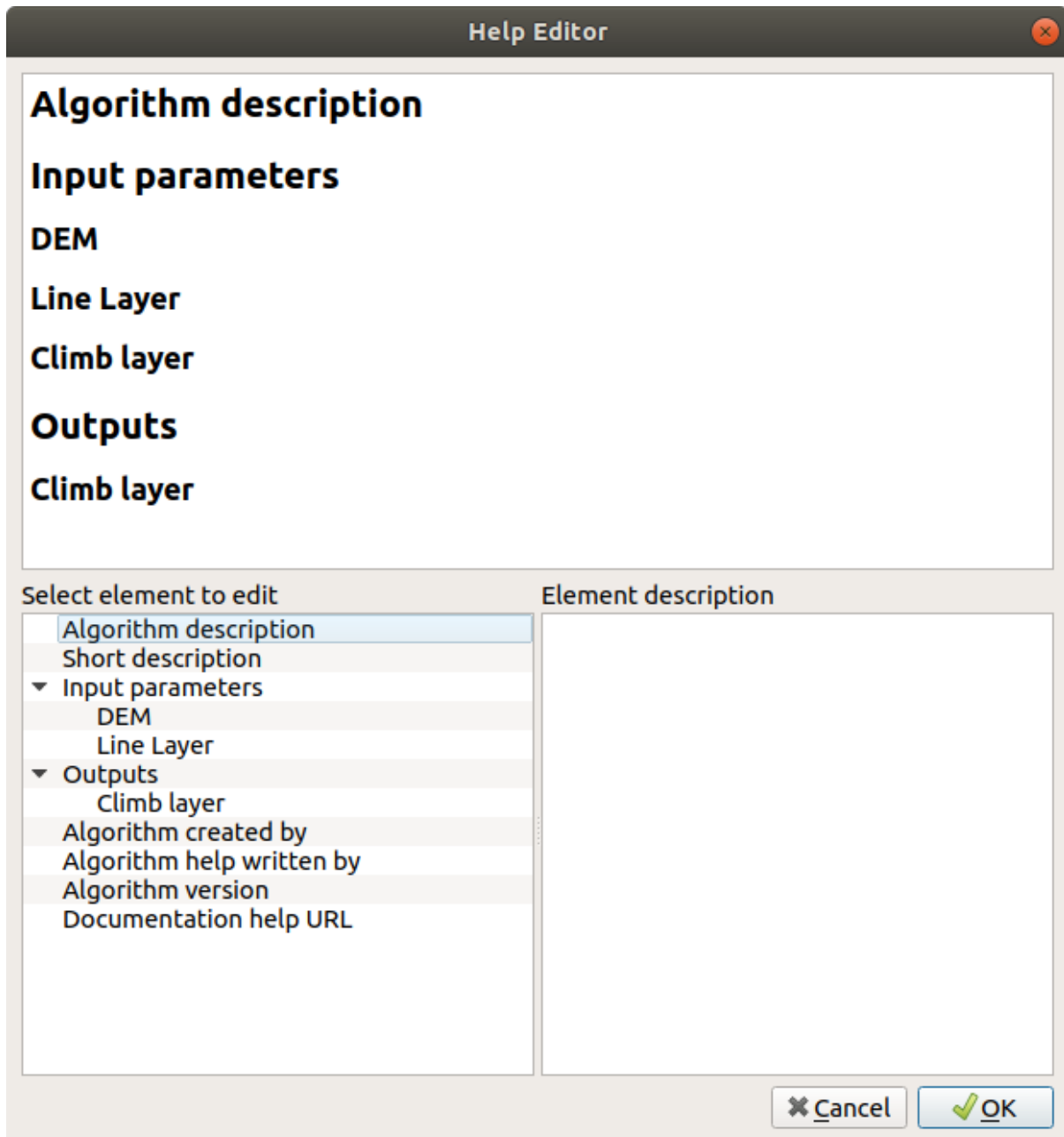




图 23.28: Editing Help

On the right-hand side, you will see a simple HTML page, created using the description of the input parameters and outputs of the algorithm, along with some additional items like a general description of the model or its author. The first time you open the help editor, all these descriptions are empty, but you can edit them using the elements on the left-hand side of the dialog. Select an element on the upper part and then write its description in the text box below.

Model help is saved as part of the model itself.

23.5.7 Exporting a model as a Python script

As we will see in a later chapter, Processing algorithms can be called from the QGIS Python console, and new Processing algorithms can be created using Python. A quick way to create such a Python script is to create a model and then export it as a Python file.

To do so, click on the  Export as Script Algorithm... in the modeler canvas or right click on the name of the model in the Processing Toolbox and choose  Export Model as Python Algorithm...

23.5.8 About available algorithms

You might notice that some algorithms that can be executed from the toolbox do not appear in the list of available algorithms when you are designing a model. To be included in a model, an algorithm must have the correct semantic. If an algorithm does not have such a well-defined semantic (for instance, if the number of output layers cannot be known in advance), then it is not possible to use it within a model, and it will not appear in the list of algorithms that you can find in the modeler dialog.

23.6 The batch processing interface

23.6.1 简介

All algorithms (including models) can be executed as a batch process. That is, they can be executed using not just a single set of inputs, but several of them, executing the algorithm as many times as needed. This is useful when processing large amounts of data, since it is not necessary to launch the algorithm many times from the toolbox.

To execute an algorithm as a batch process, right-click on its name in the toolbox and select the *Execute as batch process* option in the pop-up menu that will appear.

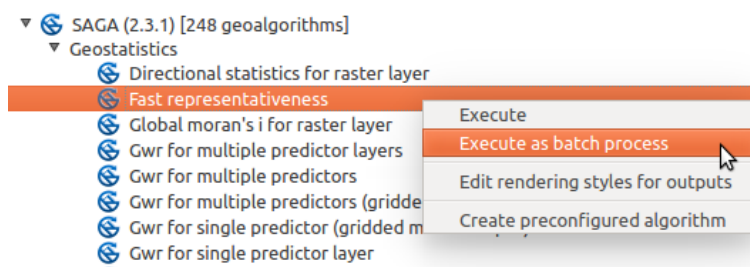


图 23.29: Batch Processing from right-click

If you have the execution dialog of the algorithm open, you can also start the batch processing interface from there, clicking on the *Run as batch process...* button.

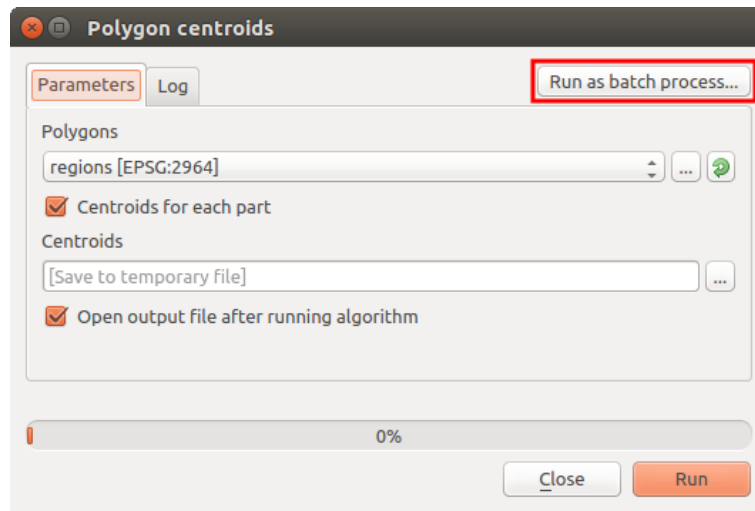


图 23.30: Batch Processing From Algorithm Dialog

23.6.2 The parameters table

Executing a batch process is similar to performing a single execution of an algorithm. Parameter values have to be defined, but in this case we need not just a single value for each parameter, but a set of them instead, one for each time the algorithm has to be executed. Values are introduced using a table like the one shown next.

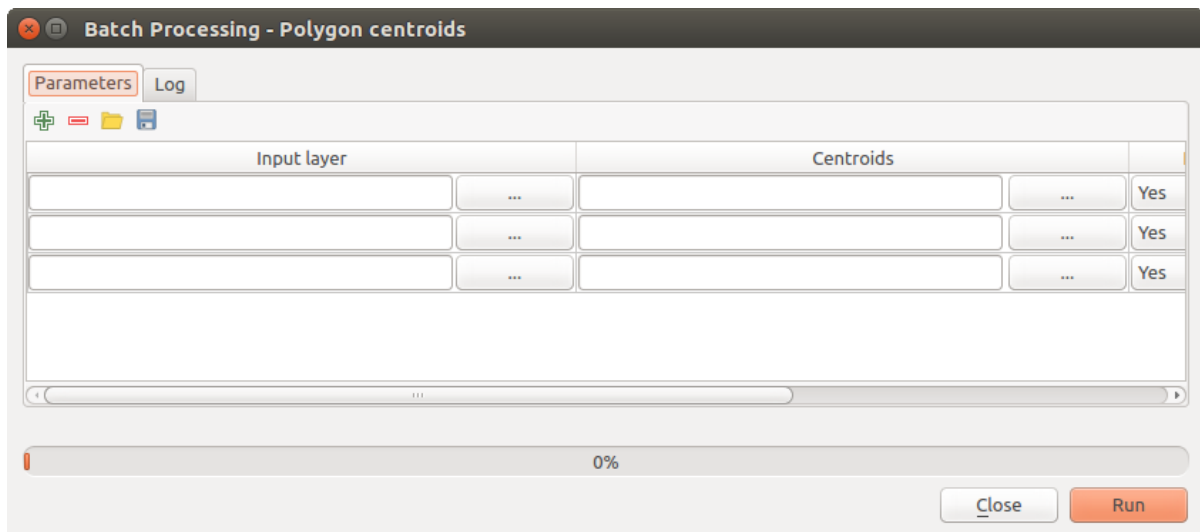


图 23.31: Batch Processing

Each line of this table represents a single execution of the algorithm, and each cell contains the value of one of the parameters. It is similar to the parameters dialog that you see when executing an algorithm from the toolbox, but with a different arrangement.

By default, the table contains just two rows. You can add or remove rows using the buttons on the lower part of the window.

Once the size of the table has been set, it has to be filled with the desired values.

23.6.3 Filling the parameters table

For most parameters, setting the value is trivial. Just type the value or select it from the list of available options, depending on the parameter type.

Filenames for input data objects are introduced directly typing or, more conveniently, clicking on the ... button on the right hand of the cell, which will show a context menu with two options: one for selecting from the layers currently opened and another to select from the filesystem. This second option, when selected, shows a typical file chooser dialog. Multiple files can be selected at once. If the input parameter represents a single data object and several files are selected, each one of them will be put in a separate row, adding new ones if needed. If the parameter represents a multiple input, all the selected files will be added to a single cell, separated by semicolons (;).

Layer identifiers can be directly typed in the parameter text box. You can enter the full path to a file or the name of a layer that is currently loaded in the current QGIS project. The name of the layer will be automatically resolved to its source path. Notice that, if several layers have the same name, this might cause unexpected results due to ambiguity.

Output data objects are always saved to a file and, unlike when executing an algorithm from the toolbox, saving to a temporary file or database is not permitted. You can type the name directly or use the file chooser dialog that appears when clicking on the accompanying button.

Once you select the file, a new dialog is shown to allow for autocompletion of other cells in the same column (same parameter).

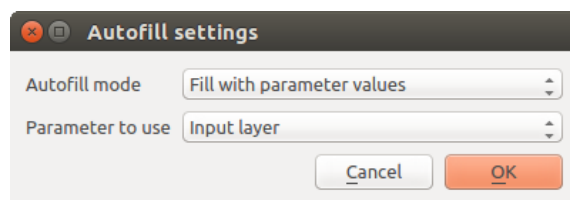


图 23.32: Batch Processing Save

If the default value ('Do not autocomplete') is selected, it will just put the selected filename in the selected cell from the parameters table. If any of the other options is selected, all the cells below the selected one will be automatically filled based on a defined criteria. This way, it is much easier to fill the table, and the batch process can be defined with less effort.

Automatic filling can be done by simply adding correlative numbers to the selected file path, or by appending the value of another field at the same row. This is particularly useful for naming output data objects according to input ones.

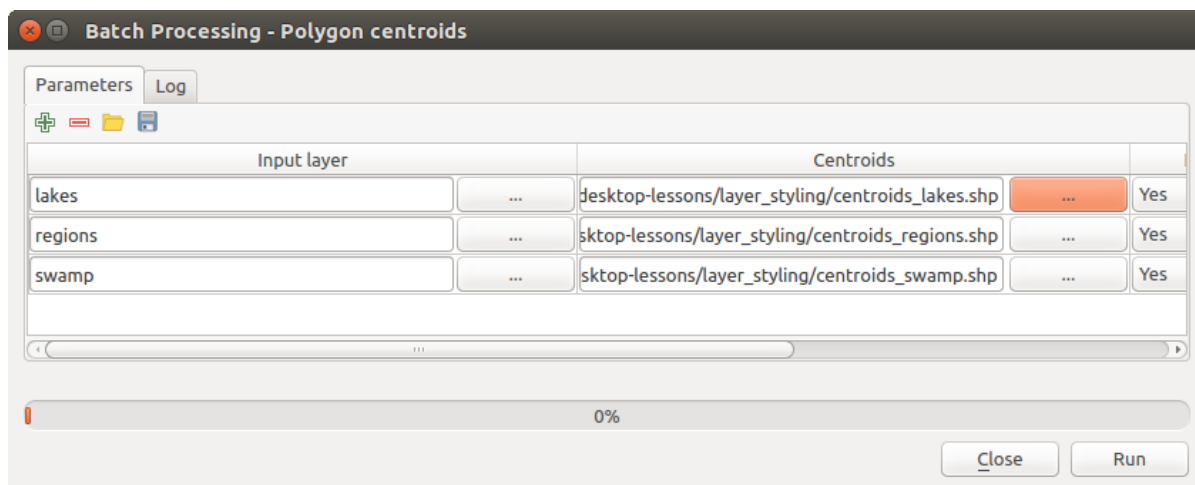


图 23.33: Batch Processing File Path

23.6.4 Executing the batch process

To execute the batch process once you have introduced all the necessary values, just click on *OK*. Progress of the global batch task will be shown in the progress bar in the lower part of the dialog.

23.7 Using processing algorithms from the console

The console allows advanced users to increase their productivity and perform complex operations that cannot be performed using any of the other GUI elements of the processing framework. Models involving several algorithms can be defined using the command-line interface, and additional operations such as loops and conditional sentences can be added to create more flexible and powerful workflows.

There is not a processing console in QGIS, but all processing commands are available instead from the QGIS built-in *Python console*. That means that you can incorporate those commands into your console work and connect processing algorithms to all the other features (including methods from the QGIS API) available from there.

The code that you can execute from the Python console, even if it does not call any specific processing method, can be converted into a new algorithm that you can later call from the toolbox, the graphical modeler or any other component, just like you do with any other algorithm. In fact, some algorithms that you can find in the toolbox are simple scripts.

In this section, we will see how to use processing algorithms from the QGIS Python console, and also how to write algorithms using Python.

23.7.1 Calling algorithms from the Python console

The first thing you have to do is to import the processing functions with the following line:

```
>>> from qgis import processing
```

Now, there is basically just one (interesting) thing you can do with that from the console: execute an algorithm. That is done using the `run()` method, which takes the name of the algorithm to execute as its first parameter, and then a variable number of additional parameters depending on the requirements of the algorithm. So the first thing you need to know is the name of the algorithm to execute. That is not the name you see in the toolbox, but rather a unique command-line name. To find the right name for your algorithm, you can use the `processingRegistry`. Type the following line in your console:

```
>>> for alg in QgsApplication.processingRegistry().algorithms():
    print(alg.id(), "->", alg.displayName())
```

You will see something like this (with some extra dashes added to improve readability).

```
3d:tessellate -----> Tessellate
gdal:aspect -----> Aspect
gdal:assignprojection -----> Assign projection
gdal:bufferectors -----> Buffer vectors
gdal:buildvirtualraster ----> Build Virtual Raster
gdal:cliprasterbyextent ----> Clip raster by extent
gdal:cliprasterbymasklayer -> Clip raster by mask layer
gdal:clipvectorbyextent ----> Clip vector by extent
gdal:clipvectorbypolygon ---> Clip vector by mask layer
gdal:colorrelief -----> Color relief
gdal:contour -----> Contour
gdal:convertformat -----> Convert format
gdal:dissolve -----> Dissolve
...
```

That's a list of all the available algorithm IDs, sorted by provider name and algorithm name, along with their corresponding names.

Once you know the command-line name of the algorithm, the next thing to do is to determine the right syntax to execute it. That means knowing which parameters are needed when calling the `run()` method.

There is a method to describe an algorithm in detail, which can be used to get a list of the parameters that an algorithm requires and the outputs that it will generate. To get this information, you can use the `algorithmHelp(id_of_the_algorithm)` method. Use the ID of the algorithm, not the full descriptive name.

Calling the method with `native:buffer` as parameter (`qgis:buffer` is an alias for `native:buffer` and will also work), you get the following description:

```
>>> processing.algorithmHelp("native:buffer")
Buffer (native:buffer)

This algorithm computes a buffer area for all the features in an
input layer, using a fixed or dynamic distance.

The segments parameter controls the number of line segments to
use to approximate a quarter circle when creating rounded
offsets.

The end cap style parameter controls how line endings are handled
in the buffer.

The join style parameter specifies whether round, miter or
beveled joins should be used when offsetting corners in a line.

The miter limit parameter is only applicable for miter join
styles, and controls the maximum distance from the offset curve
to use when creating a mitered join.

-----
Input parameters
-----

INPUT: Input layer

    Parameter type: QgsProcessingParameterFeatureSource

    Accepted data types:
      - str: layer ID
      - str: layer name
      - str: layer source
      - QgsProcessingFeatureSourceDefinition
      - QgsProperty
      - QgsVectorLayer

DISTANCE: Distance

    Parameter type: QgsProcessingParameterDistance

    Accepted data types:
      - int
      - float
      - QgsProperty

SEGMENTS: Segments

    Parameter type: QgsProcessingParameterNumber

    Accepted data types:
      - int
```

(下页继续)

(续上页)

```

    - float
    - QgsProperty

END_CAP_STYLE: End cap style

Parameter type: QgsProcessingParameterEnum

Available values:
    - 0: Round
    - 1: Flat
    - 2: Square

Accepted data types:
    - int
    - str: as string representation of int, e.g. '1'
    - QgsProperty

JOIN_STYLE: Join style

Parameter type: QgsProcessingParameterEnum

Available values:
    - 0: Round
    - 1: Miter
    - 2: Bevel

Accepted data types:
    - int
    - str: as string representation of int, e.g. '1'
    - QgsProperty

MITER_LIMIT: Miter limit

Parameter type: QgsProcessingParameterNumber

Accepted data types:
    - int
    - float
    - QgsProperty

DISSOLVE: Dissolve result

Parameter type: QgsProcessingParameterBoolean

Accepted data types:
    - bool
    - int
    - str
    - QgsProperty

OUTPUT: Buffered

Parameter type: QgsProcessingParameterFeatureSink

Accepted data types:
    - str: destination vector file, e.g. 'd:/test.shp'
    - str: 'memory:' to store result in temporary memory layer
    - str: using vector provider ID prefix and destination URI,
          e.g. 'postgres:...' to store result in PostGIS table
    - QgsProcessingOutputLayerDefinition
    - QgsProperty

```

(下页继续)

```
-----
Outputs
-----

OUTPUT: <QgsProcessingOutputVectorLayer>
        Buffered
```

Now you have everything you need to run any algorithm. As we have already mentioned, algorithms can be run using: `run()`. Its syntax is as follows:

```
>>> processing.run(name_of_the_algorithm, parameters)
```

Where `parameters` is a dictionary of parameters that depend on the algorithm you want to run, and is exactly the list that the `algorithmHelp()` method gives you.

```
1 >>> processing.run("native:buffer", {'INPUT': '/data/lines.shp',
2     'DISTANCE': 100.0,
3     'SEGMENTS': 10,
4     'DISSOLVE': True,
5     'END_CAP_STYLE': 0,
6     'JOIN_STYLE': 0,
7     'MITER_LIMIT': 10,
8     'OUTPUT': '/data/buffers.shp'})
```

If a parameter is optional and you do not want to use it, then don't include it in the dictionary.

If a parameter is not specified, the default value will be used.

Depending on the type of parameter, values are introduced differently. The next list gives a quick review of how to introduce values for each type of input parameter:

- Raster Layer, Vector Layer or Table. Simply use a string with the name that identifies the data object to use (the name it has in the QGIS Table of Contents) or a filename (if the corresponding layer is not opened, it will be opened but not added to the map canvas). If you have an instance of a QGIS object representing the layer, you can also pass it as parameter.
- Enumeration. If an algorithm has an enumeration parameter, the value of that parameter should be entered using an integer value. To know the available options, you can use the `algorithmHelp()` command, as above. For instance, the `native:buffer` algorithm has an enumeration called `JOIN_STYLE`:

```
JOIN_STYLE: Join style

Parameter type: QgsProcessingParameterEnum

Available values:
- 0: Round
- 1: Miter
- 2: Bevel

Accepted data types:
- int
- str: as string representation of int, e.g. '1'
- QgsProperty
```

In this case, the parameter has three options. Notice that ordering is zero-based.

- Boolean. Use `True` or `False`.
- Multiple input. The value is a string with input descriptors separated by semicolons (;). As in the case of single layers or tables, each input descriptor can be the data object name, or its file path.
- Table Field from XXX. Use a string with the name of the field to use. This parameter is case-sensitive.

- Fixed Table. Type the list of all table values separated by commas (,) and enclosed between quotes ("). Values start on the upper row and go from left to right. You can also use a 2-D array of values representing the table.
- CRS. Enter the EPSG code number of the desired CRS.
- Extent. You must use a string with `xmin`, `xmax`, `ymin` and `ymax` values separated by commas (,).

Boolean, file, string and numerical parameters do not need any additional explanations.

Input parameters such as strings, booleans, or numerical values have default values. The default value is used if the corresponding parameter entry is missing.

For output data objects, type the file path to be used to save it, just as it is done from the toolbox. If the output object is not specified, the result is saved to a temporary file (or skipped if it is an optional output). The extension of the file determines the file format. If you enter a file extension not supported by the algorithm, the default file format for that output type will be used, and its corresponding extension appended to the given file path.

Unlike when an algorithm is executed from the toolbox, outputs are not added to the map canvas if you execute that same algorithm from the Python console using `run()`, but `runAndLoadResults()` will do that.

The `run()` method returns a dictionary with one or more output names (the ones shown in the algorithm description) as keys and the file paths of those outputs as values:

```

1  >>> myresult = processing.run("native:buffer", {'INPUT': '/data/lines.shp',
2          'DISTANCE': 100.0,
3          'SEGMENTS': 10,
4          'DISSOLVE': True,
5          'END_CAP_STYLE': 0,
6          'JOIN_STYLE': 0,
7          'MITER_LIMIT': 10,
8          'OUTPUT': '/data/buffers.shp'})
9  >>> myresult['OUTPUT']
10 /data/buffers.shp

```

You can load feature output by passing the corresponding file paths to the `load()` method. Or you could use `runAndLoadResults()` instead of `run()` to load them immediately.

If you want to open an algorithm dialog from the console you can use the `createAlgorithmDialog` method. The only mandatory parameter is the algorithm name, but you can also define the dictionary of parameters so that the dialog will be filled automatically:

```

1  >>> my_dialog = processing.createAlgorithmDialog("native:buffer", {
2          'INPUT': '/data/lines.shp',
3          'DISTANCE': 100.0,
4          'SEGMENTS': 10,
5          'DISSOLVE': True,
6          'END_CAP_STYLE': 0,
7          'JOIN_STYLE': 0,
8          'MITER_LIMIT': 10,
9          'OUTPUT': '/data/buffers.shp'})
10 >>> my_dialog.show()

```

The `execAlgorithmDialog` method opens the dialog immediately:

```

1  >>> processing.execAlgorithmDialog("native:buffer", {
2          'INPUT': '/data/lines.shp',
3          'DISTANCE': 100.0,
4          'SEGMENTS': 10,
5          'DISSOLVE': True,
6          'END_CAP_STYLE': 0,
7          'JOIN_STYLE': 0,
8          'MITER_LIMIT': 10,
9          'OUTPUT': '/data/buffers.shp'})

```

23.7.2 Creating scripts and running them from the toolbox

You can create your own algorithms by writing Python code. Processing scripts extend `QgsProcessingAlgorithm`, so you need to add some extra lines of code to implement mandatory functions. You can find *Create new script* (clean sheet) and *Create New Script from Template* (template that includes code for mandatory functions of `QgsProcessingAlgorithm`) under the *Scripts* dropdown menu on the top of the Processing toolbox. The Processing Script Editor will open, and that's where you should type your code. Saving the script from there in the `scripts` folder (the default folder when you open the save file dialog) with a `.py` extension should create the corresponding algorithm.

The name of the algorithm (the one you will see in the toolbox) is defined within the code.

Let's have a look at the following code, which defines a Processing algorithm that performs a buffer operation with a user defined buffer distance on a vector layer that is specified by the user, after first smoothing the layer.

```

1 from qgis.core import (QgsProcessingAlgorithm,
2     QgsProcessingParameterNumber,
3     QgsProcessingParameterFeatureSource,
4     QgsProcessingParameterFeatureSink)
5
6 from qgis import processing
7
8 class algTest(QgsProcessingAlgorithm):
9     INPUT_BUFFERDIST = 'BUFFERDIST'
10    OUTPUT_BUFFER = 'OUTPUT_BUFFER'
11    INPUT_VECTOR = 'INPUT_VECTOR'
12
13    def __init__(self):
14        super().__init__()
15
16    def name(self):
17        return "algTest"
18
19    def displayName(self):
20        return "algTest script"
21
22    def createInstance(self):
23        return type(self)()
24
25    def initAlgorithm(self, config=None):
26        self.addParameter(QgsProcessingParameterFeatureSource(
27            self.INPUT_VECTOR, "Input vector"))
28        self.addParameter(QgsProcessingParameterNumber(
29            self.INPUT_BUFFERDIST, "Buffer distance",
30            QgsProcessingParameterNumber.Double,
31            100.0))
32        self.addParameter(QgsProcessingParameterFeatureSink(
33            self.OUTPUT_BUFFER, "Output buffer"))
34
35    def processAlgorithm(self, parameters, context, feedback):
36        #DO SOMETHING
37        alresult = processing.run("native:smoothgeometry",
38            {'INPUT': parameters[self.INPUT_VECTOR],
39            'ITERATIONS':2,
40            'OFFSET':0.25,
41            'MAX_ANGLE':180,
42            'OUTPUT': 'memory:'},
43            context=context, feedback=feedback, is_child_algorithm=True)
44        smoothed = alresult['OUTPUT']
45        alresult = processing.run('native:buffer',
46            {'INPUT': smoothed,
47            'DISTANCE': parameters[self.INPUT_BUFFERDIST],

```

(下页继续)

(续上页)

```

48         'SEGMENTS': 5,
49         'END_CAP_STYLE': 0,
50         'JOIN_STYLE': 0,
51         'MITER_LIMIT': 10,
52         'DISSOLVE': True,
53         'OUTPUT': parameters[self.OUTPUT_BUFFER]},
54         context=context, feedback=feedback, is_child_algorithm=True)
55     buffered = algresult['OUTPUT']
56     return {self.OUTPUT_BUFFER: buffered}

```

After doing the necessary imports, the following `QgsProcessingAlgorithm` functions are specified:

- `name()`: The id of the algorithm (lowercase).
- `displayName()`: A human readable name for the algorithm.
- `createInstance()`: Create a new instance of the algorithm class.
- `initAlgorithm()`: Configure the `parameterDefinitions` and `outputDefinitions`.

Here you describe the parameters and output of the algorithm. In this case, a feature source for the input, a feature sink for the result and a number for the buffer distance.

- `processAlgorithm()`: Do the work.

Here we first run the `smoothgeometry` algorithm to smooth the geometry, and then we run the `buffer` algorithm on the smoothed output. To be able to run algorithms from within another algorithm we have to set the `is_child_algorithm` argument to `True`. You can see how input and output parameters are used as parameters to the `smoothgeometry` and `buffer` algorithms.

There are a number of different parameter types available for input and output. Below is an alphabetically sorted list:

- `QgsProcessingParameterAggregate`
- `QgsProcessingParameterAuthConfig`
- `QgsProcessingParameterBand`
- `QgsProcessingParameterBoolean`
- `QgsProcessingParameterColor`
- `QgsProcessingParameterCoordinateOperation`
- `QgsProcessingParameterCrs`
- `QgsProcessingParameterDatabaseSchema`
- `QgsProcessingParameterDatabaseTable`
- `QgsProcessingParameterDateTime`
- `QgsProcessingParameterDistance`
- `QgsProcessingParameterEnum`
- `QgsProcessingParameterExpression`
- `QgsProcessingParameterExtent`
- `QgsProcessingParameterFeatureSink`
- `QgsProcessingParameterFeatureSource`
- `QgsProcessingParameterField`
- `QgsProcessingParameterFieldMapping`
- `QgsProcessingParameterFile`
- `QgsProcessingParameterFileDestination`

- `QgsProcessingParameterFolderDestination`
- `QgsProcessingParameterLayout`
- `QgsProcessingParameterLayoutItem`
- `QgsProcessingParameterMapLayer`
- `QgsProcessingParameterMapTheme`
- `QgsProcessingParameterMatrix`
- `QgsProcessingParameterMeshLayer`
- `QgsProcessingParameterMultipleLayers`
- `QgsProcessingParameterNumber`
- `QgsProcessingParameterPoint`
- `QgsProcessingParameterProviderConnection`
- `QgsProcessingParameterRange`
- `QgsProcessingParameterRasterDestination`
- `QgsProcessingParameterRasterLayer`
- `QgsProcessingParameterScale`
- `QgsProcessingParameterString`
- `QgsProcessingParameterVectorDestination`
- `QgsProcessingParameterVectorLayer`
- `QgsProcessingParameterVectorTileWriterLayers`

The first parameter to the constructors is the name of the parameter, and the second is the description of the parameter (for the user interface). The rest of the constructor parameters are parameter type specific.

The input can be turned into QGIS classes using the `parameterAs` functions of `QgsProcessingAlgorithm`. For instance to get the number provided for the buffer distance as a double:

```
self.parameterAsDouble(parameters, self.INPUT_BUFFERDIST, context).
```

The `processAlgorithm` function should return a dictionary containing values for every output defined by the algorithm. This allows access to these outputs from other algorithms, including other algorithms contained within the same model.

Well behaved algorithms should define and return as many outputs as makes sense. Non-feature outputs, such as numbers and strings, are very useful when running your algorithm as part of a larger model, as these values can be used as input parameters for subsequent algorithms within the model. Consider adding numeric outputs for things like the number of features processed, the number of invalid features encountered, the number of features output, etc. The more outputs you return, the more useful your algorithm becomes!

Feedback

The `feedback` object passed to `processAlgorithm()` should be used for user feedback / interaction. You can use the `setProgress()` function of the `feedback` object to update the progress bar (0 to 100) to inform the user about the progress of the algorithm. This is very useful if your algorithm takes a long time to complete.

The `feedback` object provides an `isCanceled()` method that should be monitored to enable cancelation of the algorithm by the user. The `pushInfo()` method of `feedback` can be used to send information to the user, and `reportError()` is handy for pushing non-fatal errors to users.

Algorithms should avoid using other forms of providing feedback to users, such as print statements or logging to `QgsMessageLog`, and should always use the feedback object instead. This allows verbose logging for the algorithm, and is also thread-safe (which is important, given that algorithms are typically run in a background thread).

Handling errors

If your algorithm encounters an error which prevents it from executing, such as invalid input values or some other condition from which it cannot or should not recover, then you should raise a `QgsProcessingException`. E.g.:

```
if feature['value'] < 20:
    raise QgsProcessingException('Invalid input value {}, must be >= 20'.
    ↪format(feature['value']))
```

Try to avoid raising `QgsProcessingException` for non-fatal errors (e.g. when a feature has a null geometry), and instead just report these errors via `feedback.reportError()` and skip the feature. This helps make your algorithm "model-friendly", as it avoids halting the execution of an entire algorithm when a non-fatal error is encountered.

Documenting your scripts

As in the case of models, you can create additional documentation for your scripts, to explain what they do and how to use them.

`QgsProcessingAlgorithm` provides the `helpString()`, `shortHelpString()` and `helpUrl()` functions for that purpose. Specify / override these to provide more help to the user.

`shortDescription()` is used in the tooltip when hovering over the algorithm in the toolbox.

23.7.3 Pre- and post-execution script hooks

Scripts can also be used as pre- and post-execution hooks that are run before and after an algorithm is run, respectively. This can be used to automate tasks that should be performed whenever an algorithm is executed.

The syntax is identical to the syntax explained above, but an additional global variable named `alg` is available, representing the algorithm that has just been (or is about to be) executed.

In the *General* group of the processing options dialog, you will find two entries named *Pre-execution script* and *Post-execution script* where the filenames of the scripts to be run in each case can be entered.

23.8 Using processing from the command line

QGIS comes with a tool called `QGIS Processing Executor` which allows you to run Processing algorithms and models (built-in or provided by plugins) directly from the command line without starting QGIS Desktop itself.

From a command line tool, run `qgis_process` and you should get:

```
QGIS Processing Executor - 3.16.8-Hannover 'Hannover' (3.16.8-Hannover)
Usage: C:\OSGeo4W\apps\qgis-ltr\bin\qgis_process.exe [--json] [command] [algorithm_
↪id or path to model file] [parameters]

Options:

  --json          Output results as JSON objects

Available commands:

  plugins          list available and active plugins
  plugins enable  enables an installed plugin. The plugin name must be specified, ↪
↪e.g. "plugins enable cartography_tools"
  plugins disable disables an installed plugin. The plugin name must be specified, ↪
↪e.g. "plugins disable cartography_tools"
  list            list all available processing algorithms
```

(下页继续)

```

help          show help for an algorithm. The algorithm id or a path to a
↳model file must be specified.
run          runs an algorithm. The algorithm id or a path to a model file
↳and parameter values must be specified.
             Parameter values are specified after -- with PARAMETER=VALUE
↳syntax.
             Ordered list values for a parameter can be created by
↳specifying the parameter multiple times,
             e.g. --LAYERS=layer1.shp --LAYERS=layer2.shp
             If required, the ellipsoid to use for distance and area
↳calculations can be specified via the "--ELLIPSOID=name" argument.
             If required, an existing QGIS project to use during the
↳algorithm execution can be specified via the "--PROJECT_PATH=path" argument.

```

注解: Only installed plugins that advertize `hasProcessingProvider=yes` in their `metadata.txt` file are recognized and can be activated or loaded by `qgis_process` tool.

The command `list` can be used to get a list of all available providers and algorithms.

```
qgis_process list
```

The command `help` can be used to get further information about commands or algorithms.

```
qgis_process help qgis:regularpoints
```

The command `run` can be used to run an algorithm or model. Specify the name of the algorithm or a path to a model as first parameter.

```
qgis_process run qgis:buffer -- INPUT=source.shp DISTANCE=2 OUTPUT=buffered.shp
```

Where a parameter accepts a list of values, set the same variable multiple times.

```
qgis_process run native:mergevectorlayers -- LAYERS=input1.shp LAYERS=input2.shp
↳OUTPUT=merged.shp
```

While running an algorithm a text-based feedback bar is shown, and the operation can be cancelled via `CTRL+C`. The `run` command also supports further parameters.

- `--json` will format stdout output in a JSON structured way.
- `--ellipsoid` will set the ellipsoid to the specified one.
- `--distance_units` will use the specified distance units.
- `--area_units` will use the specified area units.
- `--project_path` will load the specified project for running the algorithm.

23.9 Writing new Processing algorithms as Python scripts

There are two options for writing Processing algorithms using Python.

- *Extending `QgsProcessingAlgorithm`*
- *Using the `@alg decorator`*

Within QGIS, you can use *Create new script* in the *Scripts* menu at the top of the *Processing Toolbox* to open the *Processing Script Editor* where you can write your code. To simplify the task, you can start with a script template by using *Create new script from template* from the same menu. This opens a template that extends `QgsProcessingAlgorithm`.

If you save the script in the `scripts` folder (the default location) with a `.py` extension, the algorithm will become available in the *Processing Toolbox*.

23.9.1 Extending QgsProcessingAlgorithm

The following code

1. takes a vector layer as input
2. counts the number of features
3. does a buffer operation
4. creates a raster layer from the result of the buffer operation
5. returns the buffer layer, raster layer and number of features

```

1  from qgis.PyQt.QtCore import QApplication
2  from qgis.core import (QgsProcessing,
3                        QgsProcessingAlgorithm,
4                        QgsProcessingException,
5                        QgsProcessingOutputNumber,
6                        QgsProcessingParameterDistance,
7                        QgsProcessingParameterFeatureSource,
8                        QgsProcessingParameterVectorDestination,
9                        QgsProcessingParameterRasterDestination)
10 from qgis import processing
11
12
13 class ExampleProcessingAlgorithm(QgsProcessingAlgorithm):
14     """
15     This is an example algorithm that takes a vector layer,
16     creates some new layers and returns some results.
17     """
18
19     def tr(self, string):
20         """
21         Returns a translatable string with the self.tr() function.
22         """
23         return QApplication.translate('Processing', string)
24
25     def createInstance(self):
26         # Must return a new copy of your algorithm.
27         return ExampleProcessingAlgorithm()
28
29     def name(self):
30         """
31         Returns the unique algorithm name.
32         """
33         return 'bufferrasterextend'
34
35     def displayName(self):
36         """
37         Returns the translated algorithm name.
38         """
39         return self.tr('Buffer and export to raster (extend)')
40
41     def group(self):
42         """
43         Returns the name of the group this algorithm belongs to.
44         """
45         return self.tr('Example scripts')
46

```

(下页继续)

```
47 def groupId(self):
48     """
49     Returns the unique ID of the group this algorithm belongs
50     to.
51     """
52     return 'examplescripts'
53
54 def shortHelpString(self):
55     """
56     Returns a localised short help string for the algorithm.
57     """
58     return self.tr('Example algorithm short description')
59
60 def initAlgorithm(self, config=None):
61     """
62     Here we define the inputs and outputs of the algorithm.
63     """
64     # 'INPUT' is the recommended name for the main input
65     # parameter.
66     self.addParameter(
67         QgsProcessingParameterFeatureSource(
68             'INPUT',
69             self.tr('Input vector layer'),
70             types=[QgsProcessing.TypeVectorAnyGeometry]
71         )
72     )
73     self.addParameter(
74         QgsProcessingParameterVectorDestination(
75             'BUFFER_OUTPUT',
76             self.tr('Buffer output'),
77         )
78     )
79     # 'OUTPUT' is the recommended name for the main output
80     # parameter.
81     self.addParameter(
82         QgsProcessingParameterRasterDestination(
83             'OUTPUT',
84             self.tr('Raster output')
85         )
86     )
87     self.addParameter(
88         QgsProcessingParameterDistance(
89             'BUFFERDIST',
90             self.tr('BUFFERDIST'),
91             defaultValue = 1.0,
92             # Make distance units match the INPUT layer units:
93             parentParameterName='INPUT'
94         )
95     )
96     self.addParameter(
97         QgsProcessingParameterDistance(
98             'CELLSIZE',
99             self.tr('CELLSIZE'),
100            defaultValue = 10.0,
101            parentParameterName='INPUT'
102        )
103    )
104    self.addOutput(
105        QgsProcessingOutputNumber(
106            'NUMBEROFFEATURES',
107            self.tr('Number of features processed')
```

```

108     )
109     )
110
111     def processAlgorithm(self, parameters, context, feedback):
112         """
113         Here is where the processing itself takes place.
114         """
115         # First, we get the count of features from the INPUT layer.
116         # This layer is defined as a QgsProcessingParameterFeatureSource
117         # parameter, so it is retrieved by calling
118         # self.parameterAsSource.
119         input_featuresource = self.parameterAsSource(parameters,
120                                                     'INPUT',
121                                                     context)
122
123         numfeatures = input_featuresource.featureCount()
124
125         # Retrieve the buffer distance and raster cell size numeric
126         # values. Since these are numeric values, they are retrieved
127         # using self.parameterAsDouble.
128         bufferdist = self.parameterAsDouble(parameters, 'BUFFERDIST',
129                                             context)
130
131         rastercellsize = self.parameterAsDouble(parameters, 'CELLSIZE',
132                                                context)
133
134         if feedback.isCanceled():
135             return {}
136
137         buffer_result = processing.run(
138             'native:buffer',
139             {
140                 # Here we pass on the original parameter values of INPUT
141                 # and BUFFER_OUTPUT to the buffer algorithm.
142                 'INPUT': parameters['INPUT'],
143                 'OUTPUT': parameters['BUFFER_OUTPUT'],
144                 'DISTANCE': bufferdist,
145                 'SEGMENTS': 10,
146                 'DISSOLVE': True,
147                 'END_CAP_STYLE': 0,
148                 'JOIN_STYLE': 0,
149                 'MITER_LIMIT': 10
150             },
151             # Because the buffer algorithm is being run as a step in
152             # another larger algorithm, the is_child_algorithm option
153             # should be set to True
154             is_child_algorithm=True,
155             #
156             # It's important to pass on the context and feedback objects to
157             # child algorithms, so that they can properly give feedback to
158             # users and handle cancelation requests.
159             context=context,
160             feedback=feedback)
161
162         # Check for cancelation
163         if feedback.isCanceled():
164             return {}
165
166         # Run the separate rasterization algorithm using the buffer result
167         # as an input.
168         rasterized_result = processing.run(
169             'qgis:rasterize',
170             {
171                 # Here we pass the 'OUTPUT' value from the buffer's result
172                 # dictionary off to the rasterize child algorithm.

```

```

169         'LAYER': buffer_result['OUTPUT'],
170         'EXTENT': buffer_result['OUTPUT'],
171         'MAP_UNITS_PER_PIXEL': rastercellsize,
172         # Use the original parameter value.
173         'OUTPUT': parameters['OUTPUT']
174     },
175     is_child_algorithm=True,
176     context=context,
177     feedback=feedback)
178
179     if feedback.isCanceled():
180         return {}
181
182     # Return the results
183     return {'OUTPUT': rasterized_result['OUTPUT'],
184            'BUFFER_OUTPUT': buffer_result['OUTPUT'],
185            'NUMBEROFFEATURES': numfeatures}

```

Processing algorithm standard functions:

- **createInstance (mandatory)** Must return a new copy of your algorithm. If you change the name of the class, make sure you also update the value returned here to match!
- **name (mandatory)** Returns the unique algorithm name, used for identifying the algorithm.
- **displayName (mandatory)** Returns the translated algorithm name.
- **group** Returns the name of the group this algorithm belongs to.
- **groupId** Returns the unique ID of the group this algorithm belongs to.
- **shortHelpString** Returns a localised short help string for the algorithm.
- **initAlgorithm (mandatory)** Here we define the inputs and outputs of the algorithm.

INPUT and OUTPUT are recommended names for the main input and main output parameters, respectively.

If a parameter depends on another parameter, `parentParameterName` is used to specify this relationship (could be the field / band of a layer or the distance units of a layer).

- **processAlgorithm (mandatory)** This is where the processing takes place.

Parameters are retrieved using special purpose functions, for instance `parameterAsSource` and `parameterAsDouble`.

`processing.run` can be used to run other processing algorithms from a processing algorithm. The first parameter is the name of the algorithm, the second is a dictionary of the parameters to the algorithm. `is_child_algorithm` is normally set to `True` when running an algorithm from within another algorithm. `context` and `feedback` inform the algorithm about the environment to run in and the channel for communicating with the user (catching cancel request, reporting progress, providing textual feedback). When using the (parent) algorithm's parameters as parameters to "child" algorithms, the original parameter values should be used (e.g. `parameters['OUTPUT']`).

It is good practice to check the feedback object for cancelation as much as is sensibly possible! Doing so allows for responsive cancelation, instead of forcing users to wait for unwanted processing to occur.

The algorithm should return values for all the output parameters it has defined as a dictionary. In this case, that's the buffer and rasterized output layers, and the count of features processed. The dictionary keys must match the original parameter/output names.

23.9.2 The @alg decorator

Using the @alg decorator, you can create your own algorithms by writing the Python code and adding a few extra lines to supply additional information needed to make it a proper Processing algorithm. This simplifies the creation of algorithms and the specification of inputs and outputs.

One important limitation with the decorator approach is that algorithms created in this way will always be added to a user's Processing Scripts provider -- it is not possible to add these algorithms to a custom provider, e.g. for use in plugins.

The following code uses the @alg decorator to

1. use a vector layer as input
2. count the number of features
3. do a buffer operation
4. create a raster layer from the result of the buffer operation
5. returns the buffer layer, raster layer and number of features

```

1 from qgis import processing
2 from qgis.processing import alg
3 from qgis.core import QgsProject
4
5 @alg(name='bufferrasteralg', label='Buffer and export to raster (alg)',
6      group='examplescripts', group_label='Example scripts')
7 # 'INPUT' is the recommended name for the main input parameter
8 @alg.input(type=alg.SOURCE, name='INPUT', label='Input vector layer')
9 # 'OUTPUT' is the recommended name for the main output parameter
10 @alg.input(type=alg.RASTER_LAYER_DEST, name='OUTPUT',
11           label='Raster output')
12 @alg.input(type=alg.VECTOR_LAYER_DEST, name='BUFFER_OUTPUT',
13           label='Buffer output')
14 @alg.input(type=alg.DISTANCE, name='BUFFERDIST', label='BUFFER DISTANCE',
15           default=1.0)
16 @alg.input(type=alg.DISTANCE, name='CELLSIZE', label='RASTER CELL SIZE',
17           default=10.0)
18 @alg.output(type=alg.NUMBER, name='NUMBEROFFEATURES',
19            label='Number of features processed')
20
21 def bufferrasteralg(instance, parameters, context, feedback, inputs):
22     """
23     Description of the algorithm.
24     (If there is no comment here, you will get an error)
25     """
26     input_featuresource = instance.parameterAsSource(parameters,
27                                                    'INPUT', context)
28     numfeatures = input_featuresource.featureCount()
29     bufferdist = instance.parameterAsDouble(parameters, 'BUFFERDIST',
30                                           context)
31     rastercellsize = instance.parameterAsDouble(parameters, 'CELLSIZE',
32                                               context)
33     if feedback.isCanceled():
34         return {}
35     buffer_result = processing.run('native:buffer',
36                                  {'INPUT': parameters['INPUT'],
37                                   'OUTPUT': parameters['BUFFER_OUTPUT'],
38                                   'DISTANCE': bufferdist,
39                                   'SEGMENTS': 10,
40                                   'DISSOLVE': True,
41                                   'END_CAP_STYLE': 0,
42                                   'JOIN_STYLE': 0,

```

(下页继续)

```

43         'MITER_LIMIT': 10
44     },
45     is_child_algorithm=True,
46     context=context,
47     feedback=feedback)
48
49     if feedback.isCanceled():
50         return {}
51     rasterized_result = processing.run('qgis:rasterize',
52                                     {'LAYER': buffer_result['OUTPUT'],
53                                      'EXTENT': buffer_result['OUTPUT'],
54                                      'MAP_UNITS_PER_PIXEL': rastercellsize,
55                                      'OUTPUT': parameters['OUTPUT']
56                                     },
57                                     is_child_algorithm=True, context=context,
58                                     feedback=feedback)
59
60     if feedback.isCanceled():
61         return {}
62     return {'OUTPUT': rasterized_result['OUTPUT'],
63            'BUFFER_OUTPUT': buffer_result['OUTPUT'],
64            'NUMBEROFFEATURES': numfeatures}

```

As you can see, it involves two algorithms ('native:buffer' and 'qgis:rasterize'). The last one ('qgis:rasterize') creates a raster layer from the buffer layer that was generated by the first one ('native:buffer').

The part of the code where this processing takes place is not difficult to understand if you have read the previous chapter. The first lines, however, need some additional explanation. They provide the information that is needed to turn your code into an algorithm that can be run from any of the GUI components, like the toolbox or the graphical modeler.

These lines are all calls to the @alg decorator functions that help simplify the coding of the algorithm.

- The @alg decorator is used to define the name and location of the algorithm in the Toolbox.
- The @alg.input decorator is used to define the inputs of the algorithm.
- The @alg.output decorator is used to define the outputs of the algorithm.

23.9.3 Input and output types for Processing Algorithms

Here is the list of input and output types that are supported in Processing with their corresponding alg decorator constants (algfactory.py contains the complete list of alg constants). Sorted on class name.

Input types

Class	Alg constant	Description
QgsProcessingParameterAuthConfig	alg.AUTH_CFG	Allows users to select from available authentication configurations or create new authentication configurations
QgsProcessingParameterBand	alg.BAND	A band of a raster layer
QgsProcessingParameterBoolean	alg.BOOL	A boolean value
QgsProcessingParameterColor	alg.COLOR	A color
QgsProcessingParameterCoordinateOperation	alg.COORDINATE_OPERATION	A coordinate operation (for CRS transformations)

下页继续

表 23.1 – 续上页

Class	Alg constant	Description
QgsProcessingParameterCrS	alg.CRS	A Coordinate Reference System
QgsProcessingParameterDatabaseSchema	alg.DATABASE_SCHEMA	A database schema
QgsProcessingParameterDatabaseTable	alg.DATABASE_TABLE	A database table
QgsProcessingParameterDateTime	alg.DATETIME	A datetime (or a pure date or time)
QgsProcessingParameterDistance	alg.DISTANCE	A double numeric parameter for distance values
QgsProcessingParameterEnum	alg.ENUM	An enumeration, allowing for selection from a set of predefined values
QgsProcessingParameterExpression	alg.EXPRESSION	An expression
QgsProcessingParameterExtent	alg.EXTENT	A spatial extent defined by xmin, xmax, ymin, ymax
QgsProcessingParameterField	alg.FIELD	A field in the attribute table of a vector layer
QgsProcessingParameterFile	alg.FILE	A filename of an existing file
QgsProcessingParameterFileDestination	alg.FILE_DEST	A filename for a newly created output file
QgsProcessingParameterFolderDestination	alg.FOLDER_DEST	A folder (destination folder)
QgsProcessingParameterNumber	alg.INT	An integer
QgsProcessingParameterLayout	alg.LAYOUT	A layout
QgsProcessingParameterLayoutItem	alg.LAYOUT_ITEM	A layout item
QgsProcessingParameterMapLayer	alg.MAP_LAYER	A map layer
QgsProcessingParameterMapTheme	alg.MAP_THEME	A project map theme
QgsProcessingParameterMatrix	alg.MATRIX	A matrix
QgsProcessingParameterMeshLayer	alg.MESH_LAYER	A mesh layer
QgsProcessingParameterMultipleLayers	alg.MULTILAYER	A set of layers
QgsProcessingParameterNumber	alg.NUMBER	A numerical value
QgsProcessingParameterPoint	alg.POINT	A point
QgsProcessingParameterProviderConnection	alg.PROVIDER_CONNECTION	An available connection for a database provider
QgsProcessingParameterRange	alg.RANGE	A number range
QgsProcessingParameterRasterLayer	alg.RASTER_LAYER	A raster layer
QgsProcessingParameterRasterDestination	alg.RASTER_LAYER_DEST	A raster layer
QgsProcessingParameterScale	alg.SCALE	A map scale
QgsProcessingParameterFeatureSink	alg.SINK	A feature sink
QgsProcessingParameterFeatureSource	alg.SOURCE	A feature source
QgsProcessingParameterString	alg.STRING	A text string
QgsProcessingParameterVectorLayer	alg.VECTOR_LAYER	A vector layer
QgsProcessingParameterVectorDestination	alg.VECTOR_LAYER_DEST	A vector layer

Output types

Class	Alg constant	Description
<code>QgsProcessingOutputBoolean</code>	<code>alg.BOOL</code>	A boolean value
<code>QgsProcessingOutputNumber</code>	<code>alg.DISTANCE</code>	A double numeric parameter for distance values
<code>QgsProcessingOutputFile</code>	<code>alg.FILE</code>	A filename of an existing file
<code>QgsProcessingOutputFolder</code>	<code>alg.FOLDER</code>	A folder
<code>QgsProcessingOutputHtml</code>	<code>alg.HTML</code>	HTML
<code>QgsProcessingOutputNumber</code>	<code>alg.INT</code>	A integer
<code>QgsProcessingOutputLayerDefinition</code>	<code>alg.LAYERDEF</code>	A layer definition
<code>QgsProcessingOutputMapLayer</code>	<code>alg.MAPLAYER</code>	A map layer
<code>QgsProcessingOutputMultipleLayers</code>	<code>alg.MULTILAYER</code>	A set of layers
<code>QgsProcessingOutputNumber</code>	<code>alg.NUMBER</code>	A numerical value
<code>QgsProcessingOutputRasterLayer</code>	<code>alg.RASTER_LAYER</code>	A raster layer
<code>QgsProcessingOutputString</code>	<code>alg.STRING</code>	A text string
<code>QgsProcessingOutputVectorLayer</code>	<code>alg.VECTOR_LAYER</code>	A vector layer

23.9.4 Handing algorithm output

When you declare an output representing a layer (raster or vector), the algorithm will try to add it to QGIS once it is finished.

- Raster layer output: `QgsProcessingParameterRasterDestination` / `alg.RASTER_LAYER_DEST`.
- Vector layer output: `QgsProcessingParameterVectorDestination` / `alg.VECTOR_LAYER_DEST`.

So even if the `processing.run()` method does not add the layers it creates to the user's current project, the two output layers (buffer and raster buffer) will be loaded, since they are saved to the destinations entered by the user (or to temporary destinations if the user does not specify destinations).

If a layer is created as output of an algorithm, it should be declared as such. Otherwise, you will not be able to properly use the algorithm in the modeler, since what is declared will not match what the algorithm really creates.

You can return strings, numbers and more by specifying them in the result dictionary (as demonstrated for "NUMBEROFFEATURES"), but they should always be explicitly defined as outputs from your algorithm. We encourage algorithms to output as many useful values as possible, since these can be valuable for use in later algorithms when your algorithm is used as part of a model.

23.9.5 Communicating with the user

If your algorithm takes a long time to process, it is a good idea to inform the user about the progress. You can use `feedback` (`QgsProcessingFeedback`) for this.

The progress text and progressbar can be updated using two methods: `setProgressText(text)` and `setProgress(percent)`.

You can provide more information by using `pushCommandInfo(text)`, `pushDebugInfo(text)`, `pushInfo(text)` and `reportError(text)`.

If your script has a problem, the correct way of handling it is to raise a `QgsProcessingException`. You can pass a message as an argument to the constructor of the exception. Processing will take care of handling it and communicating with the user, depending on where the algorithm is being executed from (toolbox, modeler, Python console, ...)

23.9.6 Documenting your scripts

You can document your scripts by overloading the `helpString()` and `helpUrl()` methods of `QgsProcessingAlgorithm`.

23.9.7 Flags

You can override the `flags()` method of `QgsProcessingAlgorithm` to tell QGIS more about your algorithm. You can for instance tell QGIS that the script shall be hidden from the modeler, that it can be canceled, that it is not thread safe, and more.

小技巧: By default, Processing runs algorithms in a separate thread in order to keep QGIS responsive while the processing task runs. If your algorithm is regularly crashing, you are probably using API calls which are not safe to do in a background thread. Try returning the `QgsProcessingAlgorithm.FlagNoThreading` flag from your algorithm's `flags()` method to force Processing to run your algorithm in the main thread instead.

23.9.8 Best practices for writing script algorithms

Here's a quick summary of ideas to consider when creating your script algorithms and, especially, if you want to share them with other QGIS users. Following these simple rules will ensure consistency across the different Processing elements such as the toolbox, the modeler or the batch processing interface.

- Do not load resulting layers. Let Processing handle your results and load your layers if needed.
- Always declare the outputs your algorithm creates.
- Do not show message boxes or use any GUI element from the script. If you want to communicate with the user, use the methods of the feedback object (`QgsProcessingFeedback`) or throw a `QgsProcessingException`.

There are already many processing algorithms available in QGIS. You can find code on https://github.com/qgis/QGIS/blob/release-3_16/python/plugins/processing/algs/qgis.

23.10 Configuring external applications

The processing framework can be extended using additional applications. Algorithms that rely on external applications are managed by their own algorithm providers. Additional providers can be found as separate plugins, and installed using the QGIS Plugin Manager.

This section will show you how to configure the Processing framework to include these additional applications, and it will explain some particular features of the algorithms based on them. Once you have correctly configured the system, you will be able to execute external algorithms from any component like the toolbox or the graphical modeler, just like you do with any other algorithm.

By default, algorithms that rely on an external application not shipped with QGIS are not enabled. You can enable them in the Processing settings dialog if they are installed on your system.

23.10.1 A note for Windows users

If you are not an advanced user and you are running QGIS on Windows, you might not be interested in reading the rest of this chapter. Make sure you install QGIS in your system using the standalone installer. That will automatically install SAGA and GRASS in your system and configure them so they can be run from QGIS. All the algorithms from these providers will be ready to be run without needing any further configuration. If installing with the OSGeo4W application, make sure that you also select SAGA and GRASS for installation.

23.10.2 A note on file formats

When using external software, opening a file in QGIS does not mean that it can be opened and processed in that other software. In most cases, other software can read what you have opened in QGIS, but in some cases, that might not be true. When using databases or uncommon file formats, whether for raster or vector layers, problems might arise. If that happens, try to use well-known file formats that you are sure are understood by both programs, and check the console output (in the log panel) to find out what is going wrong.

You might for instance get trouble and not be able to complete your work if you call an external algorithm with a GRASS raster layers as input. For this reason, such layers will not appear as available to algorithms.

You should, however, not have problems with vector layers, since QGIS automatically converts from the original file format to one accepted by the external application before passing the layer to it. This adds extra processing time, which might be significant for large layers, so do not be surprised if it takes more time to process a layer from a DB connection than a layer from a Shapefile format dataset of similar size.

Providers not using external applications can process any layer that you can open in QGIS, since they open it for analysis through QGIS.

All raster and vector output formats produced by QGIS can be used as input layers. Some providers do not support certain formats, but all can export to common formats that can later be transformed by QGIS automatically. As for input layers, if a conversion is needed, that might increase the processing time.

23.10.3 A note on vector layer selections

External applications may also be made aware of the selections that exist in vector layers within QGIS. However, that requires rewriting all input vector layers, just as if they were originally in a format not supported by the external application. Only when no selection exists, or the *Use only selected features* option is not enabled in the processing general configuration, can a layer be directly passed to an external application.

In other cases, exporting only selected features is needed, which causes longer execution times.

23.10.4 SAGA

SAGA algorithms can be run from QGIS if SAGA is included with the QGIS installation.

If you are running Windows, both the stand-alone installer and the OSGeo4W installer include SAGA.

About SAGA grid system limitations

Most SAGA algorithms that require several input raster layers require them to have the same grid system. That is, they must cover the same geographic area and have the same cell size, so their corresponding grids match. When calling SAGA algorithms from QGIS, you can use any layer, regardless of its cell size and extent. When multiple raster layers are used as input for a SAGA algorithm, QGIS resamples them to a common grid system and then passes them to SAGA (unless the SAGA algorithm can operate with layers from different grid systems).

The definition of that common grid system is controlled by the user, and you will find several parameters in the SAGA group of the settings window to do so. There are two ways of setting the target grid system:

- Setting it manually. You define the extent by setting the values of the following parameters:

- *Resampling min X*
- *Resampling max X*
- *Resampling min Y*
- *Resampling max Y*
- *Resampling cellsize*

Notice that QGIS will resample input layers to that extent, even if they do not overlap with it.

- Setting it automatically from input layers. To select this option, just check the *Use min covering grid system for resampling* option. All the other settings will be ignored and the minimum extent that covers all the input layers will be used. The cell size of the target layer is the maximum of all cell sizes of the input layers.

For algorithms that do not use multiple raster layers, or for those that do not need a unique input grid system, no resampling is performed before calling SAGA, and those parameters are not used.

Limitations for multi-band layers

Unlike QGIS, SAGA has no support for multi-band layers. If you want to use a multiband layer (such as an RGB or multispectral image), you first have to split it into single-banded images. To do so, you can use the 'SAGA/Grid - Tools/Split RGB image' algorithm (which creates three images from an RGB image) or the 'SAGA/Grid - Tools/Extract band' algorithm (to extract a single band).

Limitations in cell size

SAGA assumes that raster layers have the same cell size in the X and Y axis. If you are working with a layer with different values for horizontal and vertical cell size, you might get unexpected results. In this case, a warning will be added to the processing log, indicating that an input layer might not be suitable to be processed by SAGA.

Logging

When QGIS calls SAGA, it does so using its command-line interface, thus passing a set of commands to perform all the required operations. SAGA shows its progress by writing information to the console, which includes the percentage of processing already done, along with additional content. This output is filtered and used to update the progress bar while the algorithm is running.

Both the commands sent by QGIS and the additional information printed by SAGA can be logged along with other processing log messages, and you might find them useful to track what is going on when QGIS runs a SAGA algorithm. You will find two settings, namely *Log console output* and *Log execution commands*, to activate that logging mechanism.

Most other providers that use external applications and call them through the command-line have similar options, so you will find them as well in other places in the processing settings list.

23.10.5 R scripts

To enable R in Processing you need to install the **Processing R Provider** plugin and configure R for QGIS.

Configuration is done in *Provider ► R* in the *Processing* tab of *Settings ► Options*.

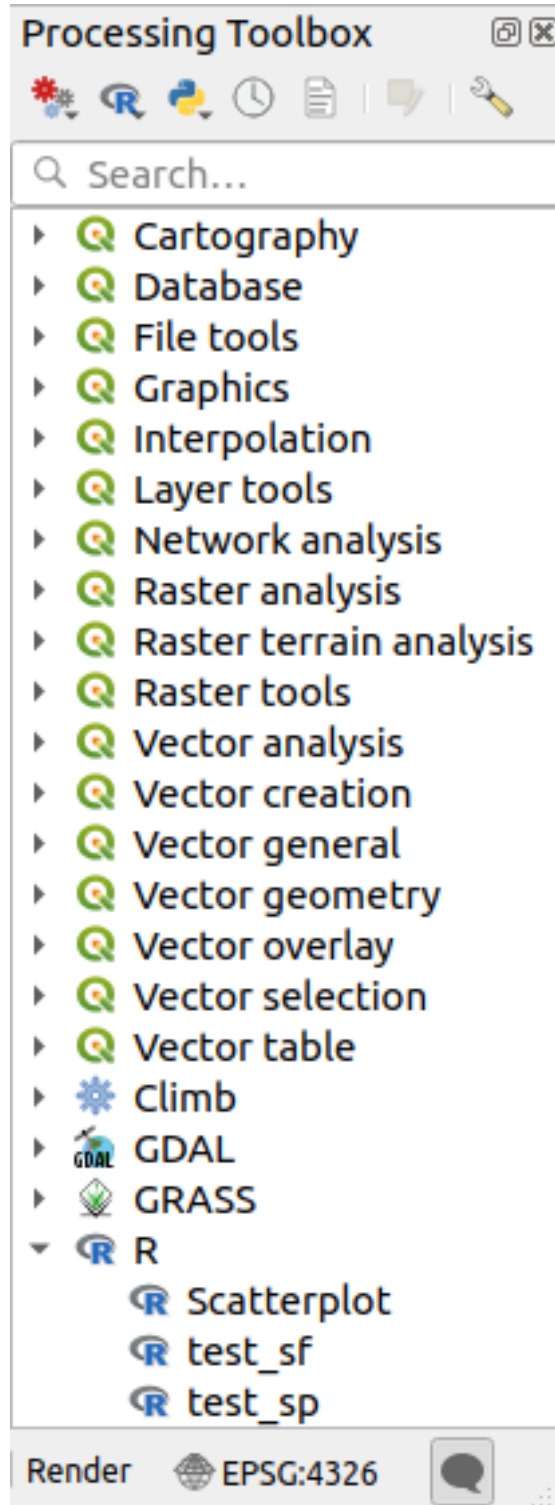
Depending on your operating system, you may have to use *R folder* to specify where your R binaries are located.

注解: On **Windows** the R executable file is normally in a folder (R-<version>) under C:\Program Files\R\. Specify the folder and **NOT** the binary!

On **Linux** you just have to make sure that the R folder is in the PATH environment variable. If R in a terminal window starts R, then you are ready to go.

After installing the **Processing R Provider** plugin, you will find some example scripts in the *Processing Toolbox*:

- *Scatterplot* runs an R function that produces a scatter plot from two numerical fields of the provided vector layer.
- *test_sf* does some operations that depend on the *sf* package and can be used to check if the R package *sf* is installed. If the package is not installed, R will try to install it (and all the packages it depends on) for you, using the *Package repository* specified in *Provider ► R* in the Processing options. The default is <https://cran.at.r-project.org/>. Installing may take some time...
- *test_sp* can be used to check if the R package *sp* is installed. If the package is not installed, R will try to install it for you.



If you have R configured correctly for QGIS, you should be able to run these scripts.

Adding R scripts from the QGIS collection

R integration in QGIS is different from that of SAGA in that there is not a predefined set of algorithms you can run (except for some example script that come with the *Processing R Provider* plugin).

A set of example R scripts is available in the QGIS Repository. Perform the following steps to load and enable them using the *QGIS Resource Sharing* plugin.

1. Add the *QGIS Resource Sharing* plugin (you may have to enable *Show also experimental plugins* in the *Plugin Manager Settings*)
2. Open it (Plugins --> Resource Sharing --> Resource Sharing)
3. Choose the *Settings* tab
4. Click *Reload repositories*
5. Choose the *All* tab
6. Select *QGIS R script collection* in the list and click on the *Install* button
7. The collection should now be listed in the *Installed* tab
8. Close the plugin
9. Open the *Processing Toolbox*, and if everything is OK, the example scripts will be present under R, in various groups (only some of the groups are expanded in the screenshot below).

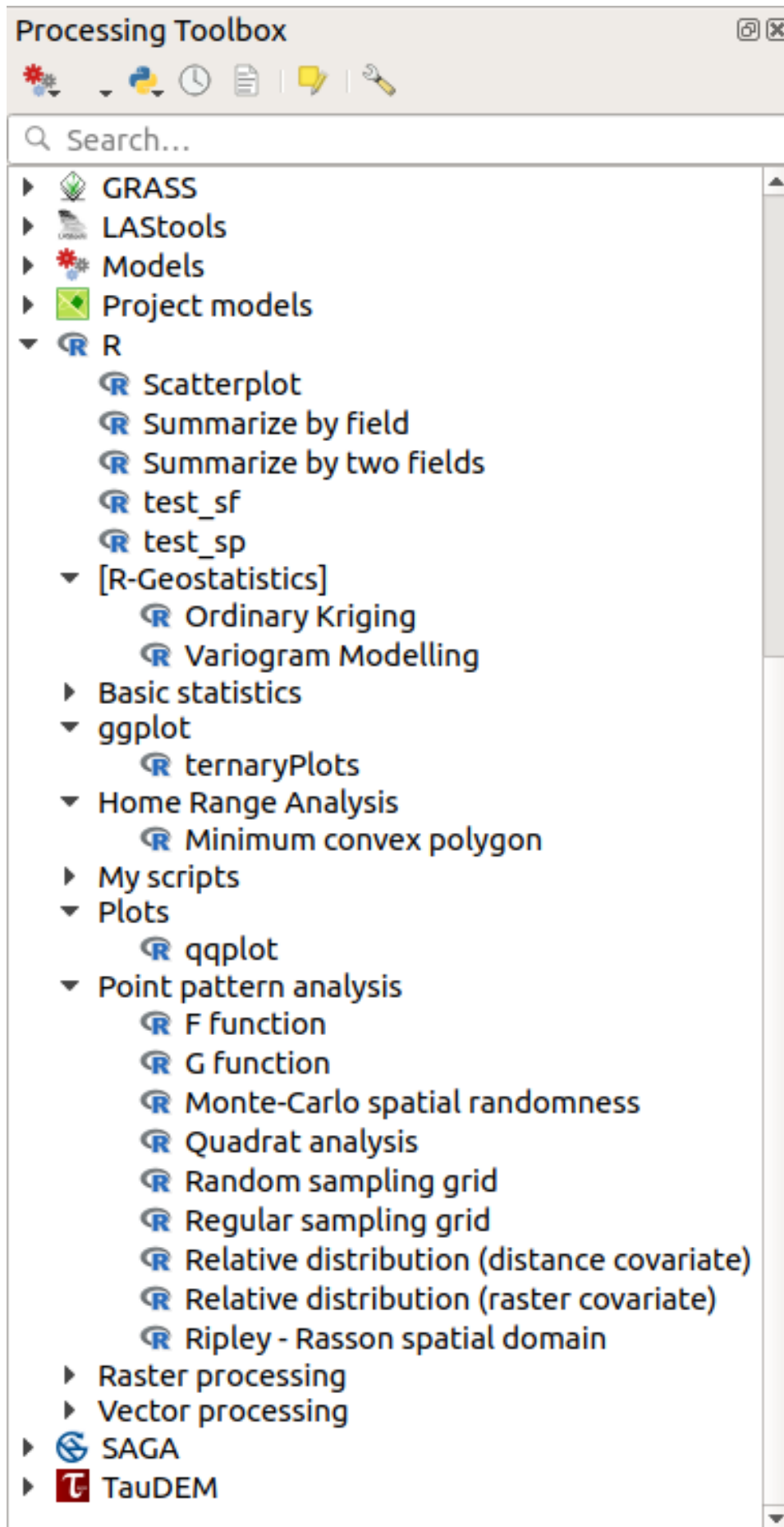


图 23.34: The Processing Toolbox with some R scripts shown

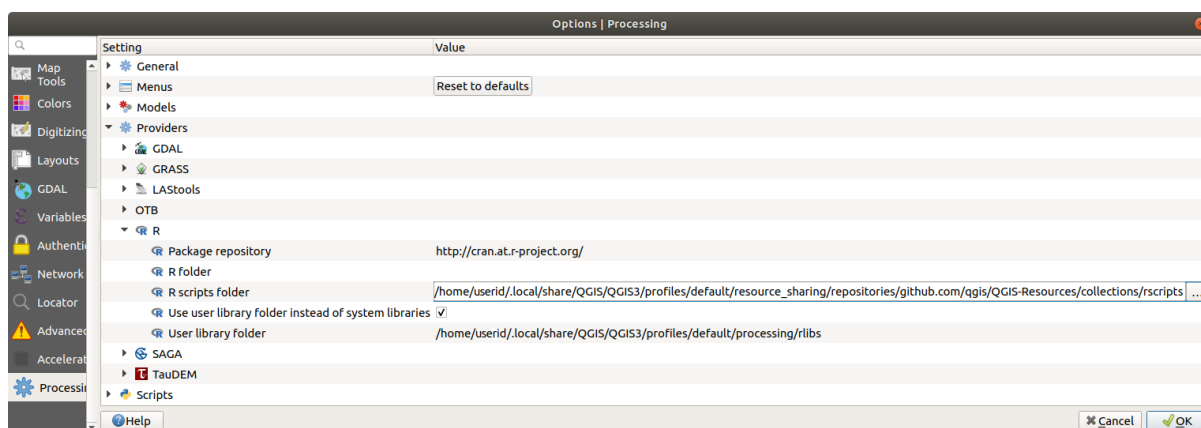
The scripts at the top are the example scripts from the *Processing R Provider* plugin.

10. If, for some reason, the scripts are not available in the *Processing Toolbox*, you can try to:

1. Open the Processing settings (*Settings* ► *Options* ► *Processing* tab)
2. Go to *Providers* ► *R* ► *R scripts folder*

- On Ubuntu, set the path to (or, better, include in the path):

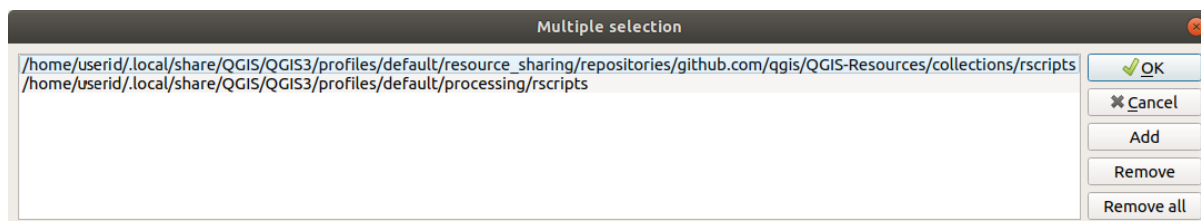
```
/home/<user>/.local/share/QGIS/QGIS3/profiles/default/resource_sharing/repositories/github.com/qgis/QGIS-Resources/collections/rscripts
```



- On Windows, set the path to (or, better, include in the path):

```
C:\Users<user>\AppData\Roaming\QGIS\QGIS3\profiles\default\resource_sharing\repositories\github.com\qgis-Resources\collections\rscripts
```

To edit, double-click. You can then choose to just paste / type the path, or you can navigate to the directory by using the ... button and press the *Add* button in the dialog that opens. It is possible to provide several directories here. They will be separated by a semicolon (“;”).



If you would like to get all the R scrips from the QGIS 2 on-line collection, you can select *QGIS R script collection (from QGIS 2)* instead of *QGIS R script collection*. You will probably find that scripts that depend on vector data input or output will not work.

Creating R scripts

You can write scripts and call R commands, as you would do from R. This section shows you the syntax for using R commands in QGIS, and how to use QGIS objects (layers, tables) in them.

To add an algorithm that calls an R function (or a more complex R script that you have developed and you would like to have available from QGIS), you have to create a script file that performs the R commands.

R script files have the extension `.rsx`, and creating them is pretty easy if you just have a basic knowledge of R syntax and R scripting. They should be stored in the R scripts folder. You can specify the folder (*R scripts folder*) in the R settings group in Processing settings dialog).

Let's have a look at a very simple script file, which calls the R method `spsample` to create a random grid within the boundary of the polygons in a given polygon layer. This method belongs to the `maptools` package. Since almost all the algorithms that you might like to incorporate into QGIS will use or generate spatial data, knowledge of spatial packages like `maptools` and `sp/sf`, is very useful.

```
##Random points within layer extent=name
##Point pattern analysis=group
##Vector_layer=vector
##Number_of_points=number 10
##Output=output vector
library(sp)
spatpoly = as(Vector_layer, "Spatial")
pts=spsample(spatpoly,Number_of_points,type="random")
spdf=SpatialPointsDataFrame(pts, as.data.frame(pts))
Output=st_as_sf(spdf)
```

The first lines, which start with a double Python comment sign (`##`), define the display name and group of the script, and tell QGIS about its inputs and outputs.

注解: To find out more about how to write your own R scripts, have a look at the R Intro section in the training manual and consult the [QGIS R Syntax](#) section.

When you declare an input parameter, QGIS uses that information for two things: creating the user interface to ask the user for the value of that parameter, and creating a corresponding R variable that can be used as R function input.

In the above example, we have declared an input of type `vector`, named `Vector_layer`. When executing the algorithm, QGIS will open the layer selected by the user and store it in a variable named `Vector_layer`. So, the name of a parameter is the name of the variable that you use in R for accessing the value of that parameter (you should therefore avoid using reserved R words as parameter names).

Spatial parameters such as vector and raster layers are read using the `st_read()` (or `readOGR`) and `brick()` (or `readGDAL`) commands (you do not have to worry about adding those commands to your description file -- QGIS will do it), and they are stored as `sf` (or `Spatial*DataFrame`) objects.

Table fields are stored as strings containing the name of the selected field.

Vector files can be read using the `readOGR()` command instead of `st_read()` by specifying `##load_vector_using_rgdal`. This will produce a `Spatial*DataFrame` object instead of an `sf` object.

Raster files can be read using the `readGDAL()` command instead of `brick()` by specifying `##load_raster_using_rgdal`.

If you are an advanced user and do not want QGIS to create the object for the layer, you can use `##pass_filenames` to indicate that you prefer a string with the filename. In this case, it is up to you to open the file before performing any operation on the data it contains.

With the above information, it is possible to understand the first lines of the R script (the first line not starting with a Python comment character).

```
library(sp)
spatpoly = as(Vector_layer, "Spatial")
pts=spsample(polyg,numpoints,type="random")
```

The `spsample` function is provided by the `sp` library, so the first thing we do is to load that library. The variable `Vector_layer` contains an `sf` object. Since we are going to use a function (`spsample`) from the `sp` library, we must convert the `sf` object to a `SpatialPolygonsDataFrame` object using the `as` function.

Then we call the `spsample` function with this object and the `numpoints` input parameter (which specifies the number of points to generate).

Since we have declared a vector output named `Output`, we have to create a variable named `Output` containing an `sf` object.

We do this in two steps. First we create a `SpatialPolygonsDataFrame` object from the result of the function, using the `SpatialPointsDataFrame` function, and then we convert that object to an `sf` object using the `st_as_sf` function (of the `sf` library).

You can use whatever names you like for your intermediate variables. Just make sure that the variable storing your final result has the defined name (in this case `Output`), and that it contains a suitable value (an `sf` object for vector layer output).

In this case, the result obtained from the `spsample` method had to be converted explicitly into an `sf` object via a `SpatialPointsDataFrame` object, since it is itself an object of class `ppp`, which can not be returned to QGIS.

If your algorithm generates raster layers, the way they are saved will depend on whether or not you have used the `##dontuserasterpackage` option. If you have used it, layers are saved using the `writeGDAL()` method. If not, the `writeRaster()` method from the `raster` package will be used.

If you have used the `##pass_filenames` option, outputs are generated using the `raster` package (with `writeRaster()`).

If your algorithm does not generate a layer, but a text result in the console instead, you have to indicate that you want the console to be shown once the execution is finished. To do so, just start the command lines that produce the results you want to print with the `>` ('greater than') sign. Only output from lines prefixed with `>` are shown. For instance, here is the description file of an algorithm that performs a normality test on a given field (column) of the attributes of a vector layer:

```
##layer=vector
##field=field layer
##nortest=group
library(nortest)
>lillie.test(layer[[field]])
```

The output of the last line is printed, but the output of the first is not (and neither are the outputs from other command lines added automatically by QGIS).

If your algorithm creates any kind of graphics (using the `plot()` method), add the following line (`output_plots_to_html` used to be `showplots`):

```
##output_plots_to_html
```

This will cause QGIS to redirect all R graphical outputs to a temporary file, which will be opened once R execution has finished.

Both graphics and console results will be available through the processing results manager.

For more information, please check the R scripts in the official QGIS collection (you download and install them using the *QGIS Resource Sharing* plugin, as explained elsewhere). Most of them are rather simple and will greatly help you understand how to create your own scripts.

注解: The `sf`, `rgdal` and `raster` libraries are loaded by default, so you do not have to add the corresponding `library()` commands. However, other libraries that you might need have to be explicitly loaded by typing: `library(ggplot2)` (to load the `ggplot2` library). If the package is not already installed on your machine, Processing will try to download and install it. In this way the package will also become available in R Standalone. **Be aware** that if the package has to be downloaded, the script may take a long time to run the first time.

23.10.6 R libraries

The R script `sp_test` tries to load the R packages `sp` and `raster`.

R libraries installed when running `sf_test`

The R script `sf_test` tries to load `sf` and `raster`. If these two packages are not installed, R may try to load and install them (and all the libraries that they depend on).

The following R libraries end up in `~/.local/share/QGIS/QGIS3/profiles/default/processing/rscripts` after `sf_test` has been run from the Processing Toolbox on Ubuntu with version 2.0 of the *Processing R Provider* plugin and a fresh install of R 3.4.4 (*apt* package `r-base-core` only):

```
abind, askpass, assertthat, backports, base64enc, BH, bit, bit64, blob,
brew, callr, classInt, cli, colorspace, covr, crayon, crosstalk, curl,
DBI, deldir, desc, dichromat, digest, dplyr, e1071, ellipsis, evaluate,
fansI, farver, fastmap, gdtools, ggplot2, glue, goftest, gridExtra,
gtable, highr, hms, htmltools, htmlwidgets, httpuv, httr, jsonlite, knitr,
labeling, later, lazyeval, leafem, leaflet, leaflet.providers, leafpop,
leafsync, lifecycle, lwgeom, magrittr, maps, mapview, markdown, memoise,
microbenchmark, mime, munsell, odbc, openssl, pillar, pkgbuild, pkgconfig,
pkgload, plogr, plyr, png, polyclip, praise, prettyunits, processx,
promises, ps, purrr, R6, raster, RColorBrewer, Rcpp, reshape2, rex, rgeos,
rlang, rmarkdown, RPostgres, RPostgreSQL, rprojroot, RSQLite, rstudioapi,
satellite, scales, sf, shiny, sourcetools, sp, spatstat, spatstat.data,
spatstat.utils, stars, stringi, stringr, svglite, sys, systemfonts, tensor,
testthat, tibble, tidyselect, tinytex, units, utf8, uuid, vctrs, viridis,
viridisLite, webshot, withr, xfun, XML, xtable
```

23.10.7 GRASS

Configuring GRASS is not much different from configuring SAGA. First, the path to the GRASS folder has to be defined, but only if you are running Windows.

By default, the Processing framework tries to configure its GRASS connector to use the GRASS distribution that ships along with QGIS. This should work without problems for most systems, but if you experience problems, you might have to configure the GRASS connector manually. Also, if you want to use a different GRASS installation, you can change the setting to point to the folder where the other version is installed. GRASS 7 is needed for algorithms to work correctly.

If you are running Linux, you just have to make sure that GRASS is correctly installed, and that it can be run without problem from a terminal window.

GRASS algorithms use a region for calculations. This region can be defined manually using values similar to the ones found in the SAGA configuration, or automatically, taking the minimum extent that covers all the input layers used to execute the algorithm each time. If the latter approach is the behavior you prefer, just check the *Use min covering region* option in the GRASS configuration parameters.

23.10.8 激光雷达处理工具 (LAStools)

To use **LAStools** in QGIS, you need to download and install LAStools on your computer and install the LAStools plugin (available from the official repository) in QGIS.

On Linux platforms, you will need **Wine** to be able to run some of the tools.

LAStools is activated and configured in the Processing options (*Settings* ► *Options*, *Processing* tab, *Providers* ► *LAStools*), where you can specify the location of LAStools (*LAStools folder*) and Wine (*Wine folder*). On Ubuntu, the default Wine folder is `/usr/bin`.

23.10.9 OTB Applications

OTB applications are fully supported within the QGIS Processing framework.

OTB (Orfeo ToolBox) is an image processing library for remote sensing data. It also provides applications that provide image processing functionalities. The list of applications and their documentation are available in [OTB CookBook](#)

注解: Note that OTB is not distributed with QGIS and needs to be installed separately. Binary packages for OTB can be found on the [download page](#).

To configure QGIS processing to find the OTB library:

1. Open the processing settings: *Settings* ► *Options* ► *Processing* (left panel)*
2. You can see OTB under "Providers":
 1. Expand the *OTB* tab
 2. Tick the *Activate* option
 3. Set the *OTB folder*. This is the location of your OTB installation.
 4. Set the *OTB application folder*. This is the location of your OTB applications (`<PATH_TO_OTB_INSTALLATION>/lib/otb/applications`)
 5. Click "ok" to save the settings and close the dialog.

If settings are correct, OTB algorithms will be available in the *Processing Toolbox*.

Documentation of OTB settings available in QGIS Processing

- **Activate:** This is a checkbox to activate or deactivate the OTB provider. An invalid OTB setting will uncheck this when saved.
- **OTB folder:** This is the directory where OTB is available.
- **OTB application folder:** This is the location(s) of OTB applications.
Multiple paths are allowed.
- **Logger level** (optional): Level of logger to use by OTB applications.
The level of logging controls the amount of detail printed during algorithm execution. Possible values for logger level are INFO, WARNING, CRITICAL, DEBUG. This value is INFO by default. This is an advanced user configuration.
- **Maximum RAM to use** (optional): by default, OTB applications use all available system RAM.
You can, however, instruct OTB to use a specific amount of RAM (in MB) using this option. A value of 256 is ignored by the OTB processing provider. This is an advanced user configuration.

- **Geoid file** (optional): Path to the geoid file.

This option sets the value of the `elev.dem.geoid` and `elev.geoid` parameters in OTB applications. Setting this value globally enables users to share it across multiple processing algorithms. Empty by default.

- **SRTM tiles folder** (optional): Directory where SRTM tiles are available.

SRTM data can be stored locally to avoid downloading of files during processing. This option sets the value of `elev.dem.path` and `elev.dem` parameters in OTB applications. Setting this value globally enables users to share it across multiple processing algorithms. Empty by default.

Compatibility between QGIS and OTB versions

All OTB versions (from OTB 6.6.1) are compatible with the latest QGIS version.

Troubleshoot

If you have issues with OTB applications in QGIS Processing, please open an issue on the [OTB bug tracker](#), using the `qgis` label.

Additional information about OTB and QGIS can be found [here](#)

数据处理提供程序和算法

这里记录了处理算法及其参数 (如用户界面所示)。

24.1 QGIS algorithm provider

QGIS algorithm provider implements various analysis and geoprocessing operations using mostly only QGIS API. So almost all algorithms from this provider will work "out of the box" without any additional configuration.

This provider incorporates some algorithms from plugins and also adds its own algorithms.

24.1.1 Cartography

Align points to features

Calculates the rotation required to align point features with their nearest feature from another reference layer. A new field is added to the output layer which is filled with the angle (in degrees, clockwise) to the nearest reference feature.

Optionally, the output layer's symbology can be set to automatically use the calculated rotation field to rotate marker symbols. If desired, a maximum distance to use when aligning points can be set, to avoid aligning isolated points to distant features.

提示: This algorithm is designed for use cases like aligning building point symbols to follow the nearest road direction.

 Allows *features in-place modification*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Point features to calculate the rotation for
Reference layer	REFERENCE_LAYER	[vector: any]	Layer to find the closest feature from for rotation calculation
Maximum distance to consider Optional	MAX_DISTANCE	[number] Default: Not set	If no reference feature is found within this distance, no rotation is assigned to the point feature.
Angle field name	FIELD_NAME	[string] Default: 'rotation'	Field in which to store the rotation value.
Automatically apply symbology	APPLY_SYMOLOGY	[boolean] Default: True	Rotates the symbol marker of the features using the angle field value
Aligned layer	OUTPUT	[vector: point] Default: [Save to temporary file]	Specify the rotated output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Aligned layer	OUTPUT	[vector: point]	The point layer appended with a rotation field. If loaded to QGIS, it is applied by default the input layer symbology, with a data-defined rotation of its marker symbol.

Python code

Algorithm ID: native:angletonearest

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Combine style databases

Combines multiple QGIS style databases into a single style database. If items of the same type with the same name exist in different source databases these will be renamed to have unique names in the output combined database.

参见:

Create style database from project

Parameters

Label	名称	Type	Description
Input databases	INPUT	[file] [list]	Files containing QGIS style items
Objects to combine	OBJECTS	[enumeration] [list]	Types of style items in the input databases you would like to put in the new database. These can be: <ul style="list-style-type: none"> • 0 --- <i>Symbols</i> • 1 --- <i>Color ramps</i> • 2 --- <i>Text formats</i> • 3 --- <i>Label settings</i>
Output style database	OUTPUT	[file] Default: [Save to temporary file]	Output .XML file combining the selected style items. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Outputs

Label	名称	Type	Description
Color ramp count	COLORRAMPS	[number]	
Label settings count	LABELSETTINGS	[number]	
Output style database	OUTPUT	[file]	Output .XML file combining the selected style items
Symbol count	SYMBOLS	[number]	
Text format count	TEXTFORMATS	[number]	

Python code

Algorithm ID: native:combinestyles

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create categorized renderer from styles

Sets a vector layer's renderer to a categorized renderer using matching symbols from a style database. If no style file is specified, symbols from the user's current *symbol library* are used instead.

A specified expression or field is used to create categories for the renderer. Each category is individually matched to the symbols which exist within the specified QGIS XML style database. Whenever a matching symbol name is found, the category's symbol will be set to this matched symbol.

If desired, outputs can also be tables containing lists of the categories which could not be matched to symbols, and symbols which were not matched to categories.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Vector layer to apply a categorized style to
Categorize using expression	FIELD	[expression]	Field or expression to categorize the features
Style database (leave blank to use saved symbols)	STYLE	[file]	File (.XML) containing the symbols to apply to the input layer categories. The file can be obtained from the Style Manager <i>Share symbols</i> tool. If no file is specified, QGIS local symbols library is used.
Use case-sensitive match to symbol names	CASE_SENSITIVE	[boolean] Default: False	If True (checked), applies a case sensitive comparison between the categories and symbols names
Ignore non-alphanumeric characters while matching	TOLERANT	[boolean] Default: False	If True (checked), non-alphanumeric characters in the categories and symbols names will be ignored, allowing greater tolerance during the match.
Non-matching categories Optional	NON_MATCHING_CATEGORIES	[table] Default: [Skip output]	Output table for categories which do not match any symbol in the database. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... The file encoding can also be changed here.
Non-matching symbol names Optional	NON_MATCHING_SYMBOLS	[table] Default: [Skip output]	Output table for symbols from the provided style database which do not match any category. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Non-matching categories	NON_MATCHING_CATEGORIES	[table]	Lists categories which could not be matched to any symbol in the provided style database
Non-matching symbol names	NON_MATCHING_SYMBOLS	[table]	Lists symbols from the provided style database which could not match any category
Categorized layer	OUTPUT	[same as input]	The input vector layer with the categorized style applied. No new layer is output.

Python code

Algorithm ID: native:categorizeusingstyle

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create style database from project

Extracts all style objects (symbols, color ramps, text formats and label settings) from a QGIS project.

The extracted symbols are saved to a QGIS style database (XML format), which can be managed and imported via the *Style Manager* dialog.

参见:

Combine style databases

Parameters

Label	名称	Type	Description
Input project (leave blank to use current) Optional	INPUT	[file]	A QGIS project file to extract the style items from
Objects to extract	OBJECTS	[enumeration] [list]	Types of style items in the input project you would like to put in the new database. These can be: <ul style="list-style-type: none"> • 0 --- <i>Symbols</i> • 1 --- <i>Color ramps</i> • 2 --- <i>Text formats</i> • 3 --- <i>Label settings</i>
Output style database	OUTPUT	[file] Default: [Save to temporary file]	Specify the output .XML file for the selected style items. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Outputs

Label	名称	Type	Description
Color ramp count	COLORRAMPS	[number]	Number of color ramps
Label settings count	LABELSETTINGS	[number]	Number of label settings
Output style database	OUTPUT	[file]	Output .XML file for the selected style items
Symbol count	SYMBOLS	[number]	Number of symbols
Text format count	TEXTFORMATS	[number]	Number of text formats

Python code

Algorithm ID: native:stylefromproject

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Export atlas layout as image

Exports the atlas of a print layout as image files (e.g. PNG or JPEG images).

If a coverage layer is set, the selected layout's atlas settings exposed in this algorithm will be overwritten. In this case, an empty filter or sort by expression will turn those settings off.

Parameters

Basic parameters

Label	名称	Type	Description
Atlas layout	LAYOUT	[layout]	Layout to export
Coverage layer Optional	COVERAGE_LAYER	[vector: any]	Layer to use to generate the atlas
Filter expression	FILTER_EXPRESSION	[expression]	Expression to use to filter out atlas features
Sort expression Optional	SORTBY_EXPRESSION	[expression]	Expression to use to sort the atlas features
Reverse sort order Optional	SORTBY_REVERSE	[boolean]	Determines if sorting should be inverted. Used when a sort expression is provided.
Output filename expression	FILENAME_EXPRESSION	[expression] Default: 'output_#@atlas_featurenumber'	Expression for use to generate filenames
Output folder	FOLDER	[folder]	Destination folder where the images will be generated

Advanced parameters

Label	名称	Type	Description
Map layers to assign to unlocked map item(s) Optional	LAYERS	[enumeration] [layer]	Layers to display in the map item(s) whose contents are not locked
Image format	EXTENSION	[list] Default: png	File format of the generated output(s). The list of available formats varies depending on OS and installed drivers.
DPI Optional	DPI Default: Not set	[number]	DPI of the output file(s). If not set, the value in the print layout settings will be used.
Generate world file	GEOREFERENCE	[boolean] Default: True	Determines if a world file should be generated
Export RDF metadata	INCLUDE_METADATA	[boolean] Default: True	Determines if RDF metadata (title, author, ...) should be generated
Enable antialiasing	ANTIALIAS	[boolean] Default: True	Determines if antialiasing should be enabled

Outputs

Label	名称	Type	Description
Image file	OUTPUT	[file]	Image files generated by the atlas layout

Python code

Algorithm ID: native:atlaslayouttoimage

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Export atlas layout as PDF

Exports the atlas of a print layout as a PDF file(s).

If a coverage layer is set, the selected layout's atlas settings exposed in this algorithm will be overwritten. In this case, an empty filter or sort by expression will turn those settings off.

Parameters

Basic parameters

Label	名称	Type	Description
Atlas layout	LAYOUT	[layout]	Layout to export
Coverage layer Optional	COVERAGE_LAYER	[vector: any]	Layer to use to generate the atlas
Filter expression	FILTER_EXPRESSION	[expression]	Expression to use to filter out atlas features
Sort expression Optional	SORTBY_EXPRESSION	[expression]	Expression to use to sort the atlas features
Reverse sort order Optional	SORTBY_REVERSE	[boolean]	Determines if sorting should be inverted. Used when a sort expression is provided.

Advanced parameters

Label	名称	Type	Description
Map layers to assign to unlocked map item(s) Optional	LAYERS	[enumeration] [layer]	Layers to display in the map item(s) whose contents are not locked
DPI Optional	DPI Default: Not set	[number]	DPI of the output file(s). If not set, the value in the print layout settings will be used.
Always export as vectors	FORCE_VECTOR	[boolean] Default: False	Determines if vectorial data should be left as vectors
Append georeference information	GEOREFERENCE	[boolean] Default: True	Determines if a world file should be generated
Export RDF metadata	INCLUDE_METADATA	[boolean] Default: True	Determines if RDF metadata (title, author, ...) should be generated
Disable tiled raster layer exports	DISABLE_TILED	[boolean] Default: False	Determines if raster should be tiled
Simplify geometries to reduce output file size	SIMPLIFY	[boolean] Default: True	Determines if geometries should be simplified to reduce output file size
Text export	TEXT_FORMAT	[list] Default: 0	Determines if text should be exported as path or text objects. Possible options are: <ul style="list-style-type: none"> • 0 - Always export text as paths (recommended) • 1 - Always export texts as text objects
PDF file	OUTPUT	[file] Default: [Save to temporary file]	Name (including path) of the output file. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Outputs

Label	名称	Type	Description
PDF file	OUTPUT	[file]	PDF file corresponding to the exported atlas layout

Python code

Algorithm ID: native:atlaslayouttopdf

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Export print layout as image

Exports a print layout as an image file (e.g. PNG or JPEG images)

Parameters

Basic parameters

Label	名称	Type	Description
Print layout	LAYOUT	[layout]	Layout to export
Image file	OUTPUT	[file] Default: [Save to temporary file]	Name (including path) of the output file. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Map layers to assign to unlocked map item(s) Optional	LAYERS	[enumeration] [layer]	Layers to display in the map item(s) whose contents are not locked
DPI Optional	DPI Default: Not set	[number]	DPI of the output file(s). If not set, the value in the print layout settings will be used.
Generate world file	GEOREFERENCE	[boolean] Default: True	Determines if a world file should be generated
Export RDF metadata	INCLUDE_METADATA	[boolean] Default: True	Determines if RDF metadata (title, author, ...) should be generated
Enable antialiasing	ANTIALIAS	[boolean] Default: True	Determines if antialiasing should be enabled

Outputs

Label	名称	Type	Description
Image file	OUTPUT	[file]	Image file corresponding to the exported print layout

Python code

Algorithm ID: native:printlayouttoimage

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Export print layout as pdf

Exports a print layout as a PDF file.

Parameters

Basic parameters

Label	名称	Type	Description
Print Layout	LAYOUT	[layout]	Layout to export
PDF file	OUTPUT	[file] Default: [Save to temporary file]	Name (including path) of the output file. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Map layers to assign to unlocked map item(s) Optional	LAYERS	[enumeration] [layer]	Layers to display in the map item(s) whose contents are not locked
DPI Optional	DPI Default: Not set	[number]	DPI of the output file(s). If not set, the value in the print layout settings will be used.
Always export as vectors	FORCE_VECTOR	[boolean] Default: False	Determines if vectorial data should be left as vectors
Append georeference information	GEOREFERENCE	[boolean] Default: True	Determines if a world file should be generated
Export RDF metadata	INCLUDE_METADATA	[boolean] Default: True	Determines if RDF metadata (title, author, ...) should be generated
Disable tiled raster layer exports	DISABLE_TILED	[boolean] Default: False	Determines if raster should be tiled
Simplify geometries to reduce output file size	SIMPLIFY	[boolean] Default: True	Determines if geometries should be simplified to reduce output file size
Text export	TEXT_FORMAT	[list] Default: 0	Determines if text should be exported as path or text objects. Possible options are: <ul style="list-style-type: none"> • 0 - Always export text as paths (recommended) • 1 - Always export texts as text objects
Export layers as separate PDF files	SEPARATE_LAYERS	[boolean] Default: False	If True, then a separate PDF file will be created per layer per map item in the layout. Additionally, separate PDF files may be created for other complex layout items, resulting in a set of PDF files which contain logical atomic components of the layout.

Outputs

Label	名称	Type	Description
PDF file	OUTPUT	[file]	PDF file(s) corresponding to the exported print layout

Python code

Algorithm ID: native:printlayouttopdf

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Print layout map extent to layer

Creates a polygon layer containing the extent of a print layout map item (or items), with attributes specifying the map size (in layout units, i.e. the *reference map* units), scale and rotation.

If the map item parameter is specified, then only the matching map extent will be exported. If it is not specified, all map extents from the layout will be exported.

Optionally, a specific output CRS can be specified. If it is not specified, the original map item CRS will be used.

Parameters

Basic parameters

Label	名称	Type	Description
Print layout	LAYOUT	[enumeration]	A print layout in the current project
Map item Optional	MAP	[enumeration] Default: <i>All the map items</i>	The map item(s) whose information you want to extract. If none is provided then all the map items are processed.
Extent	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer for the extent(s). One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... The file encoding can also be changed here.

Advanced parameters

Label	名称	Type	Description
Override CRS Optional	CRS	[crs] Default: <i>The layout CRS</i>	Select the CRS for the layer in which the information will be reported.

Outputs

Label	名称	Type	Description
Map height	HEIGHT	[number]	
Extent	OUTPUT	[vector: polygon]	Output polygon vector layer containing extents of all the input layout map item(s)
Map rotation	ROTATION	[number]	
Map scale	SCALE	[number]	
Map width	WIDTH	[number]	

Python code

Algorithm ID: native:printlayoutmapextenttolayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Set layer style

Applies a provided style to a layer. The style must be defined in a QML file.

No new output are created: the style is immediately assigned to the layer.

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[layer]	Input layer you want to apply the style to
Style file	STYLE	[file]	Path to the .qml file of the style

Outputs

Label	名称	Type	Description
	OUTPUT	[same as input]	The input layer with the new style assigned. No new layer is created.

Python code

Algorithm ID: native:setlayerstyle

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Topological coloring

Assigns a color index to polygon features in such a way that no adjacent polygons share the same color index, whilst minimizing the number of colors required.

The algorithm allows choice of method to use when assigning colors.

A minimum number of colors can be specified if desired. The color index is saved to a new attribute named **color_id**.

The following example shows the algorithm with four different colors chosen; as you can see each color class has the same amount of features.

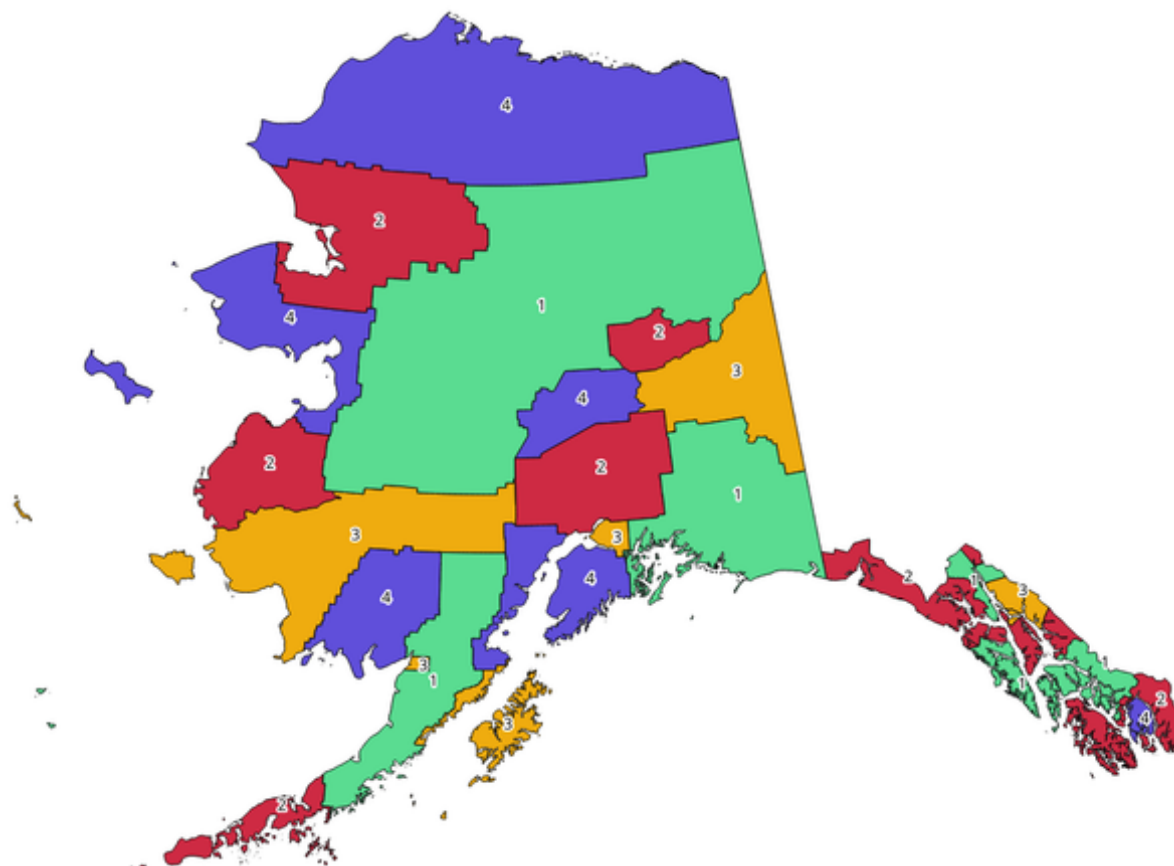


图 24.1: Topological colors example

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	The input polygon layer
Minimum number of colors	MIN_COLORS	[number] Default: 4	The minimum number of colors to assign. Minimum 1, maximum 1000.
Minimum distance between features	MIN_DISTANCE	[number] Default: 0.0	Prevent nearby (but non-touching) features from being assigned equal colors. Minimum 0.0.

下页继续

表 24.2 – 续上页

Label	名称	Type	Description
Balance color assignment	BALANCE	[enumeration] Default: 0	Options are: <ul style="list-style-type: none"> • 0 --- By feature count Attempts to assign colors so that the count of features assigned to each individual color index is balanced. • 1 --- By assigned area Assigns colors so that the total area of features assigned to each color is balanced. This mode can be useful to help avoid large features resulting in one of the colors appearing more dominant on a colored map. • 2 --- By distance between colors Assigns colors in order to maximize the distance between features of the same color. This mode helps to create a more uniform distribution of colors across a map.
Colored	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Colored	OUTPUT	[vector: polygon]	Polygon vector layer with an added <code>color_id</code> column

Python code

Algorithm ID: `qgis:topologicalcoloring`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.1.2 Database

Export to PostgreSQL

Exports a vector layer to a PostgreSQL database, creating a new relation. If a relation with the same name exists, it can be removed before the new relation is created. Prior to this a connection between QGIS and the PostgreSQL database has to be created (see eg *Creating a stored Connection*).

Parameters

Label	名称	Type	Description
Layer to import	INPUT	[vector: any]	Vector layer to add to the database
Database (connection name)	DATABASE	[string]	Name of the database connection (not the database name). Existing connections will be shown in the combobox.
Schema (schema name) Optional	SCHEMA	[string] Default: 'public'	Name of the schema to store the data. It can be a new one or already exist.
Table to import to (leave blank to use layer name) Optional	TABLENAME	[string] Default: ''	Defines a table name for the imported vector file. If nothing is added, the layer name will be used.
Primary key field Optional	PRIMARY_KEY	[tablefield: any]	Sets the primary key field from an existing field in the vector layer. A column with unique values can be used as Primary key for the database.
Geometry column	GEOMETRY_COLUMN	[string] Default: 'geom'	Defines the name of the geometry column in the new PostGIS table. Geometry information for the features is stored in this column.
Encoding Optional	ENCODING	[string] Default: 'UTF-8'	Defines the encoding of the output layer
Overwrite	OVERWRITE	[boolean] Default: True	If the specified table exists, setting this option to <code>True</code> will make sure that it is deleted and a new table will be created before the features are added. If this option is <code>False</code> and the table exists, the algorithm will throw an exception ("relation already exists").
Create spatial index	CREATEINDEX	[boolean] Default: True	Specifies whether to create a spatial index or not
Convert field names to lowercase	LOWERCASE_NAMES	[boolean] Default: True	Converts the field names of the input vector layer to lowercase
Drop length constraint on character fields	DROP_STRING_LENGTH	[boolean] Default: False	Should length constraints on character fields be dropped or not
Create single-part geometries instead of multi-part	FORCE_SINGLEPART	[boolean] Default: False	Should the features of the output layer be single-part instead of multi-part. By default the existing geometries information are preserved.

Outputs

The algorithm has no output.

Python code

Algorithm ID: qgis:importintopostgis

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Export to Spatialite

Exports a vector layer to a Spatialite database. Prior to this a connection between QGIS and the Spatialite database has to be created (see eg *Spatialite Layers*).

Parameters

Label	名称	Type	Description
Layer to import	INPUT	[vector: any]	Vector layer to add to the database
File database	DATABASE	[vector: any]	The SQLite/Spatialite database file to connect to
Table to import to (leave blank to use layer name) Optional	TABLENAME	[string] Default: ""	Defines the table name for the imported vector file. If nothing is specified, the layer name will be used.
Primary key field Optional	PRIMARY_KEY	[tablefield: any]	Use a field in the input vector layer as the primary key
Geometry column	GEOMETRY_COLUMN	[string] Default: 'geom'	Defines the name of the geometry column in the new Spatialite table. Geometry information for the features is stored in this column.
Encoding Optional	ENCODING	[string] Default: 'UTF-8'	Defines the encoding of the output layer
Overwrite	OVERWRITE	[boolean] Default: True	If the specified table exists, setting this option to <code>True</code> will make sure that it is deleted and a new table will be created before the features of the layer is added. If this option is <code>False</code> and the table exists, the algorithm will throw an exception ("table already exists").
Create spatial index	CREATEINDEX	[boolean] Default: True	Specifies whether to create a spatial index or not
Convert field names to lowercase	LOWERCASE_NAMES	[boolean] Default: True	Convert the field names of the input vector layer to lowercase
Drop length constraint on character fields	DROP_STRING_LENGTH	[boolean] Default: False	Should length constraints on character fields be dropped or not

下页继续

表 24.4 - 续上页

Label	名称	Type	Description
Create single-part geometries instead of multi-part	FORCE_SINGLEPART	[boolean] Default: False	Should the features of the output layer be single-part instead of multi-part. By default the existing geometries information are preserved.

Outputs

The algorithm has no output.

Python code

Algorithm ID: qgis:importintospatialite

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Package layers

Adds layers to a GeoPackage.

If the GeoPackage exists and `Overwrite existing GeoPackage` is checked, it will be overwritten (removed and recreated). If the GeoPackage exists and `Overwrite existing GeoPackage` is not checked, the layer will be appended.

Parameters

Label	名称	Type	Description
Input layers	LAYERS	[vector: any] [list]	The (vector) layers to import into the GeoPackage. Raster layers are not supported. If a raster layer is added, a <code>QgsProcessingException</code> will be thrown.
Overwrite existing GeoPackage	OVERWRITE	[boolean] Default: False	If the specified GeoPackage exists, setting this option to <code>True</code> will make sure that it is deleted and a new one will be created before the layers are added. If set to <code>False</code> , layers will be appended.
Save layer styles into GeoPackage	SAVE_STYLES	[boolean] Default: True	Save the layer styles
Destination GeoPackage	OUTPUT	[file] Default: [Save to temporary file]	Specify where to store the GeoPackage file. One of <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Outputs

Label	名称	Type	Description
Layers within new package	OUTPUT_LAYERS	[string] [list]	The list of layers added to the GeoPackage.

Python code

Algorithm ID: native:package

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

PostgreSQL execute and load SQL

Allows a SQL database query to be performed on a PostgreSQL database connected to QGIS and loads the result. The algorithm **won't** create a new layer: it is designed to run queries on the layer itself.

Example

1. Set all the values of an existing field to a fixed value. The SQL query string will be:

```
UPDATE your_table SET field_to_update=20;
```

In the example above, the values of the field `field_to_update` of the table `your_table` will be all set to 20.

2. Create a new area column and calculate the area of each feature with the `ST_AREA` PostGIS function.

```
-- Create the new column "area" on the table your_table"
ALTER TABLE your_table ADD COLUMN area double precision;
-- Update the "area" column and calculate the area of each feature:
UPDATE your_table SET area=ST_AREA(geom);
```

参见:

[PostgreSQL execute SQL](#), [Execute SQL](#), [Spatialite execute SQL](#)

Parameters

Label	名称	Type	Description
Database (connection name)	DATABASE	[string]	The database connection (not the database name). Existing connections will be shown in the combobox.
SQL query	SQL	[string]	Defines the SQL query, for example 'UPDATE my_table SET field=10'.
Unique ID field name	ID_FIELD	[string] Default: id	Sets the primary key field (a column in the result table)
Geometry field name Optional	GEOMETRY_FIELD	[string] Default: 'geom'	Name of the geometry column (a column in the result table)

Outputs

Label	名称	Type	Description
SQL layer	OUTPUT	[vector: any]	The resulting vector layer to be loaded into QGIS.

Python code

Algorithm ID: qgis:postgisexecuteandloadsql

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

PostgreSQL execute SQL

Allows a SQL database query to be performed on a PostgreSQL database connected to QGIS. The algorithm **won't** create a new layer: it is designed to run queries on the layer itself.

Example

1. Set all the values of an existing field to a fixed value. The SQL query string will be:

```
UPDATE your_table SET field_to_update=20;
```

In the example above, the values of the field `field_to_update` of the table `your_table` will be all set to 20.

2. Create a new area column and calculate the area of each feature with the `ST_AREA` PostGIS function.

```
-- Create the new column "area" on the table your_table"
ALTER TABLE your_table ADD COLUMN area double precision;
-- Update the "area" column and calculate the area of each feature:
UPDATE your_table SET area=ST_AREA(geom);
```

参见:

PostgreSQL execute and load SQL, Execute SQL, SpatialLite execute SQL

Parameters

Label	名称	Type	Description
Database (connection name)	DATABASE	[string]	The database connection (not the database name). Existing connections will be shown in the combobox.
SQL query	SQL	[string]	Defines the SQL query, for example 'UPDATE my_table SET field=10'.

Outputs

No output is created. The SQL query is executed in place.

Python code

Algorithm ID: native:postgisexecutesql

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Spatialite execute SQL

Allows a SQL database query to be performed on a Spatialite database. The algorithm **won't** create a new layer: it is designed to run queries on the layer itself.

参见:

PostgreSQL execute SQL, Execute SQL

For some SQL query examples see *PostGIS SQL Query Examples*.

Parameters

Label	名称	Type	Description
File Database	DATABASE	[vector]	The SQLite/Spatialite database file to connect to
SQL query	SQL	[string] Default: ""	Defines the SQL query, for example 'UPDATE my_table SET field=10'.

Outputs

No output is created. The SQL query is executed in place.

Python code

Algorithm ID: native:spatialiteexecutesql

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Spatialite execute SQL (registered DB)

Allows a SQL database query to be performed on a Spatialite database connected to QGIS. The algorithm **won't** create a new layer: it is designed to run queries on the layer itself.

参见:

PostgreSQL execute SQL, Execute SQL

For some SQL query examples see *PostGIS SQL Query Examples*.

Parameters

Label	名称	Type	Description
Database	DATABASE	[enumeration] Default: not set	Select a SQLite/Spatialite database connected to the current session
SQL query	SQL	[string] Default: ""	Defines the SQL query, for example 'UPDATE my_table SET field=10'.

Outputs

No output is created. The SQL query is executed in place.

Python code

Algorithm ID: native:spatialiteexecutesqlregistered

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.3 File tools

Download file

Downloads a file specified using a URL (using for instance `http:` or `file:`). In other words you can copy/paste a URL and download the file.

Parameters

Label	名称	Type	Description
URL	URL	[string]	The URL of the file to download.
File destination	OUTPUT	[string] Default: [Save to temporary file]	Specification of the file destination. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
File destination	OUTPUT	[string]	The location of the downloaded file

Python code

Algorithm ID: qgis:filedownloader

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.4 Interpolation

Heatmap (kernel density estimation)

Creates a density (heatmap) raster of an input point vector layer using kernel density estimation.

The density is calculated based on the number of points in a location, with larger numbers of clustered points resulting in larger values. Heatmaps allow easy identification of *hotspots* and clustering of points.

Parameters

Label	名称	Type	Description
Point layer	INPUT	[vector: point]	Point vector layer to use for the heatmap
Radius	RADIUS	[number] Default: 100.0	Heatmap search radius (or kernel bandwidth) in map units. The radius specifies the distance around a point at which the influence of the point will be felt. Larger values result in greater smoothing, but smaller values may show finer details and variation in point density.
Output raster size	PIXEL_SIZE	[number] Default: 0.1	Pixel size of the output raster layer in layer units. In the GUI, the size can be specified by the number of rows (Number of rows) / columns (Number of columns) or the pixel size (Pixel Size X / Pixel Size Y). Increasing the number of rows or columns will decrease the cell size and increase the file size of the output raster. The values in Rows, Columns, Pixel Size X and Pixel Size Y will be updated simultaneously - doubling the number of rows will double the number of columns, and the cell size will be halved. The extent of the output raster will remain the same (approximately).

下页继续

表 24.5 – 续上页

Label	名称	Type	Description
Radius from field Optional	RADIUS_FIELD	[tablefield: numeric]	Sets the search radius for each feature from an attribute field in the input layer.
Weight from field Optional	WEIGHT_FIELD	[tablefield: numeric]	Allows input features to be weighted by an attribute field. This can be used to increase the influence certain features have on the resultant heatmap.
Kernel shape	KERNEL	[enumeration] Default: 0	Controls the rate at which the influence of a point decreases as the distance from the point increases. Different kernels decay at different rates, so a triweight kernel gives features greater weight for distances closer to the point than the Epanechnikov kernel does. Consequently, triweight results in “sharper” hotspots and Epanechnikov results in “smoother” hotspots. There are many shapes available (please see the Wikipedia page for further information): <ul style="list-style-type: none"> • 0 --- Quartic • 1 --- Triangular • 2 --- Uniform • 3 --- Triweight • 4 --- Epanechnikov
Decay ratio (Triangular kernels only) Optional	DECAY	[number] Default: 0.0	Can be used with Triangular kernels to further control how heat from a feature decreases with distance from the feature. <ul style="list-style-type: none"> • A value of 0 (=minimum) indicates that the heat will be concentrated in the center of the given radius and completely extinguished at the edge. • A value of 0.5 indicates that pixels at the edge of the radius will be given half the heat as pixels at the center of the search radius. • A value of 1 means the heat is spread evenly over the whole search radius circle. (This is equivalent to the ‘Uniform’ kernel.) • A value greater than 1 indicates that the heat is higher towards the edge of the search radius than at the center.
Output value scaling	OUTPUT_VALUE	[enumeration] Default: Raw	Allow to change the values of the output heatmap raster. One of: <ul style="list-style-type: none"> • 0 --- Raw • 1 --- Scaled
Heatmap	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer with kernel density values. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Heatmap	OUTPUT	[raster]	Raster layer with kernel density values

Example: Creating a Heatmap

For the following example, we will use the `airports` vector point layer from the QGIS sample dataset (see [下载示例数据](#)). Another excellent QGIS tutorial on making heatmaps can be found at <http://qgistutorials.com>.

In [图 24.2](#), the airports of Alaska are shown.

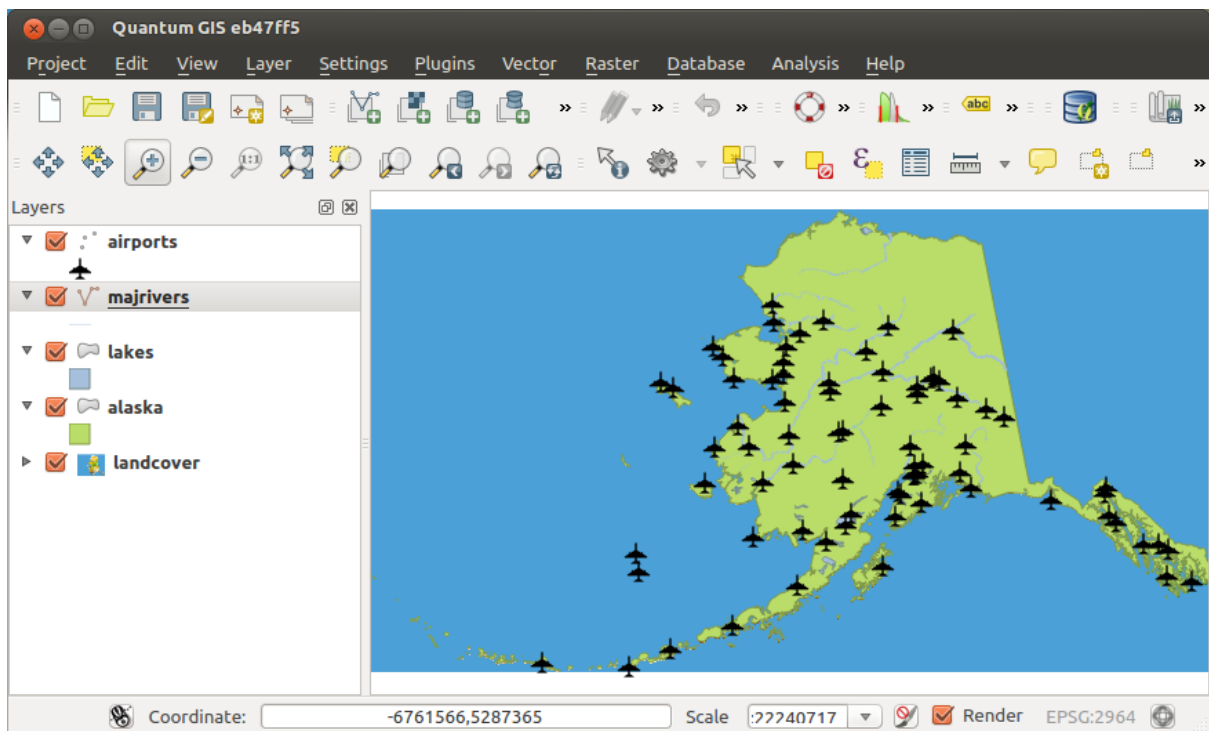


图 24.2: Airports of Alaska

1. Open the *Heatmap (Kernel Density Estimation)* algorithm from the QGIS *Interpolation* group
2. In the *Point layer* field, select `airports` from the list of point layers loaded in the current project.
3. Change the *Radius* to 1000000 meters.
4. Change the *Pixel size X* to 1000. The *Pixel size Y*, *Rows* and *Columns* will be automatically updated.
5. Click on *Run* to create and load the airports heatmap (see [图 24.4](#)).

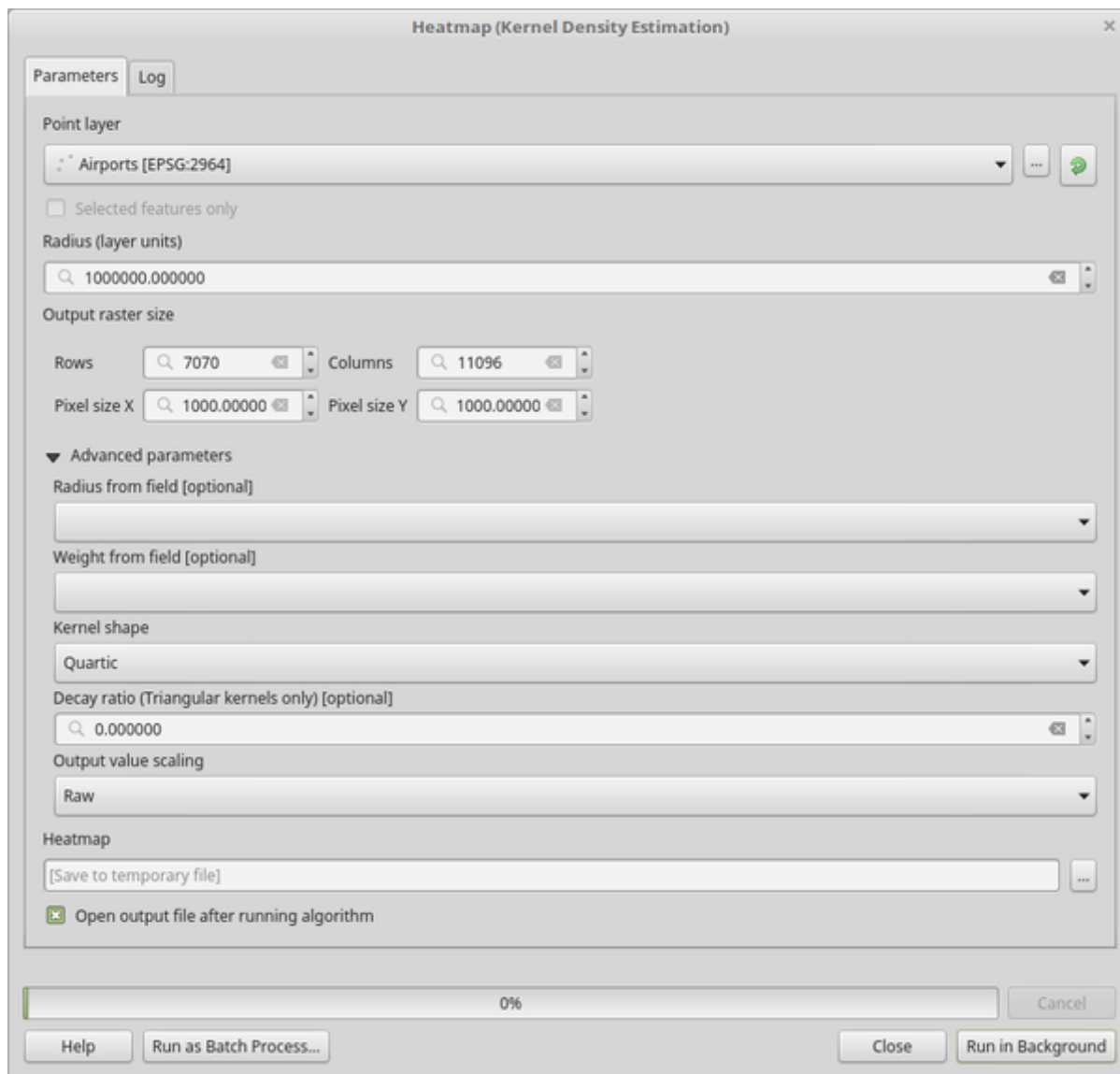


图 24.3: The Heatmap Dialog

QGIS will generate the heatmap and add it to your map window. By default, the heatmap is shaded in greyscale, with lighter areas showing higher concentrations of airports. The heatmap can now be styled in QGIS to improve its appearance.

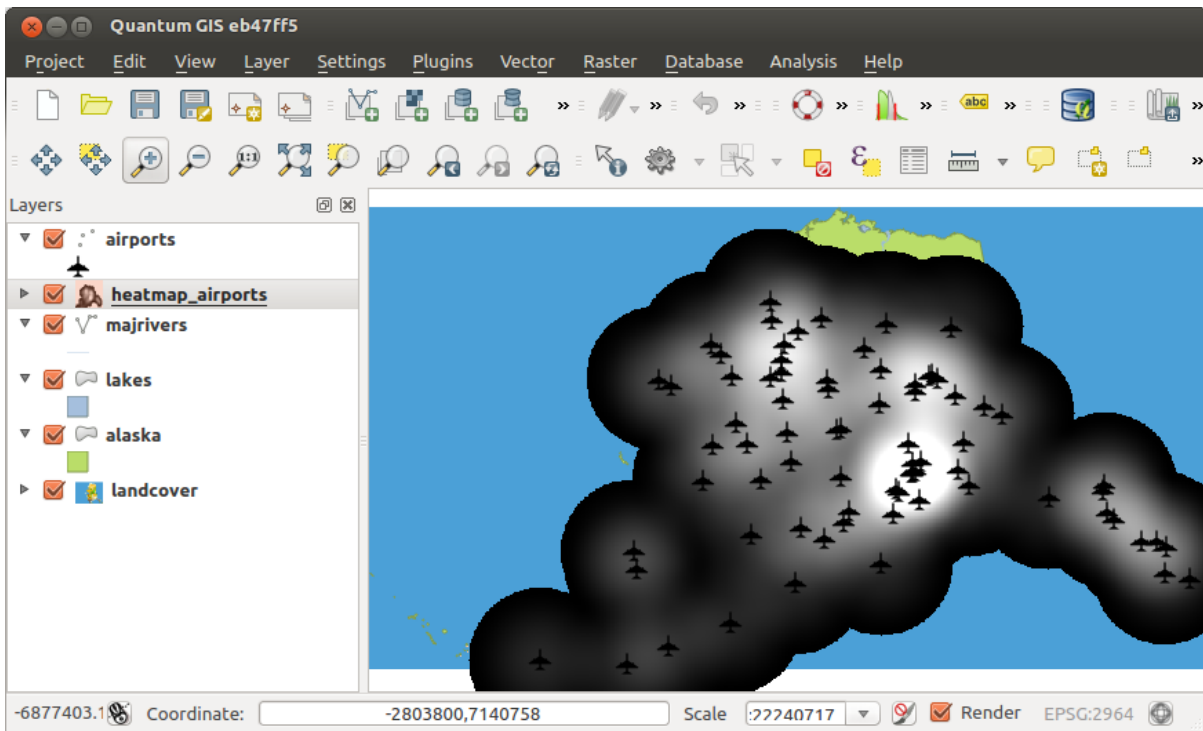


图 24.4: The heatmap after loading looks like a grey surface

1. Open the properties dialog of the `heatmap_airports` layer (select the layer `heatmap_airports`, open the context menu with the right mouse button and select *Properties*).
2. Select the *Symbology* tab.
3. Change the *Render type* to 'Singleband pseudocolor'.
4. Select a suitable *Color ramp* , for instance YlOrRd.
5. Click the *Classify* button.
6. Press *OK* to update the layer.

The final result is shown in 图 24.5.

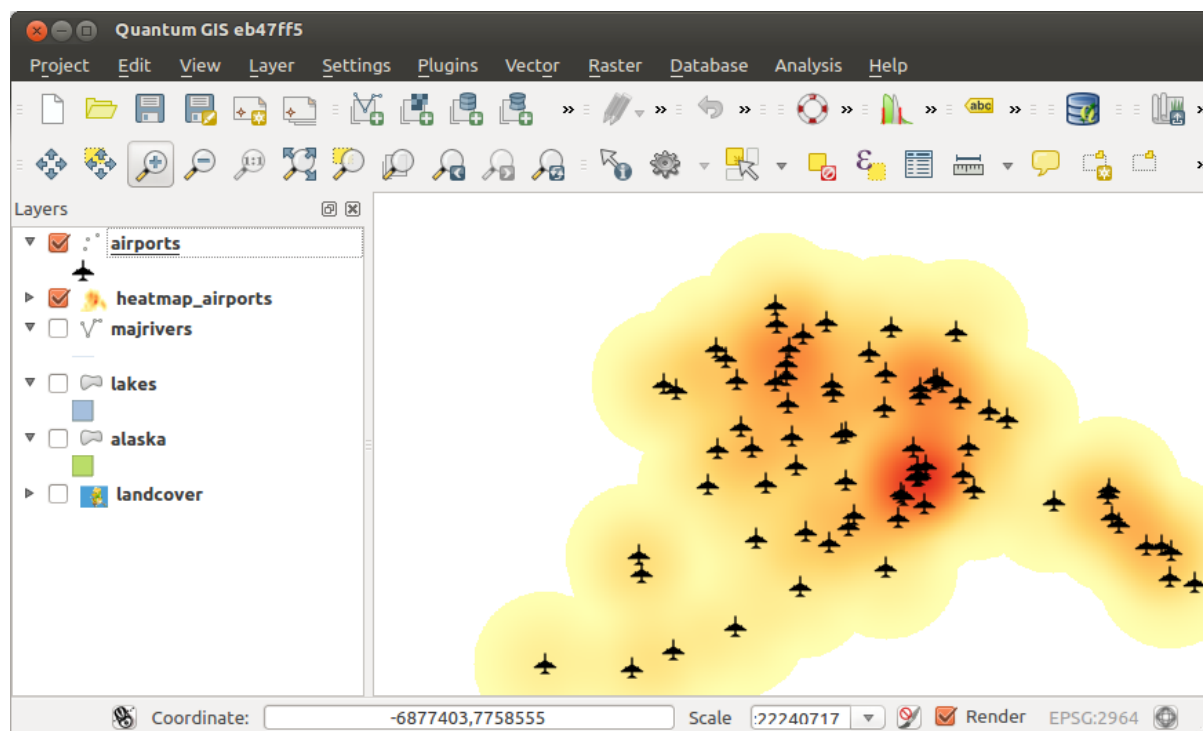


图 24.5: Styled heatmap of airports of Alaska

Python code

Algorithm ID: `qgis:heatmapkerneldensityestimation`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

IDW Interpolation

Generates an Inverse Distance Weighted (IDW) interpolation of a point vector layer.

Sample points are weighted during interpolation such that the influence of one point relative to another declines with distance from the unknown point you want to create.

The IDW interpolation method also has some disadvantages: the quality of the interpolation result can decrease, if the distribution of sample data points is uneven.

Furthermore, maximum and minimum values in the interpolated surface can only occur at sample data points.

Parameters

Label	名称	Type	Description
Input layer(s)	INTERPOLATION_DATA	[string]	<p>Vector layer(s) and field(s) to use for the interpolation, coded in a string (see the <code>ParameterInterpolationData</code> class in <code>InterpolationWidgets</code> for more details).</p> <p>The following GUI elements are provided to compose the interpolation data string:</p> <ul style="list-style-type: none"> • Vector layer [vector: any] • Interpolation attribute [tablefield: numeric]: Attribute to use in the interpolation • Use Z-coordinate for interpolation [boolean]: Uses the layer's stored Z values (Default: False) <p>For each of the added layer-field combinations, a type can be chosen:</p> <ul style="list-style-type: none"> • <i>Points</i> • <i>Structured lines</i> • <i>Break lines</i> <p>In the string, the layer-field elements are separated by ' : : : : '. The sub-elements of the layer-field elements are separated by ' : : ~ : : '.</p>
Distance coefficient P	DISTANCE_COEFFICIENT	[number] Default: 2.0	Sets the distance coefficient for the interpolation. Minimum: 0.0, maximum: 100.0.
Extent (xmin, xmax, ymin, ymax)	EXTENT	[extent]	Extent of the output raster layer. You have to declare the output extent by either choosing it from the map canvas, selecting it from another layer or type it manually.
Output raster size	PIXEL_SIZE	[number] Default: 0.1	<p>Pixel size of the output raster layer in layer units.</p> <p>In the GUI, the size can be specified by the number of rows (<code>Number of rows</code>) / columns (<code>Number of columns</code>) or the pixel size (<code>Pixel Size X</code> / <code>Pixel Size Y</code>). Increasing the number of rows or columns will decrease the cell size and increase the file size of the output raster. The values in <code>Rows</code>, <code>Columns</code>, <code>Pixel Size X</code> and <code>Pixel Size Y</code> will be updated simultaneously - doubling the number of rows will double the number of columns, and the cell size will be halved. The extent of the output raster will remain the same (approximately).</p>
Interpolated	OUTPUT	[raster] Default: [Save to temporary file]	<p>Raster layer of interpolated values. One of:</p> <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... <p>The file encoding can also be changed here.</p>

Outputs

Label	名称	Type	Description
Interpolated	OUTPUT	[raster]	Raster layer of interpolated values

Python code

Algorithm ID: qgis:ldwinterpolation

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Line Density

Calculates for each raster cell, the density measure of linear features within a circular neighbourhood. This measure is obtained by summing all the line segments intersecting the circular neighbourhood and dividing this sum by the area of such neighbourhood. A weighting factor can be applied to the line segments.

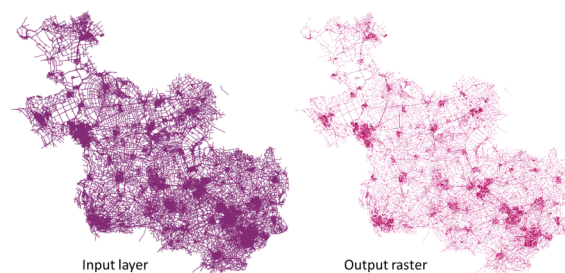


图 24.6: Line density example. Input layer source: Roads Overijssel - The Netherlands (OSM).

Parameters

Label	名称	Type	Description
Input line layer	INPUT	[vector: any]	Input vector layer containing line features
Weight field	WEIGHT	[number]	Field of the layer containing the weight factor to use during the calculation
Search Radius	RADIUS	[number] Default: 10	Radius of the circular neighbourhood. Units can be specified here.
Pixel size	PIXEL_SIZE	[number] Default: 10	Pixel size of the output raster layer in layer units. The raster has square pixels.
Line density raster	OUTPUT	[raster] Default: [Save to temporary file]	The output as a raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Line density raster	OUTPUT	[raster]	The output line density raster layer.

Python code

Algorithm ID: native:linedensity

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

TIN Interpolation

Generates a Triangulated Irregular Network (TIN) interpolation of a point vector layer.

With the TIN method you can create a surface formed by triangles of nearest neighbor points. To do this, circumcircles around selected sample points are created and their intersections are connected to a network of non overlapping and as compact as possible triangles. The resulting surfaces are not smooth.

The algorithm creates both the raster layer of the interpolated values and the vector line layer with the triangulation boundaries.

Parameters

Label	名称	Type	Description
Input layer(s)	INTERPOLATION_DATA	[string]	<p>Vector layer(s) and field(s) to use for the interpolation, coded in a string (see the <code>ParameterInterpolationData</code> class in InterpolationWidgets for more details).</p> <p>The following GUI elements are provided to compose the interpolation data string:</p> <ul style="list-style-type: none"> • Vector layer [vector: any] • Interpolation attribute [tablefield: numeric]: Attribute to use in the interpolation • Use Z-coordinate for interpolation [boolean]: Uses the layer's stored Z values (Default: False) <p>For each of the added layer-field combinations, a type can be chosen:</p> <ul style="list-style-type: none"> • <i>Points</i> • <i>Structured lines</i> • <i>Break lines</i> <p>In the string, the layer-field elements are separated by ' : : : : '. The sub-elements of the layer-field elements are separated by ' : : ~ : : '.</p>

下页继续

表 24.11 - 续上页

Label	名称	Type	Description
Interpolation method	METHOD	[enumeration] Default: 0	Set the interpolation method to be used. One of: <ul style="list-style-type: none"> • <i>Linear</i> • <i>Clough-Toucher (cubic)</i>
Extent (xmin, xmax, ymin, ymax)	EXTENT	[extent]	Extent of the output raster layer. You have to declare the output extent by either choosing it from the map canvas, selecting it from another layer or type it manually.
Output raster size	PIXEL_SIZE	[number] Default: 0.1	Pixel size of the output raster layer in layer units. In the GUI, the size can be specified by the number of rows (Number of rows) / columns (Number of columns) or the pixel size (Pixel Size X / Pixel Size Y). Increasing the number of rows or columns will decrease the cell size and increase the file size of the output raster. The values in Rows, Columns, Pixel Size X and Pixel Size Y will be updated simultaneously - doubling the number of rows will double the number of columns, and the cell size will be halved. The extent of the output raster will remain the same (approximately).
Interpolated	OUTPUT	[raster] Default: [Save to temporary file]	The output TIN interpolation as a raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.
Triangulation	TRIANGULATION	[vector: line] Default: [Skip output]	The output TIN as a vector layer. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table...

Outputs

Label	名称	Type	Description
Interpolated	OUTPUT	[raster]	The output TIN interpolation as a raster layer
Triangulation	TRIANGULATION	[vector: line]	The output TIN as a vector layer.

Python code

Algorithm ID: qgis:tininterpolation

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.5 Layer tools

Extract layer extent

Generates a vector layer with the minimum bounding box (rectangle with N-S orientation) that covers all the input features.

The output layer contains a single bounding box for the whole input layer.

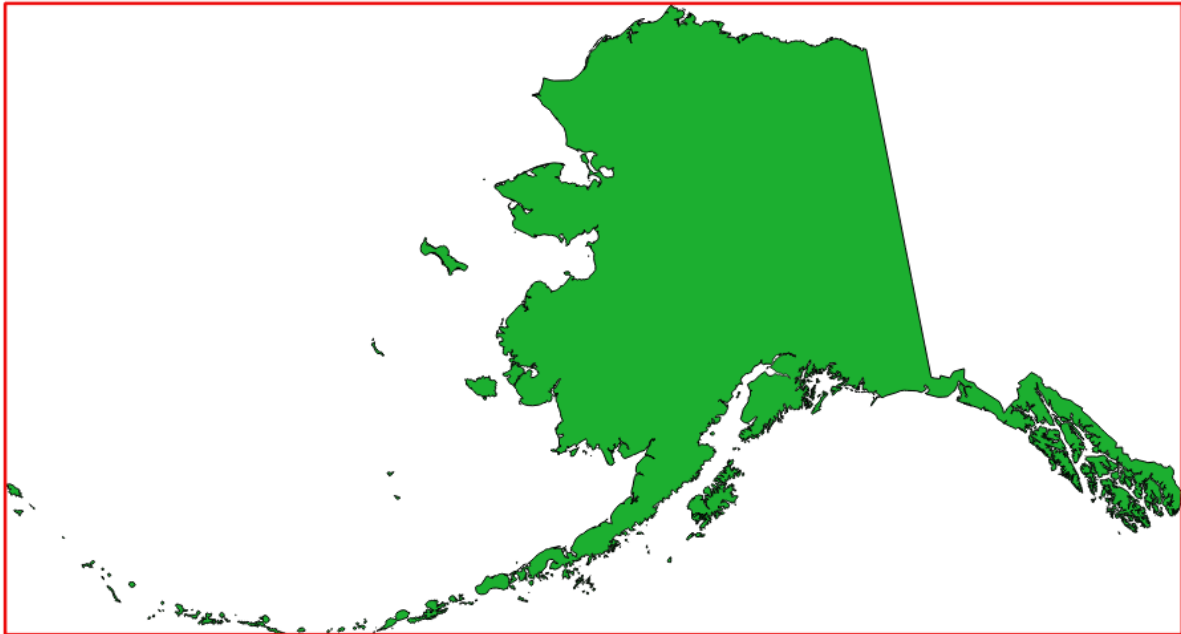


图 24.7: In red the bounding box of the source layer

Default menu: *Vector ► Research Tools*

Parameters

Label	名称	Type	Description
Layer	INPUT	[layer]	Input layer
Extent	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the polygon vector layer for the output extent. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Extent	OUTPUT	[vector: polygon]	Output (polygon) vector layer with the extent (minimum bounding box)

Python code

Algorithm ID: qgis:polygonfromlayerextent

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.6 Modeler tools

These tools are only available in the Graphical Modeler. They are not available in the Processing Toolbox.

Conditional branch

Adds a conditional branch into a model, allowing parts of the model to be executed based on the result of an expression evaluation. Mostly by using tool dependencies to control the flow of a model.

Parameters

Label	名称	Type	Description
Field	BRANCH	[string]	Name of the condition
Field	CONDITION	[expression]	Expression to evaluate

Outputs

None.

Python code

Algorithm ID: native:condition

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Load layer into project

Loads a layer to the current project.

Parameters

Label	名称	Type	Description
Layer	INPUT	[layer]	Layer to load in the legend
Loaded name	layer NAME	[string]	Name of the loaded layer

Outputs

Label	名称	Type	Description
Layer	OUTPUT	[same as input]	The (renamed) loaded layer

Python code

Algorithm ID: qgis:loadlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raise exception

Raises an exception and cancels a model's execution. The exception message can be customized, and optionally an expression based condition can be specified. If an expression condition is used, then the exception will only be raised if the expression result is true. A false result indicates that no exception will be raised, and the model execution can continue uninterrupted.

Parameters

Label	名称	Type	Description
Message	MESSAGE	[string]	Message to display
Condition	CONDITION	[expression]	Expression to evaluate if true

Outputs

None.

Python code

Algorithm ID: native:raiseexception

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Raise warning

Raises a warning message in the log. The warning message can be customized, and optionally an expression based condition can be specified. If an expression condition is used, then the warning will only be logged if the expression result is true. A false result indicates that no warning will be logged.

Parameters

Label	名称	Type	Description
Message	MESSAGE	[string]	Message to display
Condition	CONDITION	[expression]	Expression to evaluate if true

Outputs

None.

Python code

Algorithm ID: native:raisewarning

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Rename layer

Renames a layer.

Parameters

Label	名称	Type	Description
Layer	INPUT	[layer]	Layer to rename
New name	NAME	[string]	The new name of the layer

Outputs

Label	名称	Type	Description
Layer	OUTPUT	[same as input]	The (renamed) output layer

Python code

Algorithm ID: native:renamelayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Save log to file

Saves the model's execution log to a file. Optionally, the log can be saved in a HTML formatted version.

Parameters

Label	名称	Type	Description
Use HTML	USE_HTML	[Boolean] Default: False	Use HTML formatting

Outputs

Label	名称	Type	Description
File	OUTPUT	[string]	Destination of the log

Python code

Algorithm ID: native:savelog

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Set project variable

Sets an expression variable for the current project.

Parameters

Label	名称	Type	Description
Variable name	NAME	[string]	Name of the variable
Variable value	VALUE	[string]	Value to be stored

Outputs

None.

Python code

Algorithm ID: native:setprojectvariable

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

String concatenation

Concatenates two strings into a single one in the Processing Modeler.

Parameters

Label	名称	Type	Description
Input 1	INPUT_1	[string]	First string
Input 2	INPUT_2	[string]	Second string

Outputs

Label	名称	Type	Description
Concatenation	CONCATENATION	[string]	The concatenated string

Python code

Algorithm ID: qgis:stringconcatenation

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.1.7 Network analysis

Service area (from layer)

Returns all the edges or parts of edges of a network that can be reached within a distance or a time, starting from a point layer. This allows evaluation of accessibility within a network, e.g. what are the places I can navigate to on a road network without spending cost greater than a given value (the cost can be distance or time).

Parameters

Label	名称	Type	Description
Vector layer representing network	INPUT	[vector: line]	Line vector layer representing the network to be covered
Vector layer with start points	START_POINTS	[vector: point]	Point vector layer whose features are used as start points to generate the service areas
Path type to calculate	STRATEGY	[enumeration] Default: 0	The type of path to calculate. One of: <ul style="list-style-type: none"> 0 --- Shortest 1 --- Fastest
Travel cost (distance for "Shortest", time for "Fastest")	TRAVEL_COST	[number] Default: 0	The value is estimated as a distance (in the network layer units) when looking for the <i>Shortest</i> path and as time (in hours) for the <i>Fastest</i> path.

Direction field Optional	DIRECTION_FIELD	[tablefield: string] Default: 0.0	The field used to specify directions for the network edges. The values used in this field are specified with the three parameters Value for forward direction, Value for backward direction and Value for both directions. Forward and reverse directions correspond to a one-way edge, "both directions" indicates a two-way edge. If a feature does not have a value in this field, or no field is set then the default direction setting (provided with the Default direction parameter) is used.
Value for forward direction Optional	VALUE_FORWARD	[string] Default: " (empty string)	Value set in the direction field to identify edges with a forward direction
Value for backward direction Optional	VALUE_BACKWARD	[string] Default: " (empty string)	Value set in the direction field to identify edges with a backward direction
Value for both directions Optional	VALUE_BOTH	[string] Default: " (empty string)	Value set in the direction field to identify bidirectional edges
Default direction Optional	DEFAULT_DIRECTION	[enumeration] Default: 2	If a feature has no value set in the direction field or if no direction field is set, then this direction value is used. One of: <ul style="list-style-type: none"> • 0 --- Forward direction • 1 --- Backward direction • 2 --- Both directions
Speed field Optional	SPEED_FIELD	[tablefield: string]	Field providing the speed value (in km/h) for the edges of the network when looking for the fastest path. If a feature does not have a value in this field, or no field is set then the default speed value (provided with the Default speed parameter) is used.
Default speed (km/h) Optional	DEFAULT_SPEED	[number] Default: 50.0	Value to use to calculate the travel time if no speed field is provided for an edge
Topology tolerance Optional	TOLERANCE	[number] Default: 0.0	Two lines with nodes closer than the specified tolerance are considered connected
Include upper/lower bound points	INCLUDE_BOUNDS	[boolean] Default: False	Creates a point layer output with two points for each edge at the boundaries of the service area. One point is the start of that edge, the other is the end.

下页继续

表 24.15 - 续上页

Service area (lines)	OUTPUT_LINES	[vector: line] Default: [Create temporary layer]	Specify the output line layer for the service area. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Service area (boundary nodes)	OUTPUT	[vector: point] Default: [Skip output]	Specify the output point layer for the service area boundary nodes. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Service area (boundary nodes)	OUTPUT	[vector: point]	The output point layer with the service area boundary nodes.
Service area (lines)	OUTPUT_LINES	[vector: line]	Line layer representing the parts of the network that can be serviced by the start points, for the given cost.

Python code

Algorithm ID: `qgis:serviceareafromlayer`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Service area (from point)

Returns all the edges or parts of edges of a network that can be reached within a given distance or time, starting from a point feature. This allows the evaluation of accessibility within a network, e.g. what are the places I can navigate to on a road network without spending a cost greater than a given value (the cost can be distance or time).

Parameters

Label	名称	Type	Description
Vector layer representing the network	INPUT	[vector: line]	Line vector layer representing the network to be covered
Start point (x, y)	START_POINT	[coordinates]	Coordinate of the point to calculate the service area around.
Path type to calculate	STRATEGY	[enumeration] Default: 0	The type of path to calculate. One of: <ul style="list-style-type: none"> • 0 --- Shortest • 1 --- Fastest
Travel cost	TRAVEL_COST	[number] Default: 0	The value is estimated as a distance (in the network layer units) when looking for the <i>Shortest</i> path and as time (in hours) for the <i>Fastest</i> path.
Advanced parameters	GUI only		Group of advanced network analysis parameters - see below.
Service area (lines)	OUTPUT_LINES	[vector: line] Default: [Create temporary layer]	Specify the output line layer for the service area. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Service area (boundary nodes)	OUTPUT	[vector: point] Default: [Skip output]	Specify the output point layer for the service area boundary nodes. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Advanced parameters

Label	名称	Type	Description
Direction field Optional	DIRECTION_FIELD	[tablefield: string] Default: 0.0	The field used to specify directions for the network edges. The values used in this field are specified with the three parameters Value for forward direction, Value for backward direction and Value for both directions. Forward and reverse directions correspond to a one-way edge, "both directions" indicates a two-way edge. If a feature does not have a value in this field, or no field is set then the default direction setting (provided with the Default direction parameter) is used.
Value for forward direction Optional	VALUE_FORWARD	[string] Default: " (empty string)	Value set in the direction field to identify edges with a forward direction
Value for backward direction Optional	VALUE_BACKWARD	[string] Default: " (empty string)	Value set in the direction field to identify edges with a backward direction
Value for both directions Optional	VALUE_BOTH	[string] Default: " (empty string)	Value set in the direction field to identify bidirectional edges
Default direction Optional	DEFAULT_DIRECTION	[enumeration] Default: 2	If a feature has no value set in the direction field or if no direction field is set, then this direction value is used. One of: <ul style="list-style-type: none"> • 0 --- Forward direction • 1 --- Backward direction • 2 --- Both directions
Speed field Optional	SPEED_FIELD	[tablefield: string]	Field providing the speed value (in km/h) for the edges of the network when looking for the fastest path. If a feature does not have a value in this field, or no field is set then the default speed value (provided with the Default speed parameter) is used.
Default speed (km/h) Optional	DEFAULT_SPEED	[number] Default: 50.0	Value to use to calculate the travel time if no speed field is provided for an edge
Topology tolerance Optional	TOLERANCE	[number] Default: 0.0	Two lines with nodes closer than the specified tolerance are considered connected
Include upper/lower bound points	INCLUDE_BOUNDS	[boolean] Default: False	Creates a point layer output with two points for each edge at the boundaries of the service area. One point is the start of that edge, the other is the end.

Outputs

Label	名称	Type	Description
Service area (boundary nodes)	OUTPUT	[vector: point]	The output point layer with the service area boundary nodes.
Service area (lines)	OUTPUT_LINES	[vector: line]	Line layer representing the parts of the network that can be serviced by the start point, for the given cost.

Python code

Algorithm ID: qgis:serviceareafrompoint

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Shortest path (layer to point)

Computes the optimal (shortest or fastest) routes from multiple start points defined by a vector layer and a given end point.

Parameters

Label	名称	Type	Description
Vector layer representing network	INPUT	[vector: line]	Line vector layer representing the network to be covered
Path type to calculate	STRATEGY	[enumeration] Default: 0	The type of path to calculate. One of: <ul style="list-style-type: none"> • 0 --- Shortest • 1 --- Fastest
Vector layer with start points	START_POINTS	[vector: point]	Point vector layer whose features are used as start points of the routes
End point (x, y)	END_POINT	[coordinates]	Point feature representing the end point of the routes
Advanced parameters	GUI only		The Advanced parameters group:

下页继续

表 24.18 - 续上页

Label	名称	Type	Description
Direction field Optional	DIRECTION_FIELD	[tablefield: string] Default: 0.0	The field used to specify directions for the network edges. The values used in this field are specified with the three parameters Value for forward direction, Value for backward direction and Value for both directions. Forward and reverse directions correspond to a one-way edge, "both directions" indicates a two-way edge. If a feature does not have a value in this field, or no field is set then the default direction setting (provided with the Default direction parameter) is used.
Value for forward direction Optional	VALUE_FORWARD	[string] Default: " (empty string)	Value set in the direction field to identify edges with a forward direction
Value for backward direction Optional	VALUE_BACKWARD	[string] Default: " (empty string)	Value set in the direction field to identify edges with a backward direction
Value for both directions Optional	VALUE_BOTH	[string] Default: " (empty string)	Value set in the direction field to identify bidirectional edges
Default direction Optional	DEFAULT_DIRECTION	[enumeration] Default: 2	If a feature has no value set in the direction field or if no direction field is set, then this direction value is used. One of: <ul style="list-style-type: none"> • 0 --- Forward direction • 1 --- Backward direction • 2 --- Both directions
Speed field Optional	SPEED_FIELD	[tablefield: string]	Field providing the speed value (in km/h) for the edges of the network when looking for the fastest path. If a feature does not have a value in this field, or no field is set then the default speed value (provided with the Default speed parameter) is used.
Default speed (km/h) Optional	DEFAULT_SPEED	[number] Default: 50.0	Value to use to calculate the travel time if no speed field is provided for an edge
Topology tolerance Optional	TOLERANCE	[number] Default: 0.0	Two lines with nodes closer than the specified tolerance are considered connected
			End of the Advanced parameters group
Shortest path	OUTPUT	[vector: line]	Specify the output line layer for the shortest paths. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Shortest path	OUTPUT	[vector: line]	Line layer of the shortest or fastest path from each of the start points to the end point

Python code

Algorithm ID: qgis:shortestpathlayertopoint

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Shortest path (point to layer)

Computes the optimal (shortest or fastest) routes between a given start point and multiple end points defined by a point vector layer.

Parameters

Label	名称	Type	Description
Vector layer representing network	INPUT	[vector: line]	Line vector layer representing the network to be covered
Path type to calculate	STRATEGY	[enumeration] Default: 0	The type of path to calculate. One of: <ul style="list-style-type: none"> • 0 --- Shortest • 1 --- Fastest
Start point (x, y)	START_POINT	[coordinates]	Point feature representing the start point of the routes
Vector layer with end points	END_POINTS	[vector: point]	Point vector layer whose features are used as end points of the routes
Direction field Optional <i>Advanced</i>	DIRECTION_FIELD	[tablefield: string] Default: 0.0	The field used to specify directions for the network edges. The values used in this field are specified with the three parameters Value for forward direction, Value for backward direction and Value for both directions. Forward and reverse directions correspond to a one-way edge, "both directions" indicates a two-way edge. If a feature does not have a value in this field, or no field is set then the default direction setting (provided with the Default direction parameter) is used.
Value for forward direction Optional <i>Advanced</i>	VALUE_FORWARD	[string] Default: " (empty string)	Value set in the direction field to identify edges with a forward direction

下页继续

表 24.19 - 续上页

Label	名称	Type	Description
Value for backward direction Optional <i>Advanced</i>	VALUE_BACKWARD	[string] Default: " (empty string)	Value set in the direction field to identify edges with a backward direction
Value for both directions Optional <i>Advanced</i>	VALUE_BOTH	[string] Default: " (empty string)	Value set in the direction field to identify bidirectional edges
Default direction Optional <i>Advanced</i>	DEFAULT_DIRECTION	[enumeration] Default: 2	If a feature has no value set in the direction field or if no direction field is set, then this direction value is used. One of: <ul style="list-style-type: none"> • 0 --- Forward direction • 1 --- Backward direction • 2 --- Both directions
Speed field Optional <i>Advanced</i>	SPEED_FIELD	[tablefield: string]	Field providing the speed value (in km/h) for the edges of the network when looking for the fastest path. If a feature does not have a value in this field, or no field is set then the default speed value (provided with the <code>Default speed</code> parameter) is used.
Default speed (km/h) Optional <i>Advanced</i>	DEFAULT_SPEED	[number] Default: 50.0	Value to use to calculate the travel time if no speed field is provided for an edge
Topology tolerance Optional <i>Advanced</i>	TOLERANCE	[number] Default: 0.0	Two lines with nodes closer than the specified tolerance are considered connected
Shortest path	OUTPUT	[vector: line]	Specify the output line layer for the shortest paths. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Shortest path	OUTPUT	[vector: line]	Line layer of the shortest or fastest path from each of the start points to the end point

Python code

Algorithm ID: `qgis:shortestpathpointtolayer`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Shortest path (point to point)

Computes the optimal (shortest or fastest) route between a given start point and a given end point.

Parameters

Label	名称	Advanced	Type	Description
Vector layer representing network	INPUT		[vector: line]	Line vector layer representing the network to be covered
Path type to calculate	STRATEGY		[enumeration] Default: 0	The type of path to calculate. One of: <ul style="list-style-type: none"> • 0 --- Shortest • 1 --- Fastest
Start point (x, y)	START_POINT		[coordinates]	Point feature representing the start point of the routes
End point (x, y)	END_POINT		[coordinates]	Point feature representing the end point of the routes
Direction field Optional	DIRECTION_FIELD		[tablefield: string] Default: 0.0	The field used to specify directions for the network edges. The values used in this field are specified with the three parameters Value for forward direction, Value for backward direction and Value for both directions. Forward and reverse directions correspond to a one-way edge, "both directions" indicates a two-way edge. If a feature does not have a value in this field, or no field is set then the default direction setting (provided with the Default direction parameter) is used.
Value for forward direction Optional	VALUE_FORWARD		[string] Default: " (empty string)	Value set in the direction field to identify edges with a forward direction
Value for backward direction Optional	VALUE_BACKWARD		[string] Default: " (empty string)	Value set in the direction field to identify edges with a backward direction
Value for both directions Optional	VALUE_BOTH	X	[string] Default: " (empty string)	Value set in the direction field to identify bidirectional edges
Default direction Optional	DEFAULT_DIRECTION		[enumeration] Default: 2	If a feature has no value set in the direction field or if no direction field is set, then this direction value is used. One of: <ul style="list-style-type: none"> • 0 --- Forward direction • 1 --- Backward direction • 2 --- Both directions

下页继续

表 24.20 - 续上页

Label	名称	Advanced	Type	Description
Speed field Optional	SPEED_FIELD	X	[tablefield: string]	Field providing the speed value (in km/h) for the edges of the network when looking for the fastest path. If a feature does not have a value in this field, or no field is set then the default speed value (provided with the <code>Default speed</code> parameter) is used.
Default speed (km/h) Optional	DEFAULT_SPEED		[number] Default: 50.0	Value to use to calculate the travel time if no speed field is provided for an edge
Topology tolerance Optional	TOLERANCE	X	[number] Default: 0.0	Two lines with nodes closer than the specified tolerance are considered connected
Shortest path	OUTPUT		[vector: line]	Specify the output line layer for the shortest paths. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Shortest path	OUTPUT	[vector: line]	Line layer of the shortest or fastest path from each of the start point to the end point

Python code

Algorithm ID: `qgis:shortestpathpointtopoint`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.8 Plots

Bar plot

Creates a bar plot from a category and a layer field.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Category field name	NAME_FIELD	[tablefield: any]	Categorical field to use for grouping the bars (X axis)
Value field	VALUE_FIELD	[tablefield: any]	Value to use for the plot (Y axis).
Bar plot	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for the plot. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Bar plot	OUTPUT	[html]	HTML file with the plot. Available in the <i>Processing ► Result Viewer</i> .

Python code

Algorithm ID: qgis:barplot

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Box plot

Creates a box plot from a category field and a numerical layer field.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Category name field	NAME_FIELD	[tablefield: any]	Categorical field to use for grouping the boxes (X axis)
Value field	VALUE_FIELD	[tablefield: any]	Value to use for the plot (Y axis).

下页继续

表 24.21 - 续上页

Label	名称	Type	Description
Additional statistic lines	MSD	[enumeration] Default: 0	Additional statistics information to add to the plot. One of: <ul style="list-style-type: none"> • 0 --- Show Mean • 1 --- Show Standard Deviation • 2 --- Don't show mean and standard deviation
Box plot	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for the plot. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Box plot	OUTPUT	[html]	HTML file with the plot. Available in the <i>Processing ► Result Viewer</i> .

Python code

Algorithm ID: qgis:boxplot

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Mean and standard deviation plot

Creates a box plot with mean and standard deviation values.

Parameters

Label	名称	Type	Description
Input table	INPUT	[vector: any]	Input vector layer
Category field name	NAME_FIELD	[tablefield: any]	Categorical field to use for grouping the boxes (X axis)
Value field	VALUE_FIELD	[tablefield: any]	Value to use for the plot (Y axis).
Plot	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for the plot. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Plot	OUTPUT	[html]	HTML file with the plot. Available in the <i>Processing ► Result Viewer</i> .

Python code

Algorithm ID: qgis:meanandstandarddeviationplot

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Polar plot

Generates a polar plot based on the value of an input vector layer.

Two fields must be entered as parameters: one that defines the category each feature (to group features) and another one with the variable to plot (this has to be a numeric one).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Category name field	NAME_FIELD	[tablefield: any]	Categorical field to use for grouping the features (X axis)
Value field	VALUE_FIELD	[tablefield: any]	Value to use for the plot (Y axis).
Polar plot	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for the plot. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Polar plot	OUTPUT	[html]	HTML file with the plot. Available in the <i>Processing ► Result Viewer</i> .

Python code

Algorithm ID: qgis:polarplot

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster layer histogram

Generates a histogram with the values of a raster layer.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Band number	BAND	[raster band]	Raster band to use for the histogram
number of bins	BINS	[number] Default: 10	The number of bins to use in the histogram (X axis). Minimum 2.
Histogram	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for the plot. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Histogram	OUTPUT	[html]	HTML file with the plot. Available in the <i>Processing ► Result Viewer</i> .

Python code

Algorithm ID: qgis:rasterlayerhistogram

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Vector layer histogram

Generates a histogram with the values of the attribute of a vector layer.

The attribute to use for computing the histogram must be numeric.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Attribute	FIELD	[tablefield: any]	Value to use for the plot (Y axis).
number of bins	BINS	[number] Default: 10	The number of bins to use in the histogram (X axis). Minimum 2.
Histogram	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for the plot. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Histogram	OUTPUT	[html]	HTML file with the plot. Available in the <i>Processing ► Result Viewer</i> .

Python code

Algorithm ID: qgis:vectorlayerhistogram

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Vector layer scatterplot

Creates a simple X - Y scatter plot for a vector layer.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
X attribute	XFIELD	[tablefield: any]	Field to use for the X axis
Y attribute	YFIELD	[tablefield: any]	Field to use for the Y axis
Scatterplot	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for the plot. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Scatterplot	OUTPUT	[html]	HTML file with the plot. Available in the <i>Processing ► Result Viewer</i> .

Python code

Algorithm ID: qgis:vectorlayersscatterplot

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Vector layer scatterplot 3D

Creates a 3D scatter plot for a vector layer.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
X attribute	XFIELD	[tablefield: any]	Field to use for the X axis
Y attribute	YFIELD	[tablefield: any]	Field to use for the Y axis
Z attribute	ZFIELD	[tablefield: any]	Field to use for the Z axis
Histogram	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for the plot. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Histogram	OUTPUT	[html]	HTML file with the plot. Available in the <i>Processing ► Result Viewer</i> .

Python code

Algorithm ID: qgis:scatter3dplot

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.9 栅格分析

Cell statistics

Computes per-cell statistics based on input raster layers and for each cell writes the resulting statistics to an output raster. At each cell location, the output value is defined as a function of all overlaid cell values of the input rasters.

By default, a NoData cell in ANY of the input layers will result in a NoData cell in the output raster. If the *Ignore NoData values* option is checked, then NoData inputs will be ignored in the statistic calculation. This may result in NoData output for locations where all cells are NoData.

The *Reference layer* parameter specifies an existing raster layer to use as a reference when creating the output raster. The output raster will have the same extent, CRS, and pixel dimensions as this layer.

Calculation details: Input raster layers that do not match the cell size of the reference raster layer will be resampled using nearest neighbor resampling. The output raster data type will be set to the most complex data type present in the input datasets except when using the functions Mean, Standard deviation and Variance (data type is always Float32 or Float64 depending on input float type) or Count and Variety (data type is always Int32).

- **Count:** The count statistic will always result in the number of cells without NoData values at the current cell location.
- **Median:** If the number of input layers is even, the median will be calculated as the arithmetic mean of the two middle values of the ordered cell input values.
- **Minority/Majority:** If no unique minority or majority could be found, the result is NoData, except all input cell values are equal.

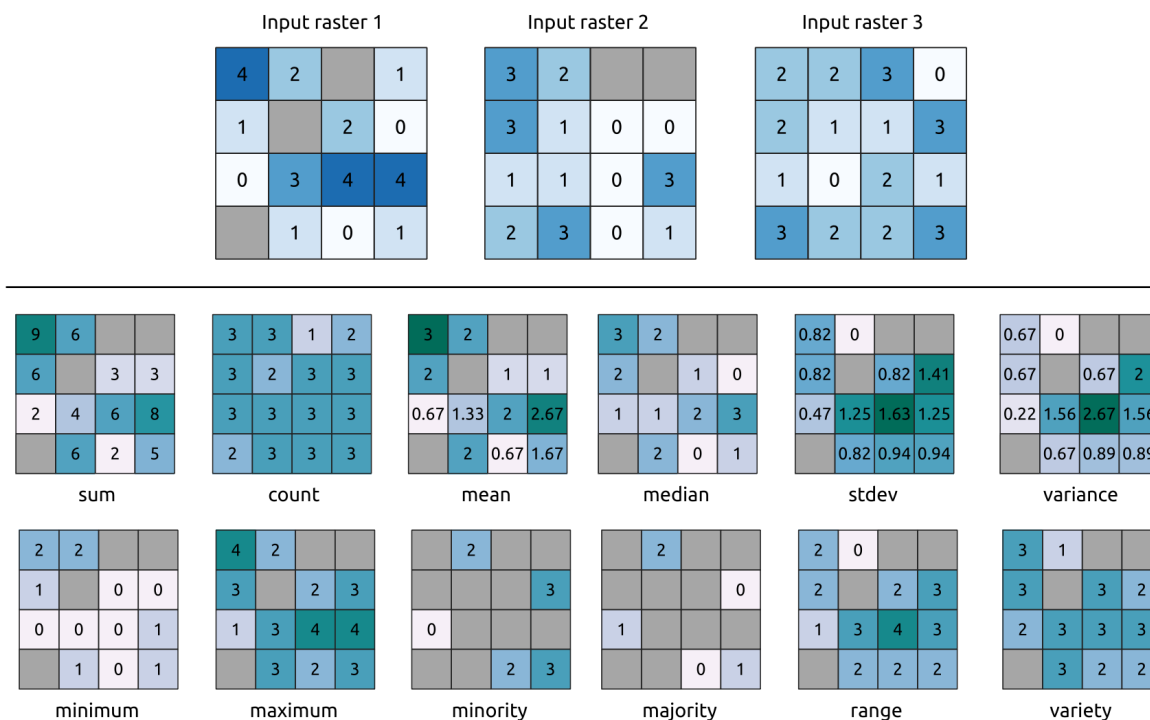


图 24.8: Example with all the statistic functions. NoData cells (grey) are taken into account.

Parameters

Label	名称	Type	Description
Input layers	INPUT	[raster] [list]	Input raster layers
Statistic	STATISTIC	[enumeration] Default: 0	Available statistics. Options: <ul style="list-style-type: none"> • 0 --- Sum • 1 --- Count • 2 --- Mean • 3 --- Median • 4 --- Standard deviation • 5 --- Variance • 6 --- Minimum • 7 --- Maximum • 8 --- Minority (least common value) • 9 --- Majority (most common value) • 10 --- Range (max - min) • 11 --- Variety (unique value count)
Ignore NoData values	IGNORE_NODATA	[boolean] Default: True	Calculate statistics also for all cells stacks, ignoring NoData occurrence.
Reference layer	REF_LAYER	[raster]	The reference layer to create the output layer from (extent, CRS, pixel dimensions)
Output no data value Optional	OUTPUT_NO_DATA	[number] Default: -9999.0	Value to use for nodata in the output layer
Output layer	OUTPUT	[same as input]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Extent	EXTENT	[extent]	The spatial extent of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Output raster	OUTPUT	[raster]	Output raster layer containing the result
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer

Python code

Algorithm ID: qgis:cellstatistics

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Equal to frequency

Evaluates on a cell-by-cell basis the frequency (number of times) the values of an input stack of rasters are equal to the value of a value layer. The output raster extent and resolution are defined by the input raster layer and is always of Int32 type.

If multiband rasters are used in the data raster stack, the algorithm will always perform the analysis on the first band of the rasters - use GDAL to use other bands in the analysis. The output NoData value can be set manually.

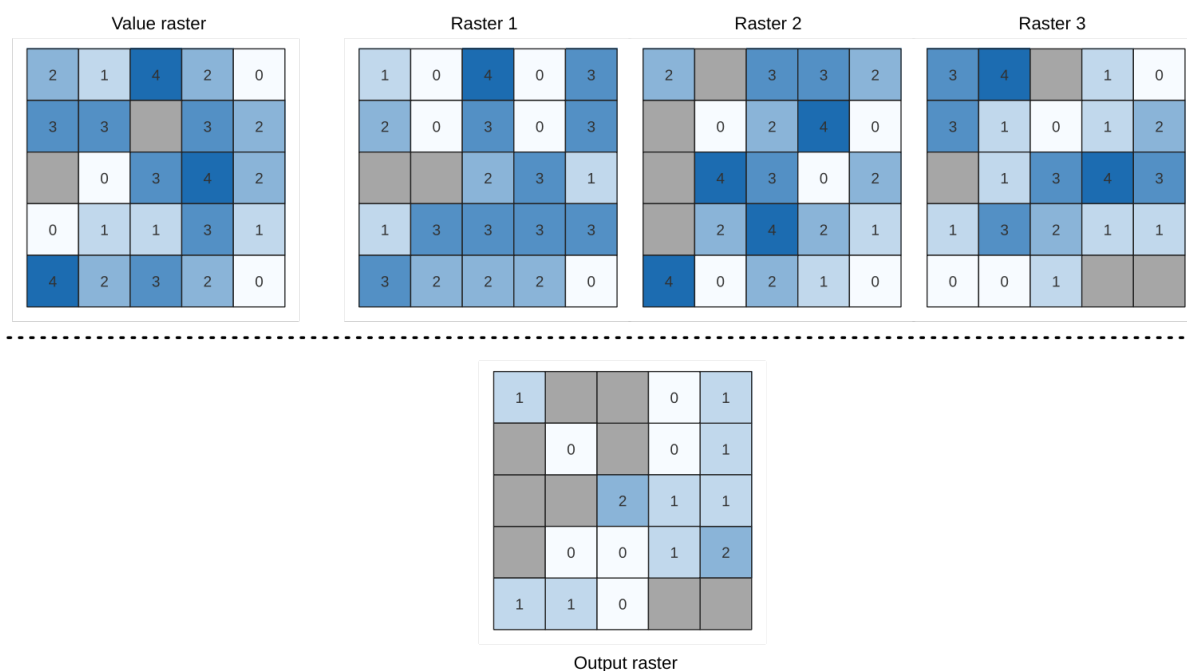


图 24.9: For each cell in the output raster, the value represents the number of times that the corresponding cells in the list of rasters are the same as the value raster. NoData cells (grey) are taken into account.

参见:

Greater than frequency, Less than frequency

Parameters

Basic parameters

Label	名称	Type	Description
Input value raster	INPUT_VALUE_RASTER	[raster]	The input value layer serves as reference layer for the sample layers
Value raster band	INPUT_VALUE_RASTER_BAND	[rasterBand] Default: The first band of the raster layer	Select the band you want to use as sample
Input raster layers	INPUT_RASTERS	[raster] [list]	Raster layers to evaluate. If multiband rasters are used in the data raster stack, the algorithm will always perform the analysis on the first band of the rasters

下页继续

表 24.22 - 续上页

Label	名称	Type	Description
Ignore NoData values	IGNORE_NODATA	[boolean] Default: False	If unchecked, any NoData cells in the value raster or the data layer stack will result in a NoData cell in the output raster
Output layer	OUTPUT	[same as input] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Output no data value Optional	OUTPUT_NO_DATA	[number] Default: -9999.0	Value to use for nodata in the output layer

Outputs

Label	名称	Type	Description
Output layer	OUTPUT	[raster]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[string]	The coordinate reference system of the output raster layer
Extent	EXTENT	[string]	The spatial extent of the output raster layer
Count of cells with equal value occurrences	FOUND_LOCATIONS	[number]	
Height in pixels	HEIGHT_IN_PIXELS	[number]	The number of rows in the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer
Mean frequency at valid cell locations	MEAN_FREQUENCY_VALID_CELL_LOCATION	[number]	
Count of value occurrences	OCCURRENCE_COUNT	[number]	
Width in pixels	WIDTH_IN_PIXELS	[integer]	The number of columns in the output raster layer

Python code

Algorithm ID: native:equaltofrequency

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Fuzzify raster (gaussian membership)

Transforms an input raster to a fuzzified raster by assigning a membership value to each pixel, using a Gaussian membership function. Membership values range from 0 to 1. In the fuzzified raster, a value of 0 implies no membership of the defined fuzzy set, whereas a value of 1 means full membership. The gaussian membership function is defined as $\mu(x) = e^{-f1*(x-f2)^2}$, where $f1$ is the spread and $f2$ the midpoint.

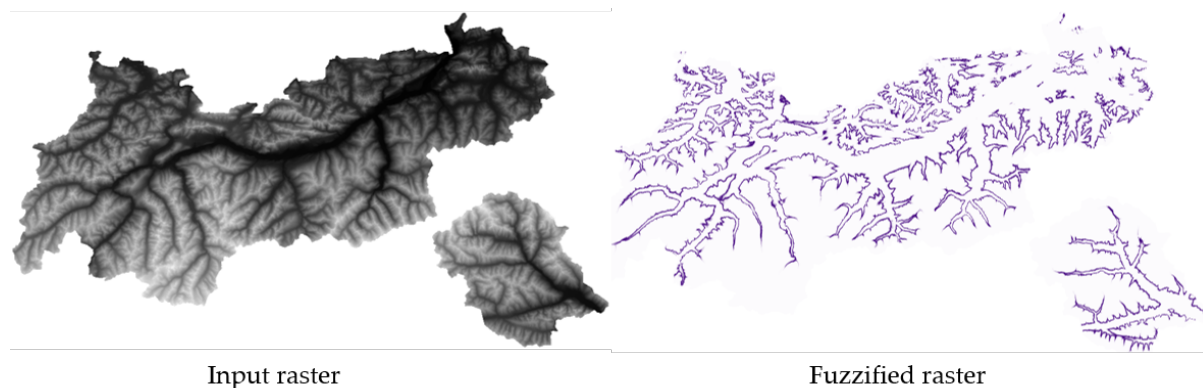


图 24.10: Fuzzify raster example. Input raster source: Land Tirol - data.tirol.gv.at.

参见:

Fuzzify raster (large membership), Fuzzify raster (linear membership), Fuzzify raster (near membership), Fuzzify raster (power membership), Fuzzify raster (small membership)

Parameters

Label	名称	Type	Description
Input Raster	INPUT	[raster]	Input raster layer
Band Number	BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band that you want to fuzzify.
Function midpoint	FUZZYMIDPOINT	[number] Default: 10	Midpoint of the gaussian function
Function spread	FUZZYSPREAD	[number] Default: 0.01	Spread of the gaussian function
Fuzzified raster	OUTPUT	[same as input]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Fuzzified raster	OUTPUT	[same as input]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Extent	EXTENT	[extent]	The spatial extent of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer

Python code

Algorithm ID: qgis:fuzzifyrastergaussianmembership

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Fuzzify raster (large membership)

Transforms an input raster to a fuzzified raster by assigning a membership value to each pixel, using a Large membership function. Membership values range from 0 to 1. In the fuzzified raster, a value of 0 implies no membership of the defined fuzzy set, whereas a value of 1 means full membership. The large membership function is defined as

$$\mu(x) = \frac{1}{1 + \left(\frac{x}{f2}\right)^{-f1}}$$

, where *f1* is the spread and *f2* the midpoint.

参见:

Fuzzify raster (gaussian membership), *Fuzzify raster (linear membership)*, *Fuzzify raster (near membership)*, *Fuzzify raster (power membership)*, *Fuzzify raster (small membership)*

Parameters

Label	名称	Type	Description
Input Raster	INPUT	[raster]	Input raster layer
Band Number	BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band that you want to fuzzify.
Function midpoint	FUZZYMIDPOINT	[number] Default: 50	Midpoint of the large function
Function spread	FUZZYSPREAD	[number] Default: 5	Spread of the large function
Fuzzified raster	OUTPUT	[same as input]	Specification of the output raster. One of: <ul style="list-style-type: none"> Save to a Temporary File Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Fuzzified raster	OUTPUT	[same as input]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Extent	EXTENT	[extent]	The spatial extent of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer

Python code

Algorithm ID: qgis:fuzzifyrasterlargemembership

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Fuzzify raster (linear membership)

Transforms an input raster to a fuzzified raster by assigning a membership value to each pixel, using a Linear membership function. Membership values range from 0 to 1. In the fuzzified raster, a value of 0 implies no membership of the defined fuzzy set, whereas a value of 1 means full membership. The linear function is defined as

$$\mu(X) \begin{cases} 0 & x \leq a \\ \frac{x-a}{b-a} & a < x < b \\ 1 & x \geq b \end{cases}, \text{ where } a \text{ is the low bound and } b \text{ the high bound. This equation assigns membership values using a linear transformation for pixel values between the low and high bounds. Pixels values smaller than the low bound are given 0 membership whereas pixel values greater than the high bound are given 1 membership.}$$

参见:

Fuzzify raster (gaussian membership), Fuzzify raster (large membership), Fuzzify raster (near membership), Fuzzify raster (power membership), Fuzzify raster (small membership)

Parameters

Label	名称	Type	Description
Input Raster	INPUT	[raster]	Input raster layer
Band Number	BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band that you want to fuzzify.
Low fuzzy membership bound	FUZZYLOWBOUND	[number] Default: 0	Low bound of the linear function
High fuzzy membership bound	FUZZYHIGHBOUND	[number] Default: 1	High bound of the linear function
Fuzzified raster	OUTPUT	[same as input]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Fuzzified raster	OUTPUT	[same as input]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Extent	EXTENT	[extent]	The spatial extent of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer

Python code

Algorithm ID: qgisfuzzifyrasterlinearmembership

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Fuzzify raster (near membership)

Transforms an input raster to a fuzzified raster by assigning a membership value to each pixel, using a Near membership function. Membership values range from 0 to 1. In the fuzzified raster, a value of 0 implies no membership of the defined fuzzy set, whereas a value of 1 means full membership. The near membership function is defined as

$$\mu(x) = \frac{1}{1 + f1 * (x - f2)^2}$$

, where *f1* is the spread and *f2* the midpoint.

参见:

Fuzzify raster (gaussian membership), *Fuzzify raster (large membership)*, *Fuzzify raster (linear membership)*, *Fuzzify raster (power membership)*, *Fuzzify raster (small membership)*

Parameters

Label	名称	Type	Description
Input Raster	INPUT	[raster]	Input raster layer
Band Number	BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band that you want to fuzzify.
Function midpoint	FUZZYMIDPOINT	[number] Default: 50	Midpoint of the near function
Function spread	FUZZYSPREAD	[number] Default: 0.01	Spread of the near function
Fuzzified raster	OUTPUT	[same as input]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Fuzzified raster	OUTPUT	[same as input]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Extent	EXTENT	[extent]	The spatial extent of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer

Python code

Algorithm ID: qgis:fuzzifyrasternearmembership

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Fuzzify raster (power membership)

Transforms an input raster to a fuzzified raster by assigning a membership value to each pixel, using a Power membership function. Membership values range from 0 to 1. In the fuzzified raster, a value of 0 implies no membership of the defined fuzzy set, whereas a value of 1 means full membership. The power function is defined as

$$\mu(x) \begin{cases} 0 & x \leq a \\ \left(\frac{x-a}{b-a}\right)^{f1} & a < x < b \\ 1 & x \geq b \end{cases}, \text{ where } a \text{ is the low bound, } b \text{ is the high bound, and } f1 \text{ the exponent. This equation assigns membership values using the power transformation for pixel values between the low and high bounds. Pixels values smaller than the low bound are given 0 membership whereas pixel values greater than the high bound are given 1 membership.}$$

参见:

Fuzzify raster (gaussian membership), Fuzzify raster (large membership), Fuzzify raster (linear membership), Fuzzify raster (near membership), Fuzzify raster (small membership)

Parameters

Label	名称	Type	Description
Input Raster	INPUT	[raster]	Input raster layer
Band Number	BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band that you want to fuzzify.
Low fuzzy membership bound	FUZZYLOWBOUND	[number] Default: 0	Low bound of the power function
High fuzzy membership bound	FUZZYHIGHBOUND	[number] Default: 1	High bound of the power function
High fuzzy membership bound	FUZZYEXPONENT	[number] Default: 2	Exponent of the power function
Fuzzified raster	OUTPUT	[same as input]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Fuzzified raster	OUTPUT	[same as input]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Extent	EXTENT	[extent]	The spatial extent of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer

Python code

Algorithm ID: qgisfuzzifyrasterpowermembership

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Fuzzify raster (small membership)

Transforms an input raster to a fuzzified raster by assigning a membership value to each pixel, using a Small membership function. Membership values range from 0 to 1. In the fuzzified raster, a value of 0 implies no membership of the defined fuzzy set, whereas a value of 1 means full membership. The small membership function is defined as

$$\mu(x) = \frac{1}{1 + \left(\frac{x}{f2}\right)^{f1}}, \text{ where } f1 \text{ is the spread and } f2 \text{ the midpoint.}$$

参见:

Fuzzify raster (gaussian membership), Fuzzify raster (large membership) Fuzzify raster (linear membership), Fuzzify raster (near membership), Fuzzify raster (power membership)

Parameters

Label	名称	Type	Description
Input Raster	INPUT	[raster]	Input raster layer
Band Number	BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band that you want to fuzzify.
Function midpoint	FUZZYMIDPOINT	[number] Default: 50	Midpoint of the small function
Function spread	FUZZYSPREAD	[number] Default: 5	Spread of the small function
Fuzzified raster	OUTPUT	[same as input]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Fuzzified raster	OUTPUT	[same as input]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Extent	EXTENT	[extent]	The spatial extent of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer

Python code

Algorithm ID: qgisfuzzifyrastersmallmembership

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Greater than frequency

Evaluates on a cell-by-cell basis the frequency (number of times) the values of an input stack of rasters are equal to the value of a value raster. The output raster extent and resolution is defined by the input raster layer and is always of Int32 type.

If multiband rasters are used in the data raster stack, the algorithm will always perform the analysis on the first band of the rasters - use GDAL to use other bands in the analysis. The output NoData value can be set manually.

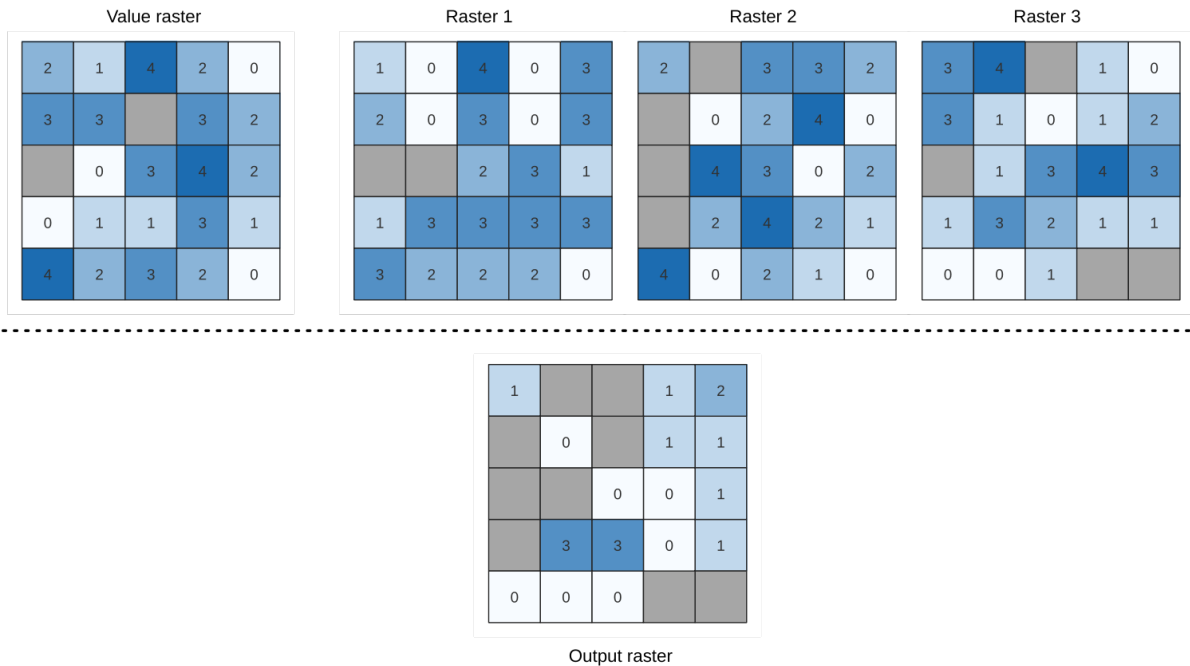


图 24.11: 对于输出栅格的每个单元，其值表示在栅格列表中对应单元的值大于值栅格单元值的次数。NoData 单元 (灰色) 被计入。

参见:

Equal to frequency, Less than frequency

Parameters

Basic parameters

Label	名称	Type	Description
Input value raster	INPUT_VALUE_RASTER	[raster]	The input value layer serves as reference layer for the sample layers
Value raster band	INPUT_VALUE_RASTER_BAND	[rasterBand] Default: The first band of the raster layer	Select the band you want to use as sample
Input raster layers	INPUT_RASTERS	[raster] [list]	Raster layers to evaluate. If multiband rasters are used in the data raster stack, the algorithm will always perform the analysis on the first band of the rasters

下页继续

表 24.25 - 续上页

Label	名称	Type	Description
Ignore NoData values	IGNORE_NODATA	[boolean] Default: False	If unchecked, any NoData cells in the value raster or the data layer stack will result in a NoData cell in the output raster
Output layer	OUTPUT	[same as input] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Output no data value Optional	OUTPUT_NO_DATA	[number] Default: -9999.0	Value to use for nodata in the output layer

Outputs

Label	名称	Type	Description
Output layer	OUTPUT	[raster]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[string]	The coordinate reference system of the output raster layer
Extent	EXTENT	[string]	The spatial extent of the output raster layer
Count of cells with equal value occurrences	FOUND_LOCATIONS	[number]	
Height in pixels	HEIGHT_IN_PIXELS	[number]	The number of rows in the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer
Mean frequency at valid cell locations	MEAN_FREQUENCY_VALID_CELL_LOCATION	[number]	
Count of value occurrences	OCCURRENCE_COUNT	[number]	
Width in pixels	WIDTH_IN_PIXELS	[integer]	The number of columns in the output raster layer

Python code

Algorithm ID: native:greaterthanfrequency

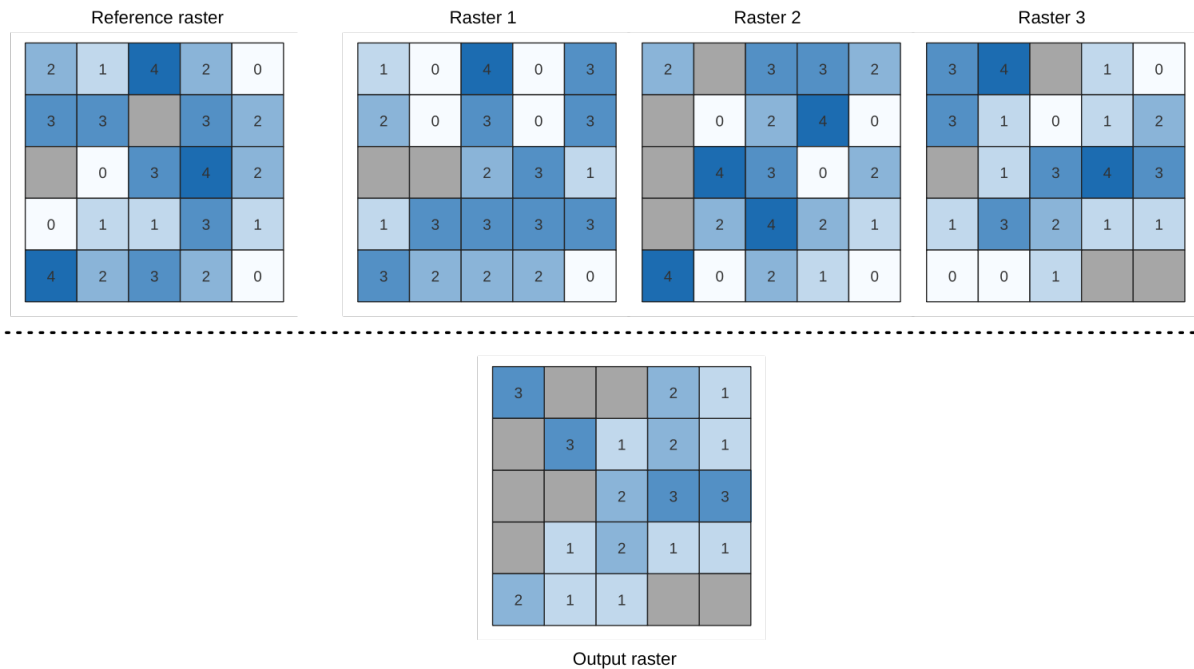
```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Highest position in raster stack

Evaluates on a cell-by-cell basis the position of the raster with the highest value in a stack of rasters. Position counts start with 1 and range to the total number of input rasters. The order of the input rasters is relevant for the algorithm. If multiple rasters feature the highest value, the first raster will be used for the position value.

If multiband rasters are used in the data raster stack, the algorithm will always perform the analysis on the first band of the rasters - use GDAL to use other bands in the analysis. Any NoData cells in the raster layer stack will result in a NoData cell in the output raster unless the "ignore NoData" parameter is checked. The output NoData value can be set manually. The output rasters extent and resolution is defined by a reference raster layer and is always of `Int32` type.



参见:

Lowest position in raster stack

Parameters

Basic parameters

Label	名称	Type	Description
Input raster layers	INPUT_RASTERS	[raster] [list]	List of raster layers to compare with
Reference layer	REFERENCE_LAYER	[raster]	The reference layer for the output layer creation (extent, CRS, pixel dimensions)
Ignore NoData values	IGNORE_NODATA	[boolean] Default: False	If unchecked, any NoData cells in the data layer stack will result in a NoData cell in the output raster
Output layer	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster containing the result. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Output no data value	OUTPUT_NODATA_VALUE	[number] Default: -9999.0	Value to use for nodata in the output layer

Outputs

Label	名称	Type	Description
Output layer	OUTPUT	[raster]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[string]	The coordinate reference system of the output raster layer
Extent	EXTENT	[string]	The spatial extent of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The number of columns in the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The number of rows in the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer

Python code

Algorithm ID: native:highestpositioninrasterstack

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Less than frequency

Evaluates on a cell-by-cell basis the frequency (number of times) the values of an input stack of rasters are less than the value of a value raster. The output raster extent and resolution is defined by the input raster layer and is always of Int32 type.

If multiband rasters are used in the data raster stack, the algorithm will always perform the analysis on the first band of the rasters - use GDAL to use other bands in the analysis. The output NoData value can be set manually.

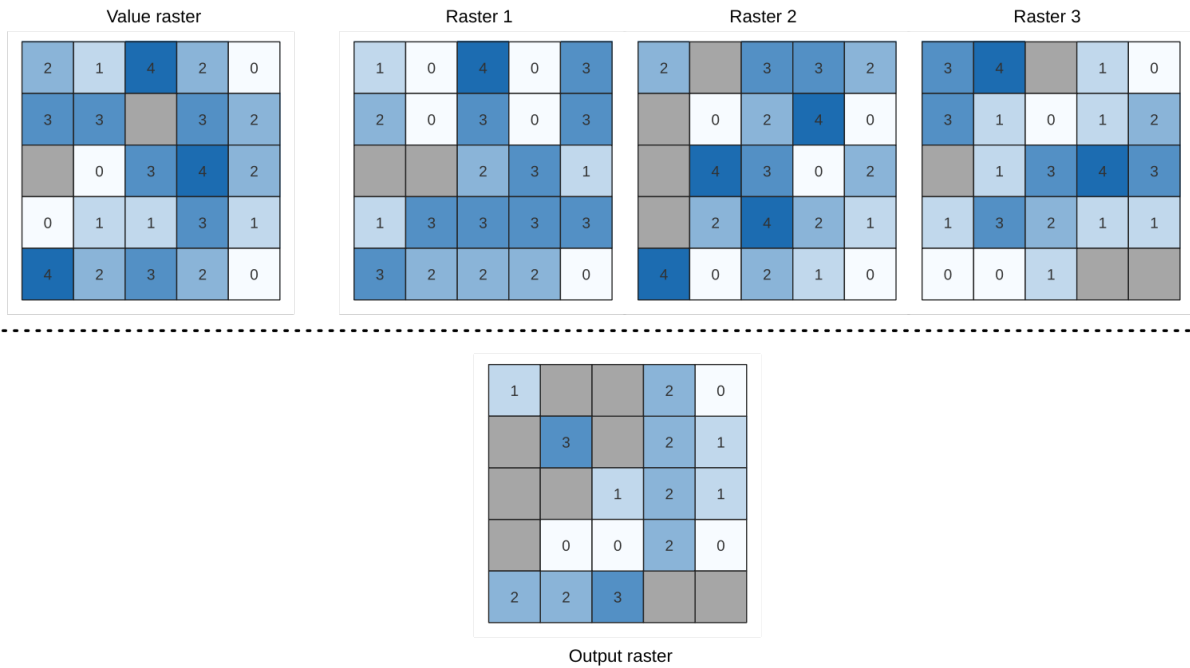


图 24.12: For each cell in the output raster, the value represents the number of times that the corresponding cells in the list of rasters are less than the value raster. NoData cells (grey) are taken into account.

参见:

Equal to frequency, Greater than frequency

Parameters

Basic parameters

Label	名称	Type	Description
Input value raster	INPUT_VALUE_RASTER	[raster]	The input value layer serves as reference layer for the sample layers
Value raster band	INPUT_VALUE_RASTER_BAND	[raster band] Default: The first band of the raster layer	Select the band you want to use as sample
Input raster layers	INPUT_RASTERS	[raster] [list]	Raster layers to evaluate. If multiband rasters are used in the data raster stack, the algorithm will always perform the analysis on the first band of the rasters
Ignore NoData values	IGNORE_NODATA	[boolean] Default: False	If unchecked, any NoData cells in the value raster or the data layer stack will result in a NoData cell in the output raster
Output layer	OUTPUT	[same as input] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Output no data value Optional	OUTPUT_NO_DATA	[number] Default: -9999.0	Value to use for nodata in the output layer

Outputs

Label	名称	Type	Description
Output layer	OUTPUT	[raster]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[string]	The coordinate reference system of the output raster layer
Extent	EXTENT	[string]	The spatial extent of the output raster layer
Count of cells with equal value occurrences	FOUND_LOCATIONS	[number]	
Height in pixels	HEIGHT_IN_PIXELS	[number]	The number of rows in the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer
Mean frequency at valid cell locations	MEAN_FREQUENCY_VALID_CELL_LOCATION	[number]	
Count of value occurrences	OCCURRENCE_COUNT	[number]	
Width in pixels	WIDTH_IN_PIXELS	[integer]	The number of columns in the output raster layer

Python code

Algorithm ID: native:lessthanfrequency

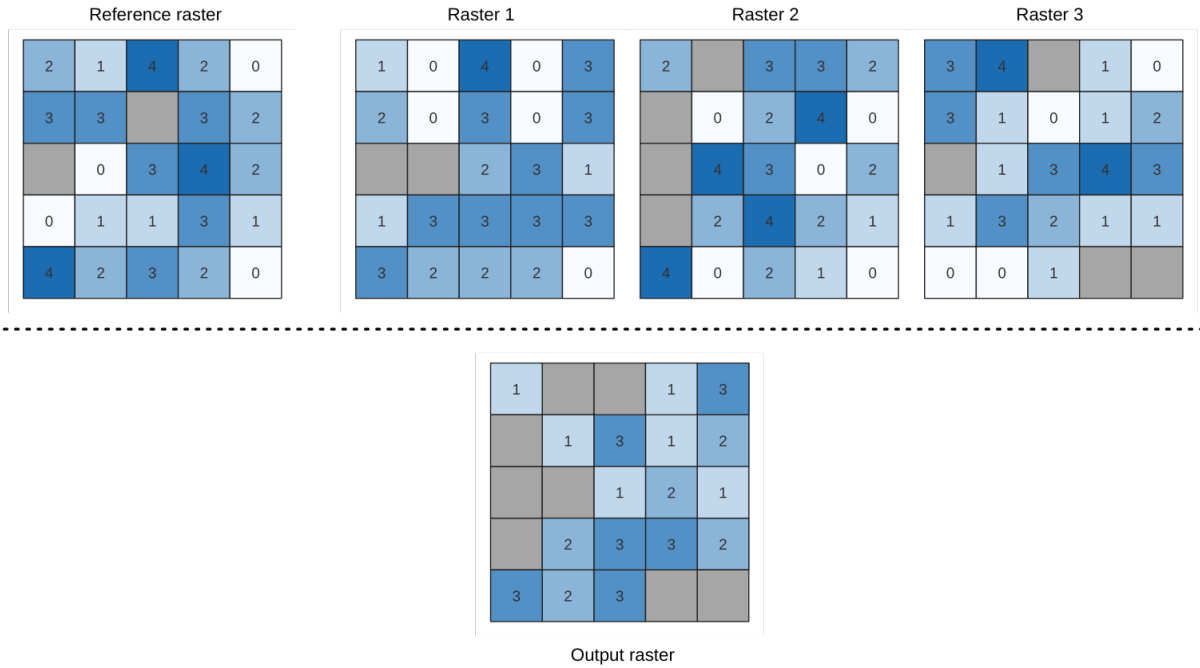
```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Lowest position in raster stack

Evaluates on a cell-by-cell basis the position of the raster with the lowest value in a stack of rasters. Position counts start with 1 and range to the total number of input rasters. The order of the input rasters is relevant for the algorithm. If multiple rasters feature the lowest value, the first raster will be used for the position value.

If multiband rasters are used in the data raster stack, the algorithm will always perform the analysis on the first band of the rasters - use GDAL to use other bands in the analysis. Any NoData cells in the raster layer stack will result in a NoData cell in the output raster unless the "ignore NoData" parameter is checked. The output NoData value can be set manually. The output rasters extent and resolution is defined by a reference raster layer and is always of Int32 type.



参见:

Highest position in raster stack

Parameters

Basic parameters

Label	名称	Type	Description
Input raster layers	INPUT_RASTERS	[raster] [list]	List of raster layers to compare with
Reference layer	REFERENCE_LAYER	[raster]	The reference layer for the output layer creation (extent, CRS, pixel dimensions)
Ignore NoData values	IGNORE_NODATA	[boolean] Default: False	If unchecked, any NoData cells in the data layer stack will result in a NoData cell in the output raster
Output layer	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster containing the result. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Output no data value	OUTPUT_NODATA_VALUE	[number] Default: -9999.0	Value to use for nodata in the output layer

Outputs

Label	名称	Type	Description
Output layer	OUTPUT	[raster]	Output raster layer containing the result
CRS authority identifier	CRS_AUTHID	[string]	The coordinate reference system of the output raster layer
Extent	EXTENT	[string]	The spatial extent of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The number of columns in the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The number of rows in the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer

Python code

Algorithm ID: native:lowestpositioninrasterstack

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster boolean AND

Calculates the boolean AND for a set of input rasters. If all of the input rasters have a non-zero value for a pixel, that pixel will be set to 1 in the output raster. If any of the input rasters have 0 values for the pixel it will be set to 0 in the output raster.

The reference layer parameter specifies an existing raster layer to use as a reference when creating the output raster. The output raster will have the same extent, CRS, and pixel dimensions as this layer.

By default, a nodata pixel in ANY of the input layers will result in a nodata pixel in the output raster. If the *Treat nodata values as false* option is checked, then nodata inputs will be treated the same as a 0 input value.

参见:

Raster boolean OR

Parameters

Label	名称	Type	Description
Input layers	INPUT	[raster] [list]	List of input raster layers
Reference layer	REF_LAYER	[raster]	The reference layer to create the output layer from (extent, CRS, pixel dimensions)
Treat nodata values as false	NODATA_AS_FALSE	[boolean] Default: False	Treat nodata values in the input files as 0 when performing the operation
Output no data value	NO_DATA	[number] Default: -9999.0	Value to use for nodata in the output layer

下页继续

表 24.33 - 续上页

Label	名称	Type	Description
Output data type	DATA_TYPE	[enumeration] Default: 5	Output raster data type. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Output layer	OUTPUT	[raster]	Output raster layer

Outputs

Label	名称	Type	Description
Extent	EXTENT	[extent]	The extent of the output raster layer
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer
NODATA pixel count	NODATA_PIXEL_COUNT	[integer]	The count of nodata pixels in the output raster layer
True pixel count	TRUE_PIXEL_COUNT	[integer]	The count of True pixels (value = 1) in the output raster layer
False pixel count	FALSE_PIXEL_COUNT	[integer]	The count of False pixels (value = 0) in the output raster layer
Output layer	OUTPUT	[raster]	Output raster layer containing the result

Python code

Algorithm ID: qgis:rasterbooleanand

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster boolean OR

Calculates the boolean OR for a set of input rasters. If all of the input rasters have a zero value for a pixel, that pixel will be set to 0 in the output raster. If any of the input rasters have 1 values for the pixel it will be set to 1 in the output raster.

The reference layer parameter specifies an existing raster layer to use as a reference when creating the output raster. The output raster will have the same extent, CRS, and pixel dimensions as this layer.

By default, a nodata pixel in ANY of the input layers will result in a nodata pixel in the output raster. If the *Treat nodata values as false* option is checked, then nodata inputs will be treated the same as a 0 input value.

参见:

[Raster boolean AND](#)

Parameters

Label	名称	Type	Description
Input layers	INPUT	[raster] [list]	List of input raster layers
Reference layer	REF_LAYER	[raster]	The reference layer to create the output layer from (extent, CRS, pixel dimensions)
Treat nodata values as false	NODATA_AS_FALSE	[boolean] Default: False	Treat nodata values in the input files as 0 when performing the operation
Output no data value	NO_DATA	[number] Default: -9999.0	Value to use for nodata in the output layer
Output data type	DATA_TYPE	[enumeration] Default: 5	Output raster data type. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Output layer	OUTPUT	[raster]	Output raster layer

Outputs

Label	名称	Type	Description
Extent	EXTENT	[extent]	The extent of the output raster layer
CRS authority identifier	CRS_AUTHID	[crs]	The coordinate reference system of the output raster layer
Width in pixels	WIDTH_IN_PIXELS	[integer]	The width in pixels of the output raster layer
Height in pixels	HEIGHT_IN_PIXELS	[integer]	The height in pixels of the output raster layer
Total pixel count	TOTAL_PIXEL_COUNT	[integer]	The count of pixels in the output raster layer
NODATA pixel count	NODATA_PIXEL_COUNT	[integer]	The count of nodata pixels in the output raster layer
True pixel count	TRUE_PIXEL_COUNT	[integer]	The count of True pixels (value = 1) in the output raster layer

下页继续

表 24.36 - 续上页

Label	名称	Type	Description
False pixel count	FALSE_PIXEL_COUNT	[integer]	The count of False pixels (value = 0) in the output raster layer
Output layer	OUTPUT	[raster]	Output raster layer containing the result

Python code

Algorithm ID: qgis:rasterbooleanor

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster calculator

Performs algebraic operations using raster layers.

The resulting layer will have its values computed according to an expression. The expression can contain numerical values, operators and references to any of the layers in the current project.

注解: When using the calculator in *The batch processing interface* or from the *QGIS Python console* the files to use have to be specified. The corresponding layers are referred using the base name of the file (without the full path). For instance, if using a layer at `path/to/my/rasterfile.tif`, the first band of that layer will be referred as `rasterfile.tif@1`.

参见:

Raster Calculator

Parameters

Label	名称	Type	Description
Layers	GUI only		Shows the list of all raster layers loaded in the legend. These can be used to fill the expression box (double click to add). Raster layers are referred by their name and the number of the band: <code>layer_name@band_number</code> . For instance, the first band from a layer named DEM will be referred as <code>DEM@1</code> .
Operators	GUI only		Contains some calculator like buttons that can be used to fill the expression box.
Expression	EXPRESSION	[string]	Expression that will be used to calculate the output raster layer. You can use the operator buttons provided to type directly the expression in this box.

下页继续

表 24.37 - 续上页

Label	名称	Type	Description
Predefined expressions	GUI only		You can use the predefined NDVI expression or you can define new expressions for calculations. The <i>Add...</i> button loads a defined expression (and lets you set the parameters). The <i>Save...</i> button lets you define a new expression.
Reference layer(s) (used for automated extent, cellsize, and CRS) Optional	LAYERS	[raster] [list]	Layer(s) that will be used to fetch extent, cell size and CRS. By choosing the layer in this box you avoid filling in all the other parameters by hand. Raster layers are referred by their name and the number of the band: <code>layer_name@band_number</code> . For instance, the first band from a layer named DEM will be referred as <code>DEM@1</code> .
Cell size (use 0 or empty to set it automatically) Optional	CELLSIZE	[number]	Cell size of the output raster layer. If the cell size is not specified, the minimum cell size of the selected reference layer(s) will be used. The cell size will be the same for the X and Y axes.
Output extent (xmin, xmax, ymin, ymax) Optional	EXTENT	[extent]	Extent of the output raster layer. If the extent is not specified, the minimum extent that covers all the selected reference layers will be used.
Output CRS Optional	CRS	[crs]	CRS of the output raster layer. If the output CRS is not specified, the CRS of the first reference layer will be used.
Output	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output	OUTPUT	[raster]	Output raster file with the calculated values.

Python code

Algorithm ID: `qgis:rastercalculator`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Raster layer statistics

Calculates basic statistics from the values in a given band of the raster layer. The output is loaded in the *Processing* ► *Results viewer* menu.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Band number	BAND	[raster band] Default: The first band of the input layer	If the raster is multiband, choose the band you want to get statistics for.
Output	OUTPUT_HTML_FILE	[html] Default: [Save to temporary file]	Specification of the output file: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Maximum value	MAX	[number]	
Mean value	MEAN	[number]	
Minimum value	MIN	[number]	
Output	OUTPUT_HTML_FILE	[html]	The output file contains the following information: <ul style="list-style-type: none"> • Analyzed file: path of the raster layer • Minimum value: minimum value of the raster • Maximum value: maximum value of the raster • Range: difference between the maximum and minimum values • Sum: total sum of the values • Mean value: mean of the values • Standard deviation: standard deviation of the values • Sum of the squares: sum of the squared differences of each observation from the overall mean
Range	RANGE	[number]	
Standard deviation	STD_DEV	[number]	
Sum	SUM	[number]	
Sum of the squares	SUM_OF_SQUARES	[number]	

Python code

Algorithm ID: qgis:rasterlayerstatistics

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster layer unique values report

Returns the count and area of each unique value in a given raster layer.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Band number	BAND	[raster band] Default: The first band of the input layer	If the raster is multiband, choose the band you want to get statistics for.
Unique values report	OUTPUT_HTML_FILE	[file] Default: [Save to temporary file]	Specification of the output file: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.
Unique values table	OUTPUT_TABLE	[table] Default: [Skip output]	Specification of the table for unique values: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer • Save to File... • Save to GeoPackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
CRS authority identifier	CRS_AUTHID	[crs]	
Extent	EXTENT	[extent]	
Height in pixels	HEIGHT_IN_PIXEL	[number]	
NODATA pixel count	NODATA_PIXEL_COUNT	[number]	
Total pixel count	TOTAL_PIXEL_COUNT	[number]	

下页继续

表 24.40 - 续上页

Label	名称	Type	Description
Unique values report	OUTPUT_HTML_FILE	[html]	The output HTML file contains the following information: <ul style="list-style-type: none"> Analyzed file: the path of the raster layer Extent: xmin, ymin, xmax, ymax coordinates of the extent Projection: projection of the layer Width in pixels: number of columns and pixel width size Height in pixels: number of rows and pixel width size Total pixel count: count of all the pixels NODATA pixel count: count of pixels with NODATA value
Unique values table	OUTPUT_TABLE	[table]	A table with three columns: <ul style="list-style-type: none"> <i>value</i>: pixel value <i>count</i>: count of pixels with this value <i>m²</i>: total area in square meters of pixels with this value.
Width in pixels	WIDTH_IN_PIXELS	[number]	

Python code

Algorithm ID: qgis:rasterlayeruniquevaluesreport

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Raster layer zonal statistics

Calculates statistics for a raster layer's values, categorized by zones defined in another raster layer.

参见:

[Zonal statistics](#)

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[raster]	Input raster layer
Band number	BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband choose the band for which you want to calculate the statistics.

下页继续

表 24.41 - 续上页

Label	名称	Type	Description
Zones layer	ZONES	[raster]	Raster layer defining zones. Zones are given by contiguous pixels having the same pixel value.
Zones band number	ZONES_BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band that defines the zones
Reference layer Optional	REF_LAYER	[enumeration] Default: 0	Raster layer used to calculate the centroids that will be used as reference when determining the zones in the output layer. One of: <ul style="list-style-type: none"> • 0 --- Input layer • 1 --- Zones layer
Statistics	OUTPUT_TABLE	[table]	Table with the calculated statistics

Outputs

Label	名称	Type	Description
CRS authority identifier	CRS_AUTHID	[crs]	
Extent	EXTENT	[extent]	
Height in pixels	HEIGHT_IN_PIXEL	[number]	
NODATA pixel count	NODATA_PIXEL_CO	[number]	
Statistics	OUTPUT_TABLE	[table]	The output layer contains the following information for each zone : <ul style="list-style-type: none"> • Area: the area in square raster units in the zone; • Sum: the total sum of the pixel values in the zone; • Count: the number of pixels in the zone; • Min: the minimum pixel value in the zone; • Max: the maximum pixel value in the zone; • Mean: the mean of the pixel values in the zone;
Total pixel count	TOTAL_PIXEL_CO	[number]	
Width in pixels	WIDTH_IN_PIXELS	[number]	

Python code

Algorithm ID: qgis:rasterlayerzonalstats

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster surface volume

Calculates the volume under a raster surface relative to a given base level. This is mainly useful for Digital Elevation Models (DEM).

Parameters

Label	名称	Type	Description
INPUT layer	INPUT	[raster]	Input raster, representing a surface
Band number	BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band that shall define the surface.
Base level	LEVEL	[number] Default: 0.0	Define a base or reference value. This base is used in the volume calculation according to the <code>Method</code> parameter (see below).
Method	METHOD	[enumeration] Default: 0	Define the method for the volume calculation given by the difference between the raster pixel value and the <code>Base level</code> . Options: <ul style="list-style-type: none"> 0 --- Count Only Above Base Level: only pixels above the base level will add to the volume. 1 --- Count Only Below Base Level: only pixels below the base level will add to the volume. 2 --- Subtract Volumes Below Base level: pixels above the base level will add to the volume, pixels below the base level will subtract from the volume. 3 --- Add Volumes Below Base level: Add the volume regardless whether the pixel is above or below the base level. This is equivalent to sum the absolute values of the difference between the pixel value and the base level.

下页继续

表 24.43 - 续上页

Label	名称	Type	Description
Surface volume report	OUTPUT_HTML_FILE	[html] Default: [Save to temporary file]	Specification of the output HTML report. One of: <ul style="list-style-type: none"> • Skip output • Save to Temporary File • Save to File... The file encoding can also be changed here.
Surface volume table	OUTPUT_TABLE	[table] Default: [Skip output]	Specification of the output table. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Volume	VOLUME	[number]	The calculated volume
Area	AREA	[number]	The area in square map units
Pixel_count	PIXEL_COUNT	[number]	The total number of pixels that have been analyzed
Surface volume report	OUTPUT_HTML_FILE	[html]	The output report (containing volume, area and pixel count) in HTML format
Surface volume table	OUTPUT_TABLE	[table]	The output table (containing volume, area and pixel count)

Python code

Algorithm ID: qgis:rastersurfacevolume

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Reclassify by layer

Reclassifies a raster band by assigning new class values based on the ranges specified in a vector table.

Parameters

Label	名称	Type	Description
Raster layer	INPUT_RASTER	[raster]	Raster layer to reclassify
Band number	RASTER_BAND	[raster band] Default: The first band of the raster layer	If the raster is multiband, choose the band you want to reclassify.
Layer containing class breaks	INPUT_TABLE	[vector: any]	Vector layer containing the values to use for classification.
Minimum class value field	MIN_FIELD	[tablefield: numeric]	Field with the minimum value of the range for the class.
Maximum class value field	MAX_FIELD	[tablefield: numeric]	Field with the maximum value of the range for the class.
Output value field	VALUE_FIELD	[tablefield: numeric]	Field with the value that will be assigned to the pixels that fall in the class (between the corresponding min and max values).
Output no data value	NO_DATA	[number] Default: -9999.0	Value to apply to no data values.
Range boundaries	RANGE_BOUNDARIES	[enumeration] Default: 0	Defines comparison rules for the classification. Options: <ul style="list-style-type: none"> • 0 --- min < value <= max • 1 --- min <= value < max • 2 --- min <= value <= max • 3 --- min < value < max
Use no data when no range matches value	NODATA_FOR_MISMATCH	[boolean] Default: False	Values that do not belong to a class will result in the no data value. If False, the original value is kept.
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Reclassified raster	OUTPUT	[raster]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Reclassified raster	OUTPUT	[raster]	Output raster layer with reclassified band values

Python code

Algorithm ID: qgis:reclassifybylayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Reclassify by table

Reclassifies a raster band by assigning new class values based on the ranges specified in a fixed table.

Parameters

Label	名称	Type	Description
Raster layer	INPUT_RASTER	[raster]	Raster layer to reclassify
Band number	RASTER_BAND	[raster band] Default: 1	Raster band for which you want to recalculate values.
Reclassification table	TABLE	[table]	A 3-columns table to fill with the values to set the boundaries of each class (Minimum and Maximum) and the new Value to assign to the band values that fall in the class.
Output no data value	NO_DATA	[number] Default: -9999.0	Value to apply to no data values.
Range boundaries	RANGE_BOUNDARIES	[enumeration] Default: 0	Defines comparison rules for the classification. Options: <ul style="list-style-type: none"> 0 --- min < value <= max 1 --- min <= value < max 2 --- min <= value <= max 3 --- min < value < max
Use no data when no range matches value	NODATA_FOR_MISSING	[boolean] Default: False	Applies the no data value to band values that do not fall in any class. If False, the original value is kept.

下一页继续

表 24.45 - 续上页

Label	名称	Type	Description
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the format of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Reclassified raster	OUTPUT	[raster] Default: '[Save to temporary file]'	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here

Outputs

Label	名称	Type	Description
Reclassified raster	OUTPUT	[raster] Default: '[Save to temporary file]'	The output raster layer.

Python code

Algorithm ID: qgis:reclassifybytable

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Rescale raster

Rescales raster layer to a new value range, while preserving the shape (distribution) of the raster's histogram (pixel values). Input values are mapped using a linear interpolation from the source raster's minimum and maximum pixel values to the destination minimum and maximum pixel range.

By default the algorithm preserves the original NODATA value, but there is an option to override it.

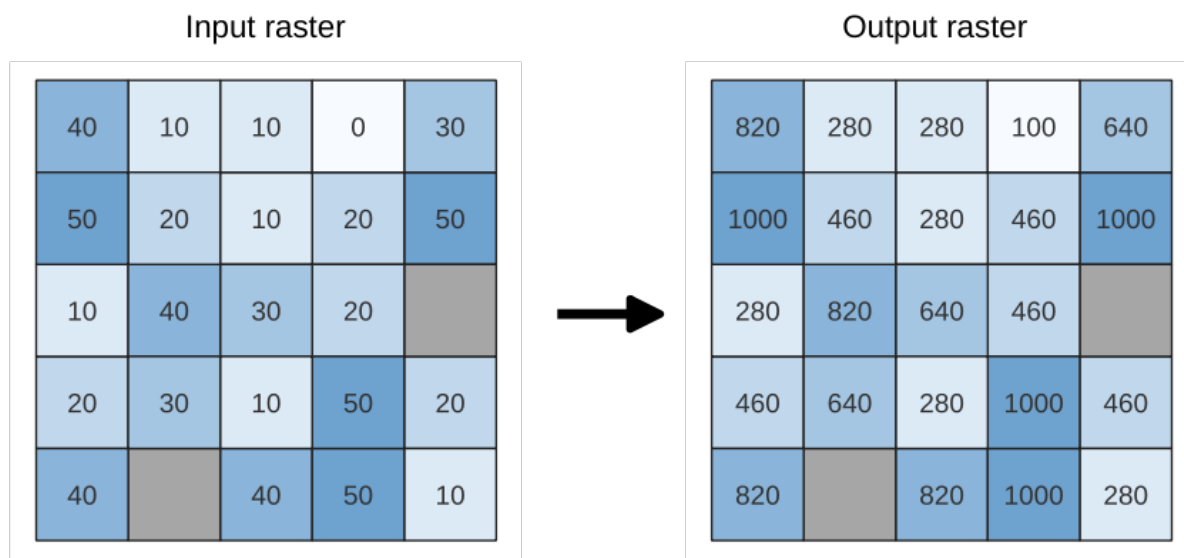


图 24.13: Rescaling values of a raster layer from [0 - 50] to [100 - 1000]

Parameters

Label	名称	Type	Description
Input Raster	INPUT	[raster]	Raster layer to use for rescaling
Band number	Band	[raster band] Default: The first band of the input layer	If the raster is multiband, choose a band.
New minimum value	MINIMUM	[number] Default value: 0.0	Minimum pixel value to use in the rescaled layer
New maximum value	MAXIMUM	[number] Default value: 255.0	Maximum pixel value to use in the rescaled layer
New NODATA value Optional	NODATA	[number] Default value: Not set	Value to assign to the NO-DATA pixels. If unset, original NODATA values are preserved.
Rescaled	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Outputs

Label	名称	Type	Description
Rescaled	OUTPUT	[raster]	Output raster layer with rescaled band values

Python code

Algorithm ID: native:rescaleraster

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Round raster

Rounds the cell values of a raster dataset according to the specified number of decimals.

Alternatively, a negative number of decimal places may be used to round values to powers of a base n. For example, with a Base value n of 10 and Decimal places of -1, the algorithm rounds cell values to multiples of 10, -2 rounds to multiples of 100, and so on. Arbitrary base values may be chosen, the algorithm applies the same multiplicative principle. Rounding cell values to multiples of a base n may be used to generalize raster layers.

The algorithm preserves the data type of the input raster. Therefore byte/integer rasters can only be rounded to multiples of a base n, otherwise a warning is raised and the raster gets copied as byte/integer raster.

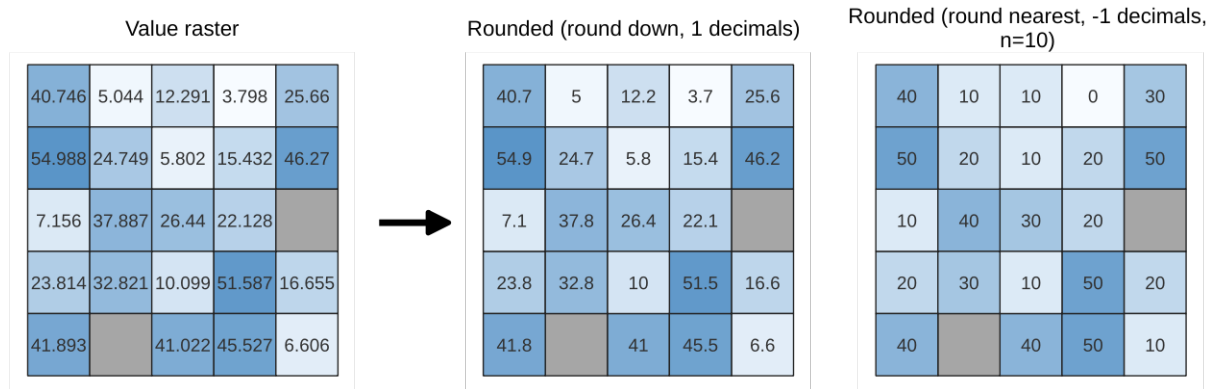


图 24.14: Rounding values of a raster

Parameters

Basic parameters

Label	名称	Type	Description
Input raster	INPUT	[raster]	The raster to process.
Band number	BAND	[number] Default: 1	The band of the raster
Rounding direction	ROUNDING_DIRECTION	[int] Default: 1	How to choose the target rounded value. Options are: 0 - Round up 1 - Round to nearest 2 - Round down
Number of decimals places	DECIMAL_PLACES	[number] Default: 2	Number of decimals places to round to. Use negative values to round cell values to a multiple of a base n

下页继续

表 24.46 - 续上页

Label	名称	Type	Description
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output file. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Base n for rounding to multiples of n	BASE_N	[number] Default: 10	When the <code>DECIMAL_PLACES</code> parameter is negative, raster values are rounded to multiples of the base n value

Outputs

Label	名称	Type	Description
Output raster	OUTPUT	[raster]	The output raster layer with values rounded for the selected band.

Python code

Algorithm ID: `native:roundrastervalues`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Sample raster values

Extracts raster values at the point locations. If the raster layer is multiband, each band is sampled.

The attribute table of the resulting layer will have as many new columns as the raster layer band count.

Parameters

Label	名称	Type	Description
Input Point Layer	INPUT	[vector: point]	Point vector layer to use for sampling
Raster Layer to sample	RASTERCOPY	[raster]	Raster layer to sample at the given point locations.
Output column prefix	COLUMN_PREFIX	[string] Default: 'rvalue'	Prefix for the names of the added columns.
Sampled Points Optional	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output layer containing the sampled values. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Sampled Points Optional	OUTPUT	[vector: point]	The output layer containing the sampled values.

Python code

Algorithm ID: qgis:rastersampling

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Zonal histogram

Appends fields representing counts of each unique value from a raster layer contained within polygon features.

The output layer attribute table will have as many fields as the unique values of the raster layer that intersects the polygon(s).

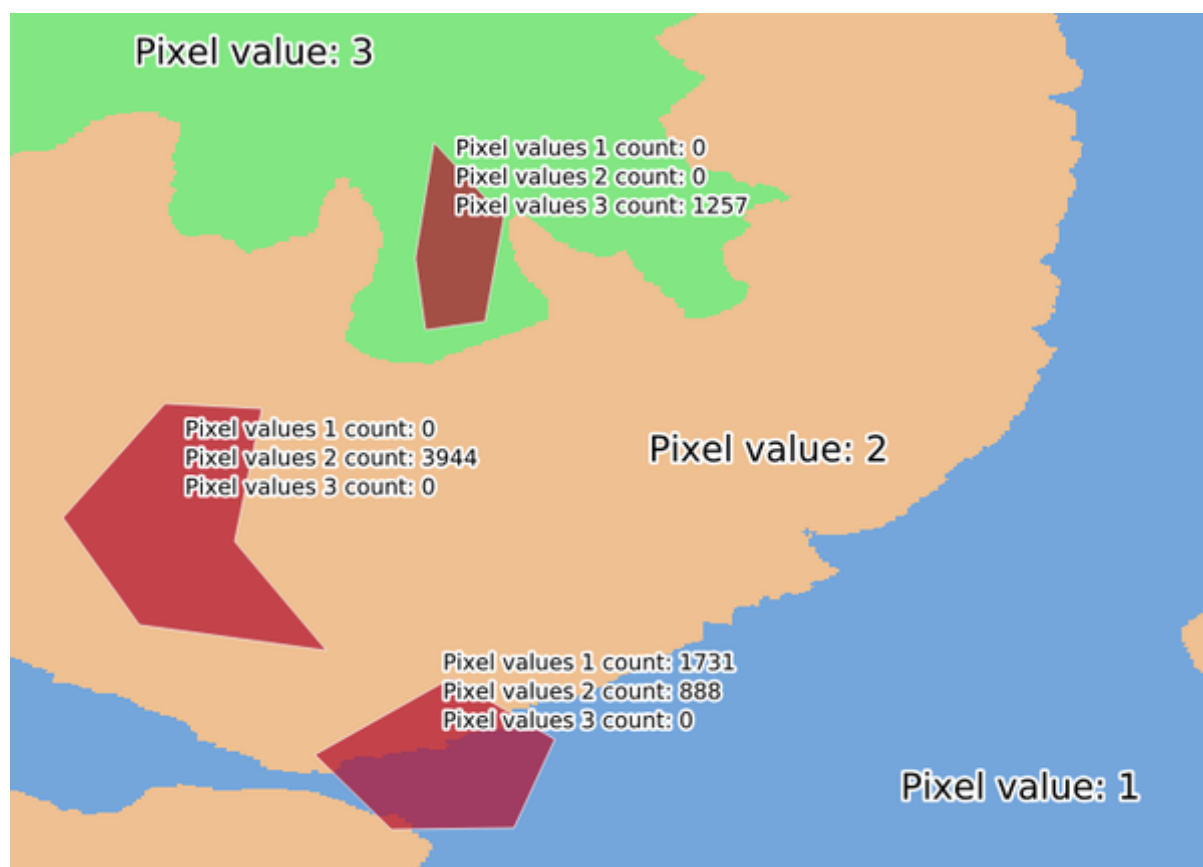


图 24.15: Raster layer histogram example

Parameters

Label	名称	Type	Description
Raster layer	INPUT_RASTER	[raster]	Input raster layer.
Band number	RASTER_BAND	[raster band] Default: The first band of the input layer	If the raster is multiband, choose a band.
Vector layer containing zones	INPUT_VECTOR	[vector: polygon]	Vector polygon layer that defines the zones.
Output column prefix	COLUMN_PREFIX Optional	[string] Default: 'HISTO_'	Prefix for the output columns names.
Output zones	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector polygon layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output zones Optional	OUTPUT	[vector: polygon] Default: [Create temporary layer]	The output vector polygon layer.

Python code

Algorithm ID: qgis:zonalhistogram

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Zonal statistics

Calculates statistics of a raster layer for each feature of an overlapping polygon vector layer.

Prior to QGIS 3.16, the algorithm edited the layer in-place, adding the new statistics fields to it. Now, it outputs a new layer with these statistics.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Vector polygon layer that contains the zones.
Raster layer	INPUT_RASTER	[raster]	Input raster layer.
Raster band	RASTER_BAND	[raster band] Default: The first band of the input layer	If the raster is multiband, choose a band for the statistics.
Output column prefix	COLUMN_PREFIX	[string] Default: '_'	Prefix for the output columns names.
Statistics to calculate	STATISTICS	[enumeration] [list] Default: [0,1,2]	List of statistical operator for the output. Options: <ul style="list-style-type: none"> • 0 --- Count • 1 --- Sum • 2 --- Mean • 3 --- Median • 4 --- St. dev. • 5 --- Minimum • 6 --- Maximum • 7 --- Range • 8 --- Minority • 9 --- Majority • 10 --- Variety • 11 --- Variance

下页继续

表 24.48 - 续上页

Label	名称	Type	Description
Zonal Statistics NEW in 3.16	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector polygon layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... • Append to Layer... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Zonal Statistics NEW in 3.16	OUTPUT	[vector: polygon]	The zone vector layer with added statistics.

Python code

Algorithm ID: qgis:zonalstatisticsfb

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.1.10 Raster Creation

Create constant raster layer

Generates raster layer for given extent and cell size filled with the specified value.

Additionally an output data type can be specified. The algorithm will abort if a value has been entered that cannot be represented by the selected output raster data type.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRIS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 0.1	Pixel size (X=Y) in map units. Minimum value: 0.01
Constant value	NUMBER	[number] Default: 1	Constant pixel value for the output raster layer.
Constant	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 5	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Integer16 • 2 --- Unsigned Integer16 • 3 --- Integer32 • 4 --- Unsigned Integer32 • 5 --- Float32 • 6 --- Float64

Outputs

Label	Name	Type	Description
Constant	OUTPUT	[raster]	Raster covering the desired extent with the specified pixel size and value.

Python code

Algorithm ID: native:createconstantrasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create random raster layer (binomial distribution)

Generates a raster layer for given extent and cell size filled with binomially distributed random values.

By default, the values will be chosen given an N of 10 and a probability of 0.5. This can be overridden by using the advanced parameter for N and probability. The raster data type is set to Integer types (Integer16 by default). The binomial distribution random values are defined as positive integer numbers. A floating point raster will represent a cast of integer values to floating point.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRIS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 0.1	Pixel size (X=Y) in map units. Minimum value: 0.01
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 0	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Integer16 • 1 --- Unsigned Integer16 • 2 --- Integer32 • 3 --- Unsigned Integer32 • 4 --- Float32 • 5 --- Float64
N	N	[number] Default: 10	
Probability	PROBABILITY	[number] Default: 0.5	

Outputs

Label	Name	Type	Description
Output raster	OUTPUT	[raster]	Raster covering the desired extent with the cell size filled with random values

Python code

Algorithm ID: native:createrandombinomialrasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create random raster layer (exponential distribution)

Generates a raster layer for given extent and cell size filled with exponentially distributed random values.

By default, the values will be chosen given a lambda of 1.0. This can be overridden by using the advanced parameter for lambda. The raster data type is set to Float32 by default as the exponential distribution random values are floating point numbers.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRIS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 1.0	Pixel size (X=Y) in map units. Minimum value: 0.01
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 0	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Float32 • 1 --- Float64
Lambda	LAMBDA	[number] Default: 1.0	

Outputs

Label	Name	Type	Description
Output raster	OUTPUT	[raster]	Raster covering the desired extent with the cell size filled with random values

Python code

Algorithm ID: native:createrandomexponentialrasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Create random raster layer (gamma distribution)

Generates a raster layer for given extent and cell size filled with gamma distributed random values.

By default, the values will be chosen given an alpha and beta value of 1.0. This can be overridden by using the advanced parameter for alpha and beta. The raster data type is set to Float32 by default as the gamma distribution random values are floating point numbers.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRIS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 1.0	Pixel size (X=Y) in map units. Minimum value: 0.01
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 0	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Float32 • 1 --- Float64
Alpha	ALPHA	[number] Default: 1.0	
Beta	BETA	[number] Default: 1.0	

Outputs

Label	Name	Type	Description
Output raster	OUTPUT	[raster]	Raster covering the desired extent with the cell size filled with randomly distributed values

Python code

Algorithm ID: native:createrandomgammarasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create random raster layer (geometric distribution)

Generates a raster layer for given extent and cell size filled with geometrically distributed random values.

By default, the values will be chosen given a probability of 0.5. This can be overridden by using the advanced parameter for mean value. The raster data type is set to Integer types (Integer16 by default). The geometric distribution random values are defined as positive integer numbers. A floating point raster will represent a cast of integer values to floating point.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> Use Canvas Extent Select Extent on Canvas Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRIS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 1.0	Pixel size (X=Y) in map units. Minimum value: 0.01
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> Save to a Temporary File Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 0	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Integer16 • 1 --- Unsigned Integer16 • 2 --- Integer32 • 3 --- Unsigned Integer32 • 4 --- Float32 • 5 --- Float64
Probability	PROBABILITY	[number] Default: 0.5	

Outputs

Label	Name	Type	Description
Output raster	OUTPUT	[raster]	Raster covering the desired extent with the cell size filled with randomly distributed values

Python code

Algorithm ID: native:createrandomgeometricrasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create random raster layer (negative binomial distribution)

Generates a raster layer for given extent and cell size filled with negative binomially distributed random values.

By default, the values will be chosen given a distribution parameter *k* of 10.0 and a probability of 0.5. This can be overridden by using the advanced parameters for *k* and probability. The raster data type is set to Integer types (Integer16 by default). The negative binomial distribution random values are defined as positive integer numbers. A floating point raster will represent a cast of integer values to floating point.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRIS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 1.0	Pixel size (X=Y) in map units. Minimum value: 0.01
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 0	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Integer16 • 1 --- Unsigned Integer16 • 2 --- Integer32 • 3 --- Unsigned Integer32 • 4 --- Float32 • 5 --- Float64
Distribution parameter k	K_PARAMETER	[number] Default: 10	
Probability	PROBABILITY	[number] Default: 0.5	

Outputs

Label	Name	Type	Description
Output raster	OUTPUT	[raster]	Raster covering the desired extent with the cell size filled with randomly distributed values

Python code

Algorithm ID: native:createrandomnegativebinomialrasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create random raster layer (normal distribution)

Generates a raster layer for given extent and cell size filled with normally distributed random values.

By default, the values will be chosen given a mean of 0.0 and a standard deviation of 1.0. This can be overridden by using the advanced parameters for mean and standard deviation value. The raster data type is set to Float32 by default as the normal distribution random values are floating point numbers.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> Use Canvas Extent Select Extent on Canvas Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 1.0	Pixel size (X=Y) in map units. Minimum value: 0.01
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> Save to a Temporary File Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 0	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> 0 --- Float32 1 --- Float64
Mean of normal distribution	MEAN	[number] Default: 0.0	
Standard deviation of normal distribution	STDDEV	[number] Default: 1.0	

Outputs

Label	Name	Type	Description
Output raster	OUTPUT	[raster]	Raster covering the desired extent with the cell size filled with randomly distributed values

Python code

Algorithm ID: native:createrandomnormalrasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create random raster layer (poisson distribution)

Generates a raster layer for given extent and cell size filled with poisson distributed random values.

By default, the values will be chosen given a mean of 1.0. This can be overridden by using the advanced parameter for mean value. The raster data type is set to Integer types (Integer16 by default). The poisson distribution random values are positive integer numbers. A floating point raster will represent a cast of integer values to floating point.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> Use Canvas Extent Select Extent on Canvas Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRIS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 1.0	Pixel size (X=Y) in map units. Minimum value: 0.01
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> Save to a Temporary File Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 0	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Integer16 • 1 --- Unsigned Integer16 • 2 --- Integer32 • 3 --- Unsigned Integer32 • 4 --- Float32 • 5 --- Float64
Mean	MEAN	[number] Default: 1.0	

Outputs

Label	Name	Type	Description
Output raster	OUTPUT	[raster]	Raster covering the desired extent with the cell size filled with randomly distributed values

Python code

Algorithm ID: native:createrandompoissonrasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create random raster layer (uniform distribution)

Generates a raster layer for given extent and cell size filled with random values.

By default, the values will range between the minimum and maximum value of the specified output raster type. This can be overridden by using the advanced parameters for lower and upper bound value. If the bounds have the same value or both are zero (default) the algorithm will create random values in the full value range of the chosen raster data type. Choosing bounds outside the acceptable range of the output raster type will abort the algorithm.

Parameters

Basic parameters

Label	Name	Type	Description
Desired extent	EXTENT	[extent]	Specify the extent (xmin, xmax, ymin, ymax) of the output raster layer. One of: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent... It will internally be extended to a multiple of the tile size.
Target CRS	TARGET_CRIS	[crs] Default: Project CRS	CRS for the output raster layer
Pixel size	PIXEL_SIZE	[number] Default: 1.0	Pixel size (X=Y) in map units. Minimum value: 0.01
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	Name	Type	Description
Output raster data type	OUTPUT_TYPE Default: 5	[enumeration]	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Integer16 • 2 --- Unsigned Integer16 • 3 --- Integer32 • 4 --- Unsigned Integer32 • 5 --- Float32 • 6 --- Float64
Lower bound for random number range	LOWER_BOUND	[number] Default: 0.0	
Upper bound for random number range	UPPER_BOUND	[number] Default: 0.0	

Outputs

Label	Name	Type	Description
Output raster	OUTPUT	[raster]	Raster covering the desired extent with the cell size filled with randomly distributed values

Python code

Algorithm ID: native:createrandomuniformrasterlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.1.11 Raster terrain analysis

Aspect

Calculates the aspect of the Digital Terrain Model in input. The final aspect raster layer contains values from 0 to 360 that express the slope direction, starting from north (0°) and continuing clockwise.

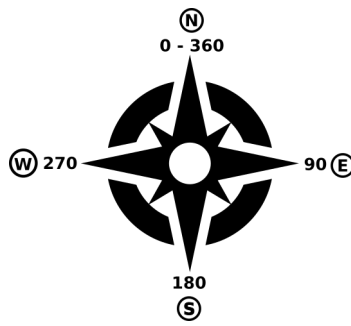


图 24.16: Aspect values

The following picture shows the aspect layer reclassified with a color ramp:

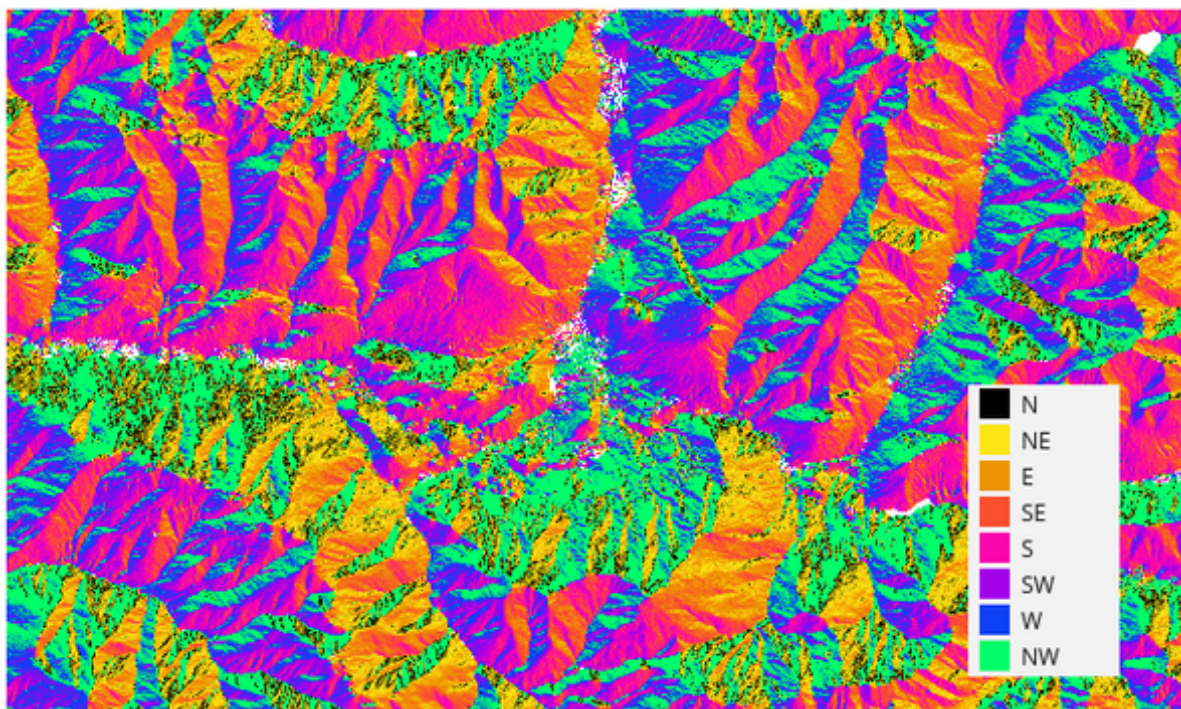


图 24.17: Aspect layer reclassified

Parameters

Label	名称	Type	Description
Elevation layer	INPUT	[raster]	Digital Terrain Model raster layer
Z factor	Z_FACTOR	[number] Default: 1.0	Vertical exaggeration. This parameter is useful when the Z units differ from the X and Y units, for example feet and meters. You can use this parameter to adjust for this. The default is 1 (no exaggeration).
Aspect	OUTPUT	[raster]	Specify the output aspect raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary Layer (TEMPORARY_OUTPUT) • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Aspect	OUTPUT	[raster]	The output aspect raster layer

Python code

Algorithm ID: qgis:aspect

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Hillshade

Calculates the hillshade raster layer given an input Digital Terrain Model.

The shading of the layer is calculated according to the sun position: you have the options to change both the horizontal angle (azimuth) and the vertical angle (sun elevation) of the sun.

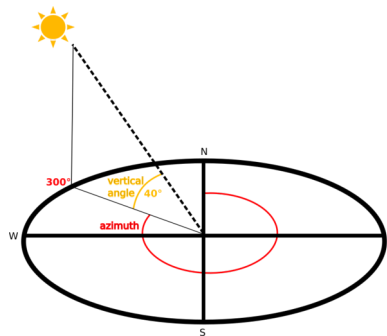


图 24.18: Azimuth and vertical angle

The hillshade layer contains values from 0 (complete shadow) to 255 (complete sun). Hillshade is used usually to better understand the relief of the area.

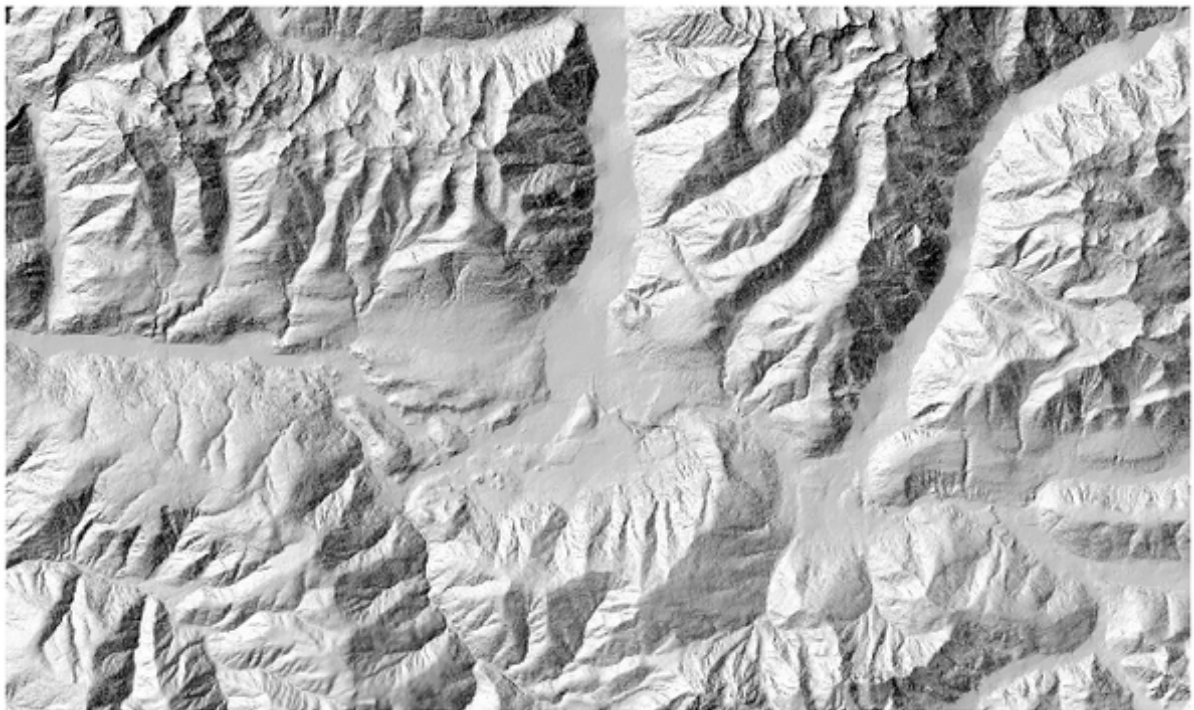


图 24.19: Hillshade layer with azimuth 300 and vertical angle 45

Particularly interesting is to give the hillshade layer a transparency value and overlap it with the elevation raster:

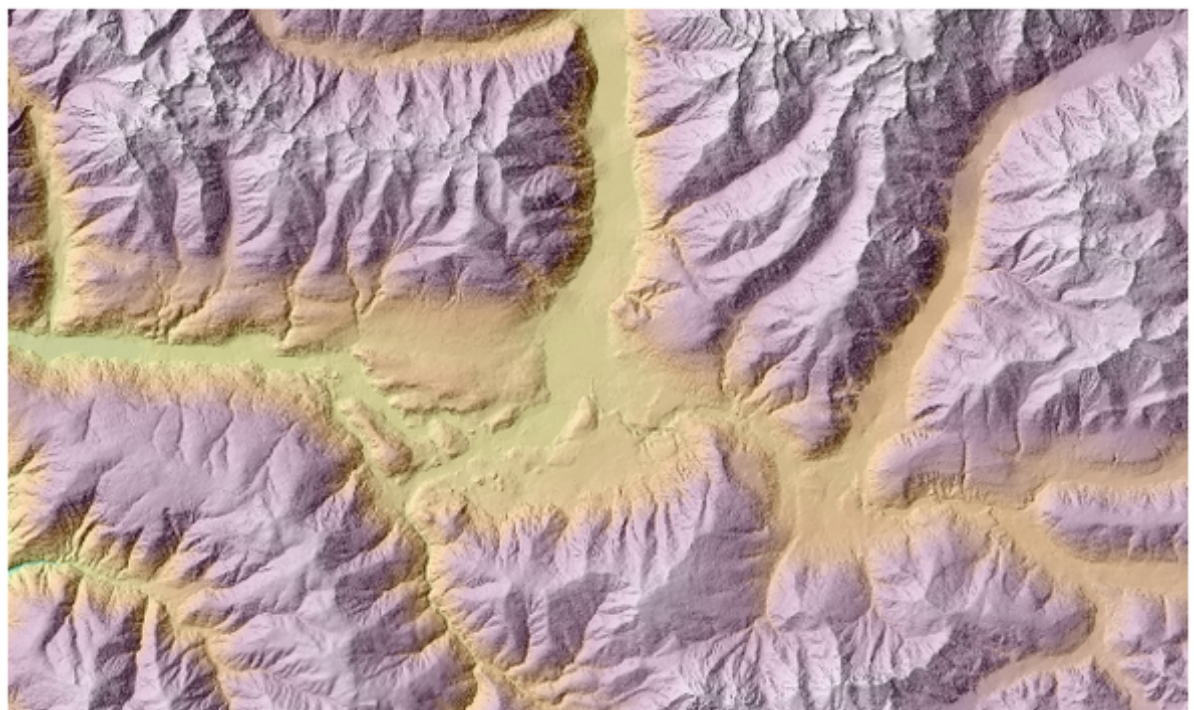


图 24.20: Overlapping the hillshade with the elevation layer

Parameters

Label	名称	Type	Description
Elevation layer	INPUT	[raster]	Digital Terrain Model raster layer
Z factor	Z_FACTOR	[number] Default: 1.0	Vertical exaggeration. This parameter is useful when the Z units differ from the X and Y units, for example feet and meters. You can use this parameter to adjust for this. Increasing the value of this parameter will exaggerate the final result (making it look more "hilly"). The default is 1 (no exaggeration).
Azimuth (horizontal angle)	AZIMUTH	[number] Default: 300.0	Set the horizontal angle (in degrees) of the sun (clockwise direction). Range: 0 to 360. 0 is north.
Vertical angle	V_ANGLE	[number] Default: 40.0	Set the vertical angle (in degrees) of the sun, that is the height of the sun. Values can go from 0 (minimum elevation) to 90 (maximum elevation).
Hillshade	OUTPUT	[raster]	Specify the output hillshade raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary Layer (TEMPORARY_OUTPUT) • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Hillshade	OUTPUT	[raster]	The output hillshade raster layer

Python code

Algorithm ID: qgis:hillshade

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Hypsometric curves

Calculates hypsometric curves for an input Digital Elevation Model. Curves are produced as CSV files in an output folder specified by the user.

A hypsometric curve is a cumulative histogram of elevation values in a geographical area.

You can use hypsometric curves to detect differences in the landscape due to the geomorphology of the territory.

Parameters

Label	名称	Type	Description
DEM to analyze	INPUT_DEM	[raster]	Digital Terrain Model raster layer to use for calculating altitudes
Boundary layer	BOUNDARY_LAYER	[vector: polygon]	Polygon vector layer with boundaries of areas used to calculate hypsometric curves
Step	STEP	[number] Default: 100.0	Vertical distance between curves
Use % of area instead of absolute value	USE_PERCENTAGE	[boolean] Default: False	Write area percentage to “Area” field of the CSV file instead of the absolute area
Hypsometric curves	OUTPUT_DIRECTORY	[folder]	Specify the output folder for the hypsometric curves. One of: <ul style="list-style-type: none"> • Save to a Temporary Layer (TEMPORARY_OUTPUT) • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Hypsometric curves	OUTPUT_DIRECTORY	[folder]	Directory containing the files with the hypsometric curves. For each feature from the input vector layer, a CSV file with area and altitude values will be created. The file names start with <code>histogram_</code> , followed by layer name and feature ID.

	A	B
1	Area	Elevation
2	177475194.383	307
3	233206029.24	407
4	295553735.793	507
5	394718815.615	607
6	501801102.615	707
7	624399019.792	807
8	828877274.39	907
9	1042693465.68	1007
10	1277373021.81	1107
11	1556443975.41	1207
12	1888617494.27	1307
13	2248520437.31	1407
14	2627916813.17	1507
15	3010880212.04	1607
16	3411087555.34	1707

Python code

Algorithm ID: qgis:hypsometriccurves

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Relief

Creates a shaded relief layer from digital elevation data. You can specify the relief color manually, or you can let the algorithm choose automatically all the relief classes.



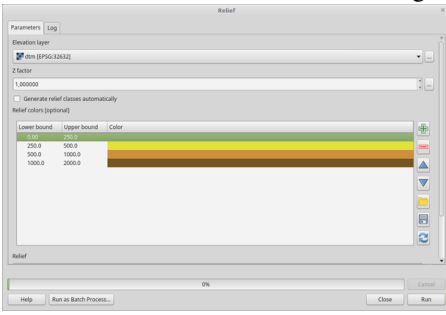
图 24.21: Relief layer

Parameters

Label	名称	Type	Description
Elevation layer	INPUT	[raster]	Digital Terrain Model raster layer
Z factor	Z_FACTOR	[number] Default: 1.0	Vertical exaggeration. This parameter is useful when the Z units differ from the X and Y units, for example feet and meters. You can use this parameter to adjust for this. Increasing the value of this parameter will exaggerate the final result (making it look more "hilly"). The default is 1 (no exaggeration).

下页继续

表 24.51 - 续上页

Label	名称	Type	Description
Generate relief classes automatically	AUTO_COLORS	[boolean] Default: False	If you check this option the algorithm will create all the relief color classes automatically
Relief colors Optional	COLORS	[table widget]	Use the table widget if you want to choose the relief colors manually. You can add as many color classes as you want: for each class you can choose the lower and upper bound and finally by clicking on the color row you can choose the color thanks to the color widget.  <p>图 24.22: Manually setting of relief color classes</p> <p>The buttons in the right side panel give you the chance to: add or remove color classes, change the order of the color classes already defined, open an existing file with color classes and save the current classes as file.</p>
Relief	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output relief raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary Layer (TEMPORARY_OUTPUT) • Save to File... The file encoding can also be changed here.
Frequency distribution	FREQUENCY_DISTRIBUTION	[table] Default: [Skip output]	Specify the CSV table for the output frequency distribution. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary Layer (TEMPORARY_OUTPUT) • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Relief	OUTPUT	[raster]	The output relief raster layer
Frequency distribution	OUTPUT	[table]	The output frequency distribution

Python code

Algorithm ID: qgis:relief

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Ruggedness index

Calculates the quantitative measurement of terrain heterogeneity described by Riley et al. (1999). It is calculated for every location, by summarizing the change in elevation within the 3x3 pixel grid.

Each pixel contains the difference in elevation from a center cell and the 8 cells surrounding it.

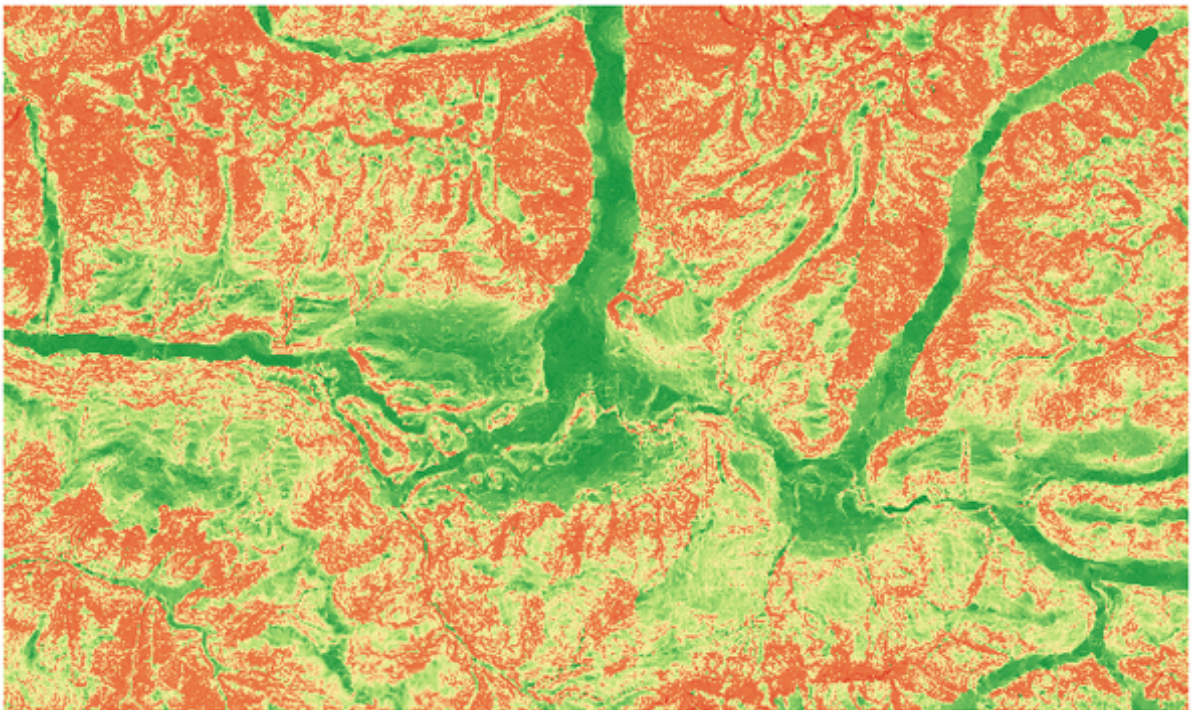


图 24.23: Ruggedness layer from low (red) to high values (green)

Parameters

Label	名称	Type	Description
Elevation layer	INPUT	[raster]	Digital Terrain Model raster layer
Z factor	Z_FACTOR	[number] Default: 1.0	Vertical exaggeration. This parameter is useful when the Z units differ from the X and Y units, for example feet and meters. You can use this parameter to adjust for this. Increasing the value of this parameter will exaggerate the final result (making it look more rugged). The default is 1 (no exaggeration).
Ruggedness	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output ruggedness raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary Layer (TEMPORARY_OUTPUT) • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Ruggedness	OUTPUT	[raster]	The output ruggedness raster layer

Python code

Algorithm ID: qgis:ruggednessindex

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Slope

Calculates the slope from an input raster layer. The slope is the angle of inclination of the terrain and is expressed in **degrees**.

In the following picture you can see to the left the DTM layer with the elevation of the terrain while to the right the calculated slope:

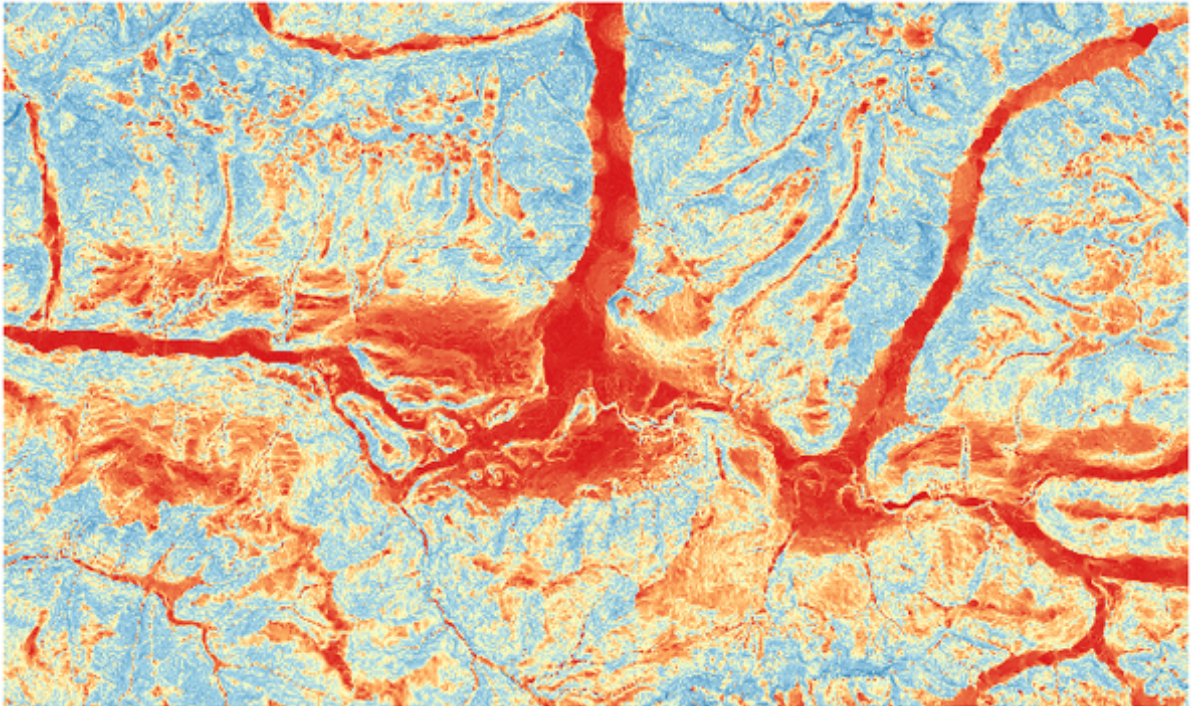


图 24.24: Flat areas in red, steep areas in blue

Parameters

Label	名称	Type	Description
Elevation layer	INPUT	[raster]	Digital Terrain Model raster layer
Z factor	Z_FACTOR	[number] Default: 1.0	Vertical exaggeration. This parameter is useful when the Z units differ from the X and Y units, for example feet and meters. You can use this parameter to adjust for this. Increasing the value of this parameter will exaggerate the final result (making it steeper). The default is 1 (no exaggeration).
Slope	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output slope raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary Layer (TEMPORARY_OUTPUT) • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Slope	OUTPUT	[raster]	The output slope raster layer

Python code

Algorithm ID: qgis:slope

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.12 Raster tools

Convert map to raster

Creates a raster image of map canvas content.

A *map theme* can be selected to render a predetermined set of layers with a defined style for each layer.

Alternatively, a single layer can be selected if no map theme is set.

If neither map theme nor layer is set, the current map content will be rendered. The minimum extent entered will internally be extended to be a multiple of the tile size.

Parameters

Label	名称	Type	Description
Minimum extent to render (xmin, xmax, ymin, ymax)	EXTENT	[extent]	Specify the extent of the output raster layer. One of: <ul style="list-style-type: none"> Use Canvas Extent Select Extent on Canvas Use Layer Extent... It will internally be extended to a multiple of the tile size.
Tile size	TILE_SIZE	[number] Default: 1024	Size of the tile of the output raster layer. Minimum value: 64.
Map units per pixel	MAP_UNITS_PER_PIXEL	[number] Default: 100.0	Pixel size (in map units). Minimum value: 0.0
Make background transparent	MAKE_BACKGROUND_TRANSPARENT	[boolean] Default: False	Allows exporting the map with a transparent background. Outputs an RGBA (instead of RGB) image if set to True.
Map theme to render Optional	MAP_THEME	[enumeration]	Use an existing <i>map theme</i> for the rendering.
Single layer to render Optional	LAYER	[enumeration]	Choose a single layer for the rendering

下页继续

表 24.52 - 续上页

Label	名称	Type	Description
Output layer	OUTPUT	[raster] Default: Save to temporary file	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output layer	OUTPUT	[raster]	Output raster layer

Python code

Algorithm ID: qgis:rasterize

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Fill NoData cells

Resets the NoData values in the input raster to a chosen value, resulting in raster dataset with no NoData pixels.

The algorithm respects the input raster data type, e.g. a floating point fill value will be truncated when applied to an integer raster.

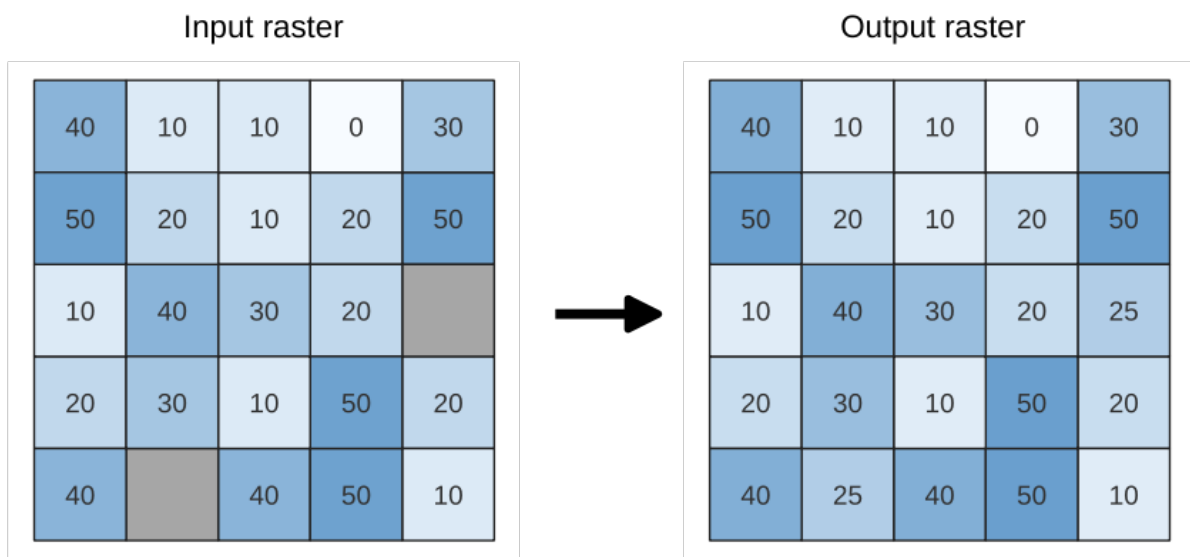


图 24.25: Filling NoData values (in grey) of a raster

Parameters

Label	名称	Type	Description
Input raster	INPUT	[raster]	The raster to process.
Band number	BAND	[number] Default: 1	The band of the raster
Fill value	FILL_VALUE	[number] Default: 1.0	Set the value to use for the NoData pixels
Output raster	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Outputs

Label	名称	Type	Description
Output raster	OUTPUT	[raster]	The output raster layer with filled data cells.

Python code

Algorithm ID: native:fillnodata

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Generate XYZ tiles (Directory)

Generates raster “XYZ” tiles using the current QGIS project as individual images to a directory structure.

Parameters

Label	名称	Type	Description
Extent (xmin, xmax, ymin, ymax)	EXTENT	[extent]	Specify the extent of the tiles. One of: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent... It will internally be extended to a multiple of the tile size.
Minimum zoom	ZOOM_MIN	[number] Default: 12	Minimum 0, maximum 25.
Maximum zoom	ZOOM_MAX	[number] Default: 12	Minimum 0, maximum 25.
DPI	DPI	[number] Default: 96	Minimum 48, maximum 600.

下页继续

表 24.53 - 续上页

Label	名称	Type	Description
Background color Optional	BACKGROUND_COLOR	[color] Default: QColor(0, 0, 0, 0)	Choose the background color for the tiles
Tile format	TILE_FORMAT	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- PNG • 1 --- JPG
Quality (JPG only) Optional	QUALITY	[number] Default: 75	Minimum 1, maximum 100.
Metatile size Optional	METATILESIZE	[number] Default: 4	Specify a custom metatile size when generating XYZ tiles. Larger values may speed up the rendering of tiles and provide better labelling (fewer gaps without labels) at the expense of using more memory. Minimum 1, maximum 20.
Tile width Optional	TILE_WIDTH	[number] Default: 256	Minimum 1, maximum 4096.
Tile height Optional	TILE_HEIGHT	[number] Default: 256	Minimum 1, maximum 4096.
Use inverted tile Y axis (TMS conventions) Optional	TMS_CONVENTION	[boolean] Default: False	
Output directory	OUTPUT_DIRECTORY	[folder] Default: [Save to temporary folder]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary Directory • Save to Directory... The file encoding can also be changed here.
Output html (Leaflet)	OUTPUT_HTML	[html] Default: [Save to temporary file]	Specification of the output HTML file. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File...

Outputs

Label	名称	Type	Description
Output directory	OUTPUT_DIRECTORY	[folder]	Output directory (for the tiles)
Output html (Leaflet)	OUTPUT_HTML	[html]	The output HTML (Leaflet) file

Python code

Algorithm ID: qgis:tilexyzdirectory

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Generate XYZ tiles (MBTiles)

Generates raster “XYZ” tiles using the current QGIS project as a single file in the “MBTiles” format.

Parameters

Label	名称	Type	Description
Extent (xmin, xmax, ymin, ymax)	EXTENT	[extent]	Specify the extent of the tiles. One of: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent... It will internally be extended to a multiple of the tile size.
Minimum zoom	ZOOM_MIN	[number] Default: 12	Minimum 0, maximum 25.
Maximum zoom	ZOOM_MAX	[number] Default: 12	Minimum 0, maximum 25.
DPI	DPI	[number] Default: 96	Minimum 48, maximum 600.
Background color Optional	BACKGROUND_COLOR	[color] Default: QColor(0, 0, 0, 0)	Choose the background color for the tiles
Tile format	TILE_FORMAT	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- PNG • 1 --- JPG
Quality (JPG only) Optional	QUALITY	[number] Default: 75	Minimum 1, maximum 100.
Metatile size Optional	METATILESIZE	[number] Default: 4	Specify a custom metatile size when generating XYZ tiles. Larger values may speed up the rendering of tiles and provide better labelling (fewer gaps without labels) at the expense of using more memory. Minimum 1, maximum 20.
Output file (for MBTiles)	OUTPUT_FILE	[file] Default: [Save to temporary file]	Specification of the output file. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output file (for MBTiles)	OUTPUT_FILE	[file]	The output file.

Python code

Algorithm ID: qgis:tilexyzmbtiles

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.13 Vector analysis

Basic statistics for fields

Generates basic statistics for a field of the attribute table of a vector layer.

Numeric, date, time and string fields are supported.

The statistics returned will depend on the field type.

Statistics are generated as an HTML file and are available in the *Processing ► Results viewer*.

Default menu: *Vector ► Analysis Tools*

Parameters

Label	名称	Type	Description
Input vector	INPUT_LAYER	[vector: any]	Vector layer to calculate the statistics on
Field to calculate statistics on	FIELD_NAME	[tablefield: any]	Any supported table field to calculate the statistics
Statistics	OUTPUT_HTML_FILE	[html]	HTML file for the calculated statistics

Outputs

Label	名称	Type	Description
Statistics	OUTPUT_HTML_FILE	[html]	HTML file with the calculated statistics
Count	COUNT	[number]	
Number of unique values	UNIQUE	[number]	
Number of empty (null) values	EMPTY	[number]	
Number of non-empty values	FILLED	[number]	
Minimum value	MIN	[same as input]	
Maximum value	MAX	[same as input]	
Minimum length	MIN_LENGTH	[number]	
Maximum length	MAX_LENGTH	[number]	

下页继续

表 24.55 - 续上页

Label	名称	Type	Description
Mean length	MEAN_LENGTH	[number]	
Coefficient of Variation	CV	[number]	
Sum	SUM	[number]	
Mean value	MEAN	[number]	
Standard deviation	STD_DEV	[number]	
Range	RANGE	[number]	
Median	MEDIAN	[number]	
Minority (rarest occurring value)	MINORITY	[same as input]	
Majority (most frequently occurring value)	MAJORITY	[same as input]	
First quartile	FIRSTQUARTILE	[number]	
Third quartile	THIRDQUARTILE	[number]	
Interquartile Range (IQR)	IQR	[number]	

Python code

Algorithm ID: qgis:basicstatisticsforfields

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Climb along line

Calculates the total climb and descent along line geometries. The input layer must have Z values present. If Z values are not available, the *Drape (set Z value from raster)* algorithm may be used to add Z values from a DEM layer.

The output layer is a copy of the input layer with additional fields that contain the total climb (`climb`), total descent (`descent`), the minimum elevation (`minelev`) and the maximum elevation (`maxelev`) for each line geometry. If the input layer contains fields with the same names as these added fields, they will be renamed (field names will be altered to "name_2", "name_3", etc, finding the first non-duplicate name).

Parameters

Label	名称	Type	Description
Line layer	INPUT	[vector: line]	Line layer to calculate the climb for. Must have Z values
Climb layer	OUTPUT	[vector: line]	The output (line) layer

Outputs

Label	名称	Type	Description
Climb layer	OUTPUT	[vector: line]	Line layer containing new attributes with the results from climb calculations.
Total climb	TOTALCLIMB	[number]	The sum of the climb for all the line geometries in the input layer
Total descent	TOTALDESCENT	[number]	The sum of the descent for all the line geometries in the input layer
Minimum elevation	MINELEVATION	[number]	The minimum elevation for the geometries in the layer
Maximum elevation	MAXELEVATION	[number]	The maximum elevation for the geometries in the layer

Python code

Algorithm ID: qgis:climbalongline

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Count points in polygon

Takes a point and a polygon layer and counts the number of points from the point layer in each of the polygons of the polygon layer.

A new polygon layer is generated, with the exact same content as the input polygon layer, but containing an additional field with the points count corresponding to each polygon.

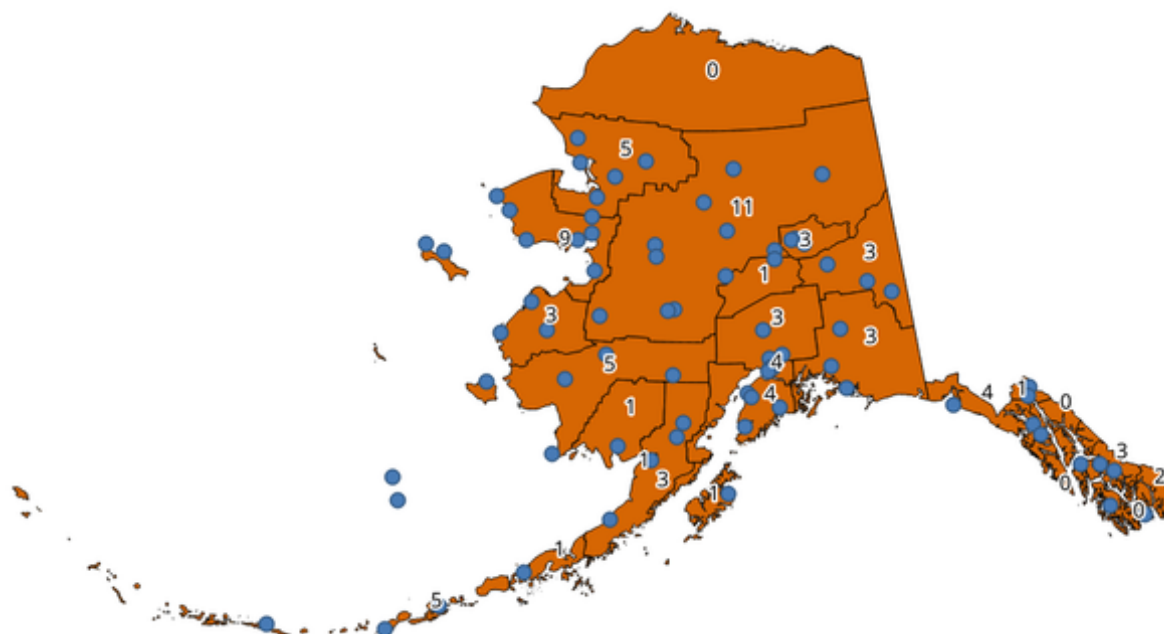


图 24.26: The labels in the polygons show the point count

An optional weight field can be used to assign weights to each point. Alternatively, a unique class field can be specified. If both options are used, the weight field will take precedence and the unique class field will be ignored.

Default menu: *Vector* ► *Analysis Tools*

Parameters

Label	名称	Type	Description
Polygons	POLYGONS	[vector: polygon]	Polygon layer whose features are associated with the count of points they contain
Points	POINTS	[vector: point]	Point layer with features to count
Weight field Optional	WEIGHT	[tablefield: any]	A field from the point layer. The count generated will be the sum of the weight field of the points contained by the polygon. If the weight field is not numeric, the count will be 0.
Class field Optional	CLASSFIELD	[tablefield: any]	Points are classified based on the selected attribute and if several points with the same attribute value are within the polygon, only one of them is counted. The final count of the points in a polygon is, therefore, the count of different classes that are found in it.
Count field name	FIELD	[string] Default: 'NUM-POINTS'	The name of the field to store the count of points
Count	OUTPUT	[vector: polygon]	Specification of the output layer

Outputs

Label	名称	Type	Description
Count	OUTPUT	[vector: polygon]	Resulting layer with the attribute table containing the new column with the points count

DBSCAN clustering

Clusters point features based on a 2D implementation of Density-based spatial clustering of applications with noise (DBSCAN) algorithm.

The algorithm requires two parameters, a minimum cluster size, and the maximum distance allowed between clustered points.

参见:

K-means clustering

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Layer to analyze
Minimum cluster size	MIN_SIZE	[number] Default: 5	Minimum number of features to generate a cluster
Maximum distance between clustered points	EPS	[number] Default: 1.0	Distance beyond which two features can not belong to the same cluster (eps)
Cluster field name	FIELD_NAME	[string] Default: 'CLUSTER_ID'	Name of the field where the associated cluster number shall be stored
Treat border points as noise (DBSCAN*) Optional	DBSCAN*	[boolean] Default: False	If checked, points on the border of a cluster are themselves treated as unclustered points, and only points in the interior of a cluster are tagged as clustered.
Clusters	OUTPUT	[vector: point]	Vector layer for the result of the clustering

Outputs

Label	名称	Type	Description
Clusters	OUTPUT	[vector: point]	Vector layer containing the original features with a field setting the cluster they belong to
Number of clusters	NUM_CLUSTERS	[number]	The number of clusters discovered

Python code

Algorithm ID: qgis:dbscanclustering

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Distance matrix

Calculates for point features distances to their nearest features in the same layer or in another layer.

Default menu: *Vector ► Analysis Tools*

参见:

Join attributes by nearest

Parameters

Label	名称	Type	Description
Input point layer	INPUT	[vector: point]	Point layer for which the distance matrix is calculated (from points)
Input unique ID field	INPUT_FIELD	[tablefield: any]	Field to use to uniquely identify features of the input layer. Used in the output attribute table.
Target point layer	TARGET	[vector: point]	Point layer containing the nearest point(s) to search (to points)
Target unique ID field	TARGET_FIELD	[tablefield: any]	Field to use to uniquely identify features of the target layer. Used in the output attribute table.
Output matrix type	MATRIX_TYPE	[enumeration] Default: 0	Different types of calculation are available: <ul style="list-style-type: none"> • 0 --- Linear ($N * k \times 3$) distance matrix: for each input point, reports the distance to each of the k nearest target points. The output matrix consists of up to k rows per input point, and each row has three columns: <i>InputID</i>, <i>TargetID</i> and <i>Distance</i>. • 1 --- Standard ($N \times T$) distance matrix • 2 --- Summary distance matrix (mean, std. dev., min, max): for each input point, reports statistics on the distances to its target points.
Use only the nearest (k) target points	NEAREST_POINTS	[number] Default: 0	You can choose to calculate the distance to all the points in the target layer (0) or limit to a number (k) of closest features.
Distance matrix	OUTPUT	[vector: point]	

Outputs

Label	名称	Type	Description
Distance matrix	OUTPUT	[vector: point]	Point (or MultiPoint for the "Linear (N * k x 3)" case) vector layer containing the distance calculation for each input feature. Its features and attribute table depend on the selected output matrix type.

Python code

Algorithm ID: qgis:distancematrix

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Distance to nearest hub (line to hub)

Creates lines that join each feature of an input vector to the nearest feature in a destination layer. Distances are calculated based on the *center* of each feature.

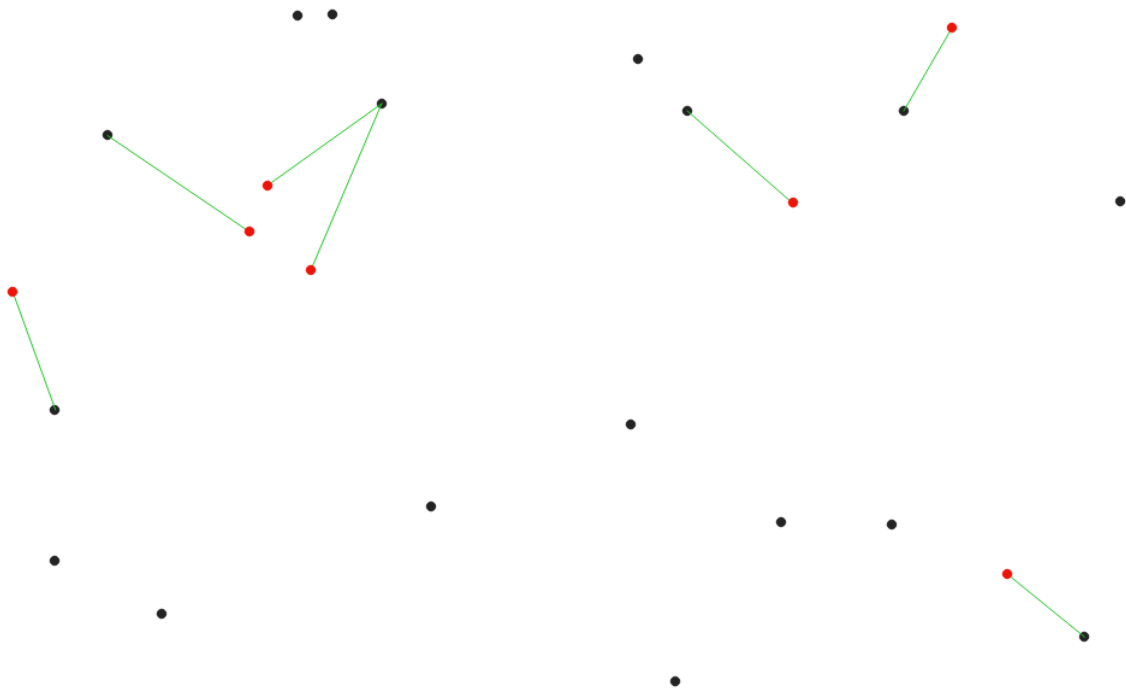


图 24.27: Display the nearest hub for the red input features

参见:

[Distance to nearest hub \(points\)](#), [Join attributes by nearest](#)

Parameters

Label	名称	Type	Description
Source points layer	INPUT	[vector: any]	Vector layer for which the nearest feature is searched
Destination hubs layer	HUBS	[vector: any]	Vector layer containing the features to search for
Hub layer name attribute	FIELD	[tablefield: any]	Field to use to uniquely identify features of the destination layer. Used in the output attribute table
Measurement unit	UNIT	[enumeration] Default: 0	Units in which to report the distance to the closest feature: <ul style="list-style-type: none"> • 0 --- Meters • 1 --- Feet • 2 --- Miles • 3 --- Kilometers • 4 --- Layer units
Hub distance	OUTPUT	[vector: line]	Line vector layer for the distance matrix output

Outputs

Label	名称	Type	Description
Hub distance	OUTPUT	[vector: line]	Line vector layer with the attributes of the input features, the identifier of their closest feature and the calculated distance.

Python code

Algorithm ID: qgis:distancetonearesthublinetohub

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Distance to nearest hub (points)

Creates a point layer representing the *center* of the input features with the addition of two fields containing the identifier of the nearest feature (based on its center point) and the distance between the points.

参见:

Distance to nearest hub (line to hub), Join attributes by nearest

Parameters

Label	名称	Type	Description
Source points layer	INPUT	[vector: any]	Vector layer for which the nearest feature is searched
Destination hubs layer	HUBS	[vector: any]	Vector layer containing the features to search for
Hub layer name attribute	FIELD	[tablefield: any]	Field to use to uniquely identify features of the destination layer. Used in the output attribute table
Measurement unit	UNIT	[enumeration] Default: 0	Units in which to report the distance to the closest feature: <ul style="list-style-type: none"> • 0 --- Meters • 1 --- Feet • 2 --- Miles • 3 --- Kilometers • 4 --- Layer units
Hub distance	OUTPUT	[vector: point]	Point vector layer for the distance matrix output.

Outputs

Label	名称	Type	Description
Hub distance	OUTPUT	[vector: point]	Point vector layer with the attributes of the input features, the identifier of their closest feature and the calculated distance.

Python code

Algorithm ID: `qgis:distancetonearesthubpoints`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Join by lines (hub lines)

Creates hub and spoke diagrams by connecting lines from points on the Spoke layer to matching points in the Hub layer.

Determination of which hub goes with each point is based on a match between the Hub ID field on the hub points and the Spoke ID field on the spoke points.

If input layers are not point layers, a point on the surface of the geometries will be taken as the connecting location.

Optionally, geodesic lines can be created, which represent the shortest path on the surface of an ellipsoid. When geodesic mode is used, it is possible to split the created lines at the antimeridian (± 180 degrees longitude), which can improve rendering of the lines. Additionally, the distance between vertices can be specified. A smaller distance results in a denser, more accurate line.

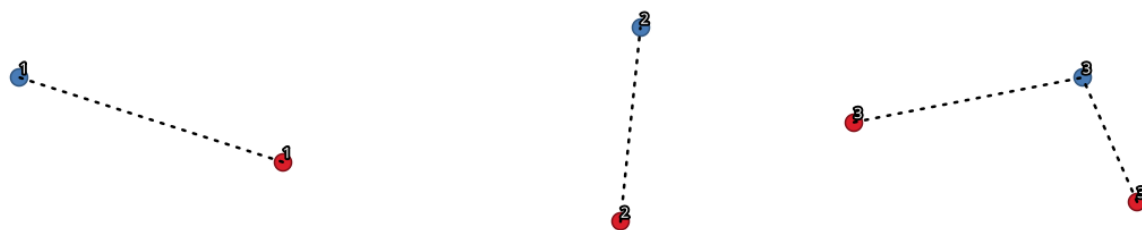


图 24.28: Join points based on a common field / attribute

Parameters

Label	名称	Type	Description
Hub layer	HUBS	[vector: any]	Input layer
Hub ID field	HUB_FIELD	[tablefield: any]	Field of the hub layer with ID to join
Hub layer fields to copy (leave empty to copy all fields) Optional	HUB_FIELDS	[tablefield: any] [list]	The field(s) of the hub layer to be copied. If no field(s) are chosen all fields are taken.
Spoke layer	SPOKES	[vector: any]	Additional spoke point layer
Spoke ID field	SPOKE_FIELD	[tablefield: any]	Field of the spoke layer with ID to join
Spoke layer fields to copy (leave empty to copy all fields) Optional	SPOKE_FIELDS	[tablefield: any] [list]	Field(s) of the spoke layer to be copied. If no fields are chosen all fields are taken.
Create geodesic lines	GEODESIC	[boolean] Default: False	Create geodesic lines (the shortest path on the surface of an ellipsoid)
Distance between vertices (geodesic lines only)	GEODESIC_DISTANCE	[number] Default: 1000.0 (kilometers)	Distance between consecutive vertices (in kilometers). A smaller distance results in a denser, more accurate line
Split lines at antimeridian (± 180 degrees longitude)	ANTIMERIDIAN_SPLIT	[boolean] Default: False	Split lines at ± 180 degrees longitude (to improve rendering of the lines)
Hub lines	OUTPUT	[vector: line]	The resulting line layer

Outputs

Label	名称	Type	Description
Hub lines	OUTPUT	[vector: line]	The resulting line layer

Python code

Algorithm ID: qgis:hublines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

K-means clustering

Calculates the 2D distance based k-means cluster number for each input feature.

K-means clustering aims to partition the features into k clusters in which each feature belongs to the cluster with the nearest mean. The mean point is represented by the barycenter of the clustered features.

If input geometries are lines or polygons, the clustering is based on the centroid of the feature.

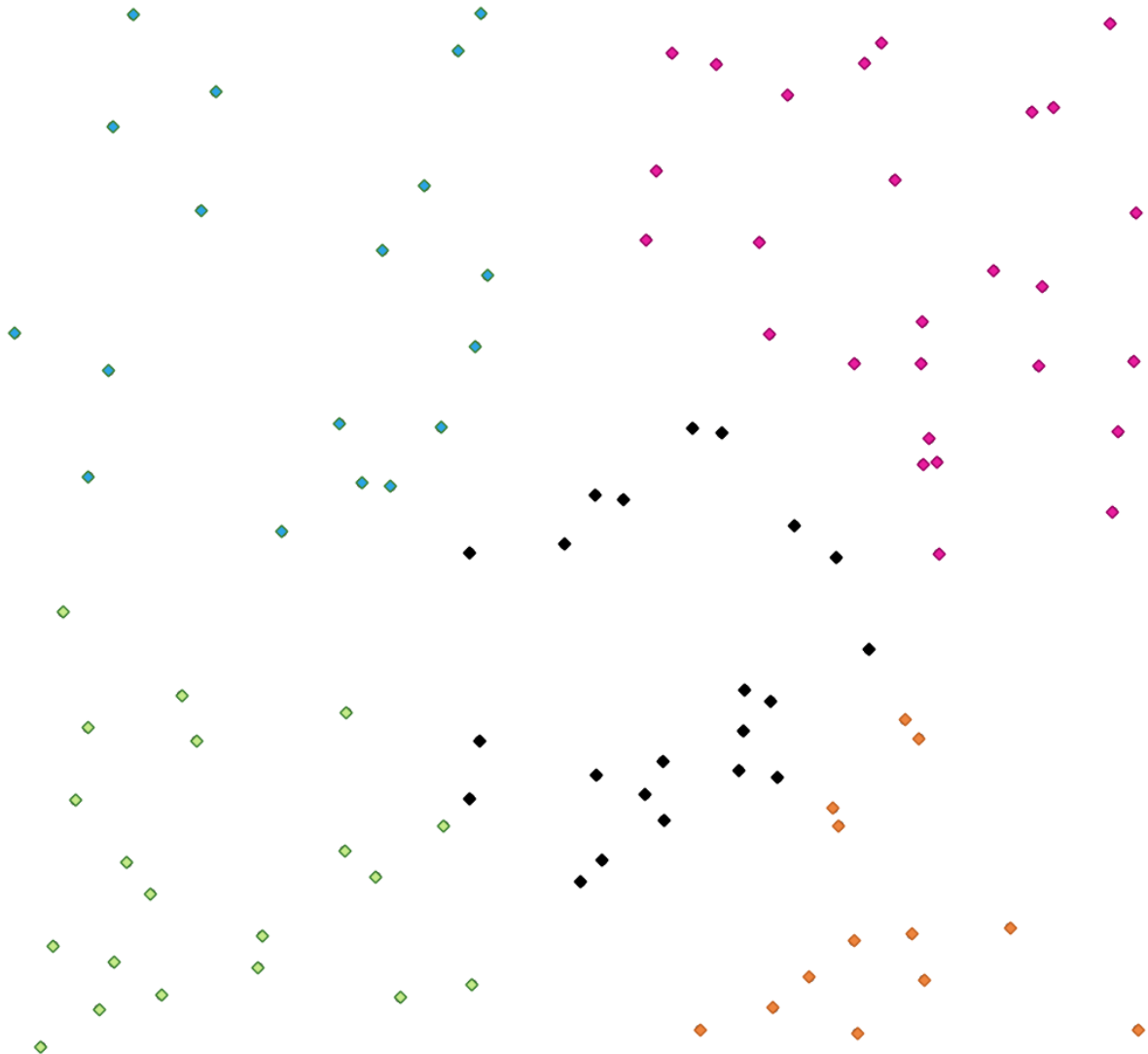


图 24.29: A five class point clusters

参见:

DBSCAN clustering

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Layer to analyze
Number of clusters	CLUSTERS	[number] Default: 5	Number of clusters to create with the features
Cluster field name	FIELD_NAME	[string] Default: 'CLUSTER_ID'	Name of the cluster number field
Clusters	OUTPUT	[vector: any]	Vector layer for generated the clusters

Outputs

Label	名称	Type	Description
Clusters	OUTPUT	[vector: any]	Vector layer containing the original features with a field specifying the cluster they belong to

Python code

Algorithm ID: qgis:kmeansclustering

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

List unique values

Lists unique values of an attribute table field and counts their number.

Default menu: *Vector ► Analysis Tools*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Layer to analyze
Target field(s)	FIELDS	[tablefield: any]	Field to analyze
Unique values	OUTPUT	[table]	Summary table layer with unique values
HTML report	OUTPUT_HTML_FILE	[html]	HTML report of unique values in the <i>Processing ► Results viewer</i>

Outputs

Label	名称	Type	Description
Unique values	OUTPUT	[table]	Summary table layer with unique values
HTML report	OUTPUT_HTML_FILE	[html]	HTML report of unique values. Can be opened from the <i>Processing ► Results viewer</i>
Total unique values	TOTAL_VALUES	[number]	The number of unique values in the input field
UNIQUE_VALUES	Unique values	[string]	A string with the comma separated list of unique values found in the input field

Python code

Algorithm ID: qgis:listuniquevalues

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Mean coordinate(s)

Computes a point layer with the center of mass of geometries in an input layer.

An attribute can be specified as containing weights to be applied to each feature when computing the center of mass.

If an attribute is selected in the parameter, features will be grouped according to values in this field. Instead of a single point with the center of mass of the whole layer, the output layer will contain a center of mass for the features in each category.

Default menu: *Vector ► Analysis Tools*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Weight field Optional	WEIGHT	[tablefield: numeric]	Field to use if you want to perform a weighted mean
Unique ID field	UID	[tablefield: numeric]	Unique field on which the calculation of the mean will be made
Mean coordinates	OUTPUT	[vector: point]	The (point vector) layer for the result

Outputs

Label	名称	Type	Description
Mean coordinates	OUTPUT	[vector: point]	Resulting point(s) layer

Python code

Algorithm ID: qgis:meancoordinates

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

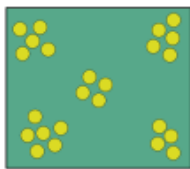
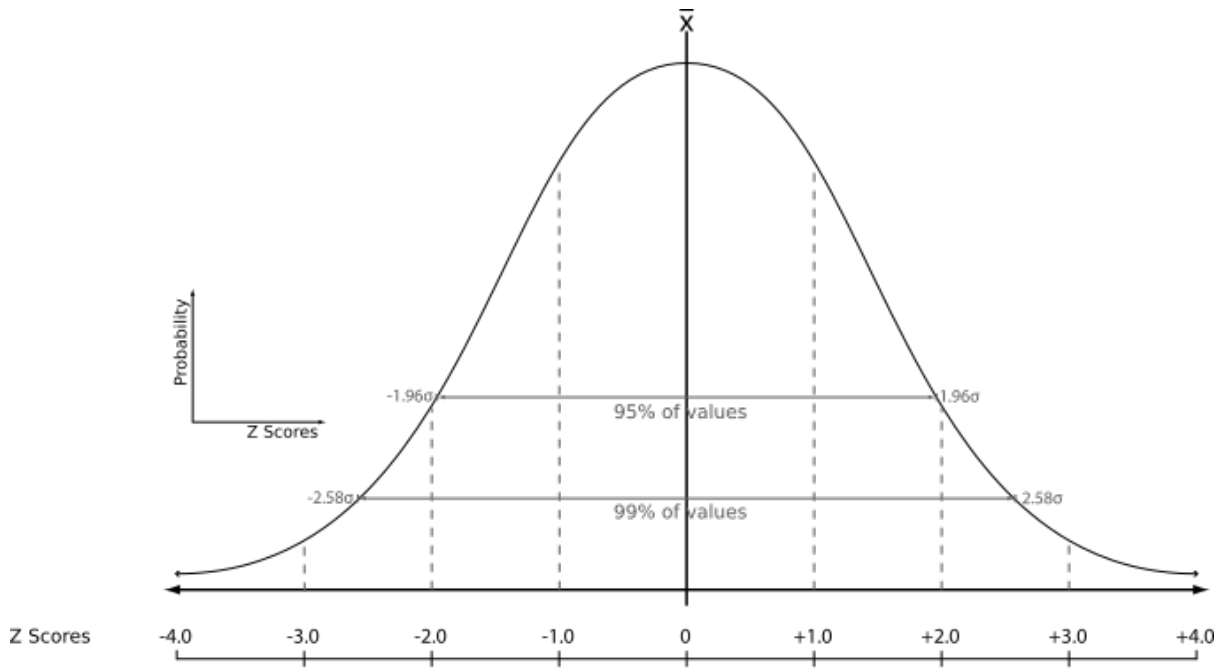
The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Nearest neighbour analysis

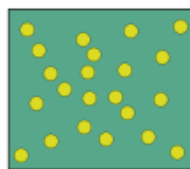
Performs nearest neighbor analysis for a point layer. The output tells you how your data are distributed (clustered, randomly or distributed).

Output is generated as an HTML file with the computed statistical values:

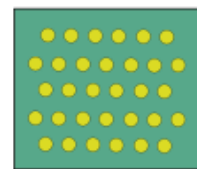
- Observed mean distance
- Expected mean distance
- Nearest neighbour index
- Number of points
- Z-Score: Comparing the Z-Score with the normal distribution tells you how your data are distributed. A low Z-Score means that the data are unlikely to be the result of a spatially random process, while a high Z-Score means that your data are likely to be a result of a spatially random process.



Clustered



Random



Dispersed

Default menu: *Vector ► Analysis Tools*

参见:

Join attributes by nearest

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Point vector layer to calculate the statistics on
Nearest neighbour	OUTPUT_HTML_FILE	[html]	HTML file for the computed statistics

Outputs

Label	名称	Type	Description
Nearest neighbour	OUTPUT_HTML_FILE	[html]	HTML file with the computed statistics
Observed mean distance	OBSERVED_MD	[number]	Observed mean distance
Expected mean distance	EXPECTED_MD	[number]	Expected mean distance
Nearest neighbour index	NN_INDEX	[number]	Nearest neighbour index
Number of points	POINT_COUNT	[number]	Number of points
Z-Score	Z_SCORE	[number]	Z-Score

Python code

Algorithm ID: qgis:nearestneighbouranalysis

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Overlap analysis

Calculates the area and percentage cover by which features from an input layer are overlapped by features from a selection of overlay layers.

New attributes are added to the output layer reporting the total area of overlap and percentage of the input feature overlapped by each of the selected overlay layers.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input layer.
Overlap layers	LAYERS	[vector: any] [list]	The overlay layers.
Output layer	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output layer	OUTPUT	[same as input]	The output layer with additional fields reporting the overlap (in map units and percentage) of the input feature overlapped by each of the selected layers.

Python code

Algorithm ID: qgis:calculatevectoroverlaps

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Statistics by categories

Calculates statistics of a field depending on a parent class. The parent class is a combination of values from other fields.

Parameters

Label	名称	Type	Description
Input vector layer	INPUT	[vector: any]	Input vector layer with unique classes and values
Field to calculate statistics on (if empty, only count is calculated) Optional	VALUES_FIELD_NAME	[tablefield: any]	If empty only the count will be calculated
Field(s) with categories	CATEGORIES_FIELD_NAME	[vector: any] [list]	The fields that (combined) define the categories
Statistics by category	OUTPUT	[table]	Table for the generated statistics

Outputs

Label	名称	Type	Description
Statistics by category	OUTPUT	[table]	Table containing the statistics

Depending on the type of the field being analyzed, the following statistics are returned for each grouped value:

Statistics	String	Numeric	Date
Count (COUNT)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

下页继续

表 24.61 - 续上页

Statistics	String	Numeric	Date
Unique values (UNIQUE)	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>
Empty (null) values (EMPTY)	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>
Non-empty values (FILLED)	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>
Minimal value (MIN)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Maximal value (MAX)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Range (RANGE)		<input checked="" type="checkbox"/>	
Sum (SUM)		<input checked="" type="checkbox"/>	
Mean value (MEAN)		<input checked="" type="checkbox"/>	
Median value (MEDIAN)		<input checked="" type="checkbox"/>	
Standard Deviation (STD_DEV)		<input checked="" type="checkbox"/>	
Coefficient of variation (CV)		<input checked="" type="checkbox"/>	
Minority (rarest occurring value - MINORITY)		<input checked="" type="checkbox"/>	
Majority (most frequently occurring value - MAJORITY)		<input checked="" type="checkbox"/>	
First Quartile (FIRSTQUARTILE)		<input checked="" type="checkbox"/>	
Third Quartile (THIRDQUARTILE)		<input checked="" type="checkbox"/>	
Inter Quartile Range (IQR)		<input checked="" type="checkbox"/>	
Minimum Length (MIN_LENGTH)	<input checked="" type="checkbox"/>		
Mean Length (MEAN_LENGTH)	<input checked="" type="checkbox"/>		
Maximum Length (MAX_LENGTH)	<input checked="" type="checkbox"/>		

Python code

Algorithm ID: qgis:statisticsbycategories

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Sum line lengths

Takes a polygon layer and a line layer and measures the total length of lines and the total number of them that cross each polygon.

The resulting layer has the same features as the input polygon layer, but with two additional attributes containing the length and count of the lines across each polygon.

Default menu: *Vector ► Analysis Tools*

Parameters

Label	名称	Type	Description
Lines	LINES	[vector: line]	Input vector line layer
Polygons	POLYGONS	[vector: polygon]	Polygon vector layer
Lines length field name	LEN_FIELD	[string] Default: 'LENGTH'	Name of the field for the lines length
Lines count field name	COUNT_FIELD	[string] Default: 'COUNT'	Name of the field for the lines count
Line length	OUTPUT	[vector: polygon]	The output polygon vector layer

Outputs

Label	名称	Type	Description
Line length	OUTPUT	[vector: polygon]	Polygon output layer with fields of lines length and line count

Python code

Algorithm ID: qgis:sumlinelengths

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.14 Vector creation

Array of offset (parallel) lines

Creates copies of line features in a layer, by creating multiple offset versions of each feature. Each new version is incrementally offset by a specified distance.

Positive distance will offset lines to the left, and negative distances will offset them to the right.

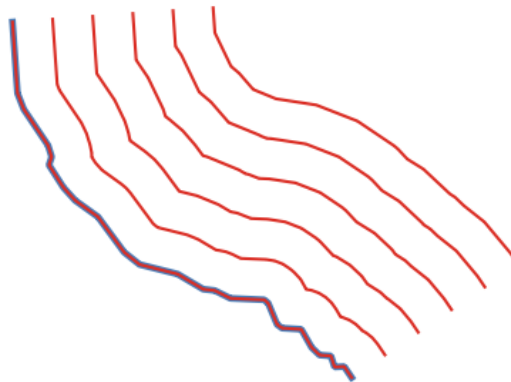




图 24.30: In blue the source layer, in red the offset set

 Allows *features in-place modification*

参见:

Offset lines, Array of translated features

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer to use for the offsets.
Number of features to create	COUNT	[number ] Default: 10	Number of offset copies to generate for each feature
Offset step distance	OFFSET	[number ] Default: 1.0	Distance between two consecutive offset copies
Segments	SEGMENTS	[number] Default: 8	Number of line segments to use to approximate a quarter circle when creating rounded offsets
Join style	JOIN_STYLE	[enumeration] Default: 0	Specify whether round, miter or beveled joins should be used when offsetting corners in a line. One of: <ul style="list-style-type: none"> • 0 --- Round • 1 --- Miter • 2 --- Bevel
Miter limit	MITER_LIMIT	[number] Default: 2.0	Only applicable for mitered join styles, and controls the maximum distance from the offset curve to use when creating a mitered join.
Offset lines	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output line layer with offset features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Offset lines	OUTPUT	[vector: line]	Output line layer with offset features. The original features are also copied.

Python code

Algorithm ID: qgis:arrayoffsetlines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Array of translated features

Creates copies of features in a layer by creating multiple translated versions of each. Each copy is incrementally displaced by a preset amount in the X, Y and/or Z axis.

M values present in the geometry can also be translated.

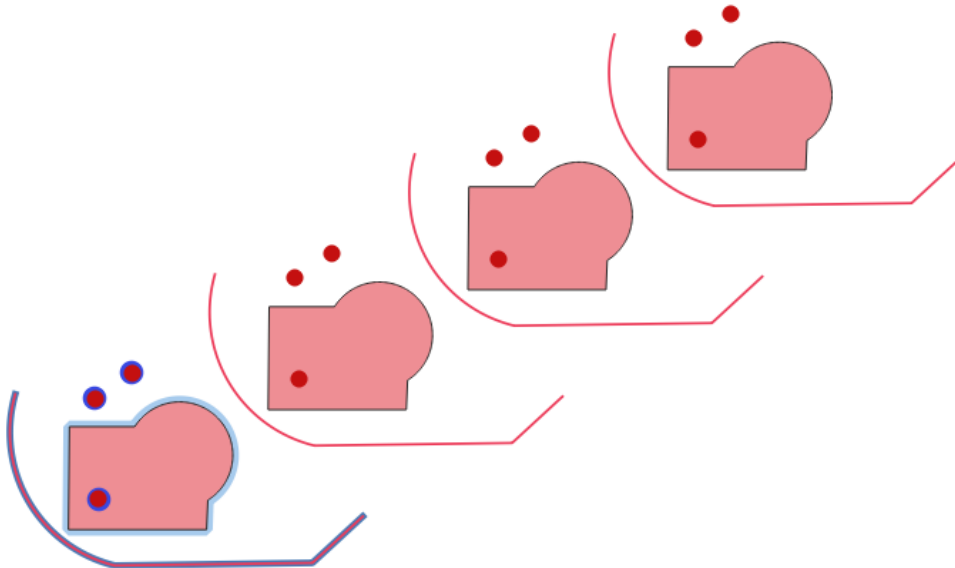




图 24.31: Input layers in blue tones, output layers with translated features in red tones

Allows *features in-place modification*

参见:




Translate, Array of offset (parallel) lines

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer to translate
Number of features to create	COUNT	[number ] Default: 10	Number of copies to generate for each feature
Step distance (x-axis)	DELTA_X	[number ] Default: 0.0	Displacement to apply on the X axis

下页继续

表 24.64 - 续上页

Label	名称	Type	Description
Step distance (y-axis)	DELTA_Y	[number ] Default: 0.0	Displacement to apply on the Y axis
Step distance (z-axis)	DELTA_Z	[number ] Default: 0.0	Displacement to apply on the Z axis
Step distance (m values)	DELTA_M	[number ] Default: 0.0	Displacement to apply on M
Translated	OUTPUT	[same as input] Default: [Create temporary layer]	Output vector layer with translated (moved) copies of the features. The original features are also copied. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Translated	OUTPUT	[same as input]	Output vector layer with translated (moved) copies of the features. The original features are also copied.

Python code

Algorithm ID: qgis:arraytranslatedfeatures

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Create grid

Creates a vector layer with a grid covering a given extent. Grid cells can have different shapes:

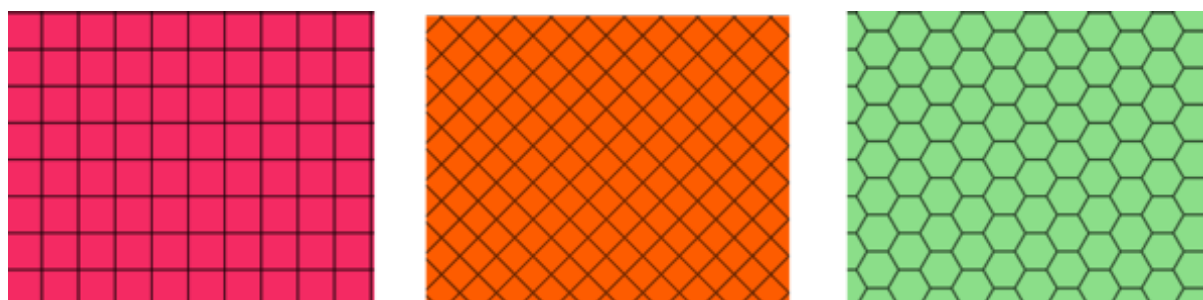


图 24.32: Different grid cell shapes

The size of each element in the grid is defined using a horizontal and vertical spacing.

The CRS of the output layer must be defined.

The grid extent and the spacing values must be expressed in the coordinates and units of this CRS.

Default menu: *Vector ► Research Tools*

Parameters

Label	名称	Type	Description
Grid type	TYPE	[enumeration] Default: 0	Shape of the grid. One of: <ul style="list-style-type: none"> • 0 --- Point • 1 --- Line • 2 --- Rectangle (polygon) • 3 --- Diamond (polygon) • 4 --- Hexagon (polygon)
Grid extent	EXTENT	[extent]	Extent of the grid
Horizontal spacing	HSPACING	[number] Default: 1.0	Size of a grid cell on the X-axis
Vertical spacing	VSPACING	[number] Default: 1.0	Size of a grid cell on the Y-axis
Horizontal overlay	HOVERLAY	[number] Default: 0.0	Overlay distance between two consecutive grid cells on the X-axis
Vertical overlay	VOVERLAY	[number] Default: 0.0	Overlay distance between two consecutive grid cells on the Y-axis
Grid CRS	CRS	[crs] Default: <i>Project CRS</i>	Coordinate reference system to apply to the grid
Grid	OUTPUT	[vector: any] Default: [Create temporary layer]	Resulting vector grid layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Grid	OUTPUT	[vector: any]	Resulting vector grid layer. The output geometry type (point, line or polygon) depends on the <i>Grid type</i> .

Python code

Algorithm ID: qgis:creategrid

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create points layer from table

Creates points layer from a table with columns that contain coordinates fields.

Besides X and Y coordinates you can also specify Z and M fields.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer or a table.
X field	XFIELD	[tablefield: any]	Field containing the X coordinate
Y field	YFIELD	[tablefield: any]	Field containing the Y coordinate
Z field Optional	ZFIELD	[tablefield: any]	Field containing the Z coordinate
M field Optional	MFIELD	[tablefield: any]	Field containing the M value
Target CRS	TARGET_CRIS	[crs] Default: EPSG:4326	Coordinate reference system to use for layer. The provided coordinates are assumed to be compliant.
Points from table	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the resulting point layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Points from table	OUTPUT	[vector: point]	The resulting point layer

Python code

Algorithm ID: qgis:createpointslayerfromtable

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Generate points (pixel centroids) along line

Generates a point vector layer from an input raster and line layer.

The points correspond to the pixel centroids that intersect the line layer.

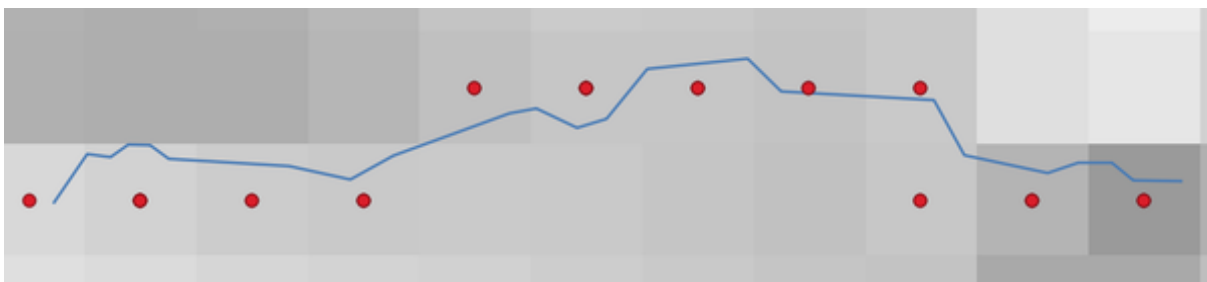


图 24.33: Points of the pixel centroids

Parameters

Label	名称	Type	Description
Raster layer	INPUT_RASTER	[raster]	Input raster layer
Vector layer	INPUT_VECTOR	[vector: line]	Input line vector layer
Points along line	OUTPUT	[vector: point] Default: [Create temporary layer]	Resulting point layer with pixel centroids. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Points along line	OUTPUT	[vector: point]	Resulting point layer with pixel centroids

Python code

Algorithm ID: qgis:generatepointspixelcentroidsalongline

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Generate points (pixel centroids) inside polygon

Generates a point vector layer from an input raster and polygon layer.

The points correspond to the pixel centroids that intersect the polygon layer.

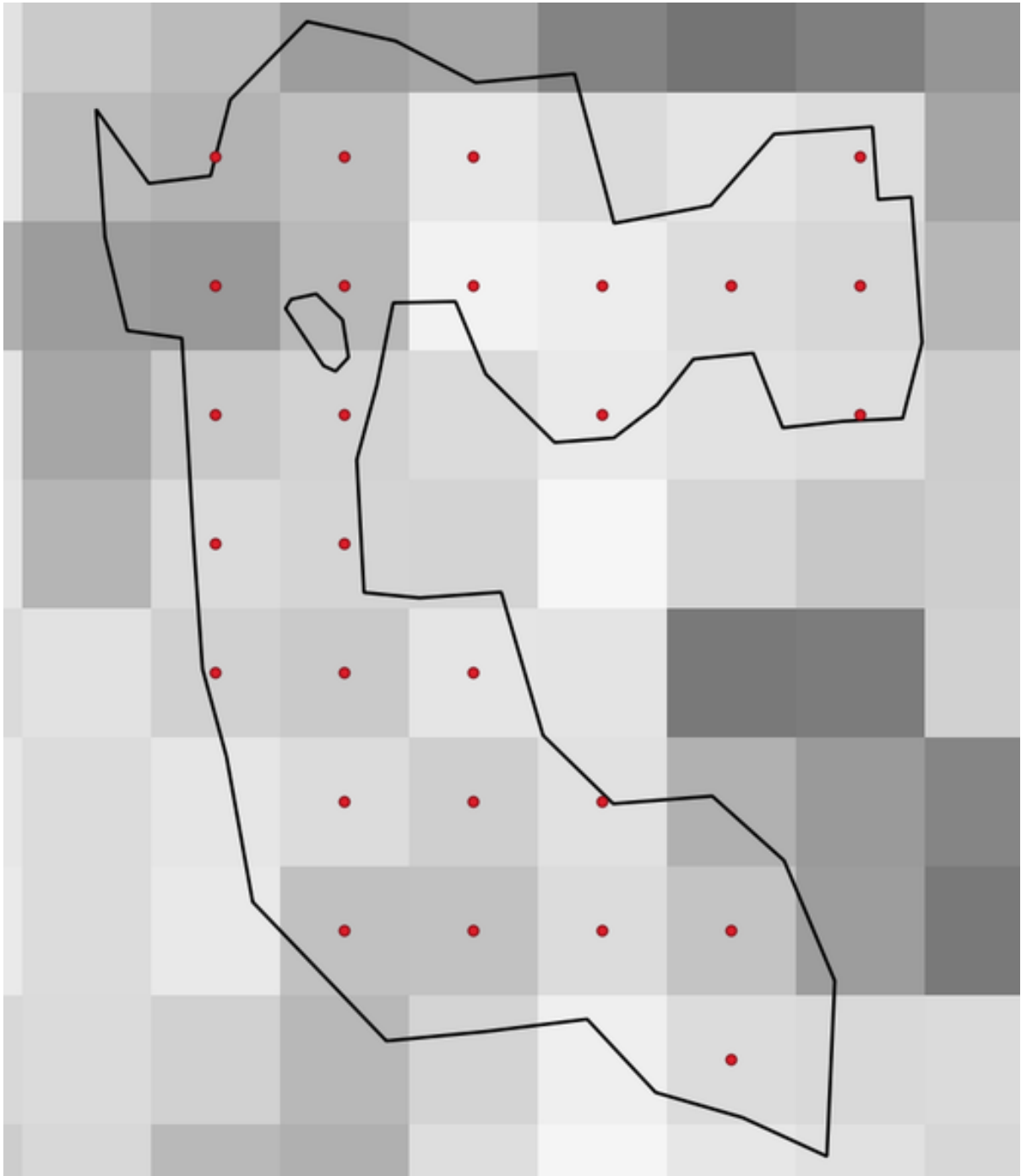


图 24.34: Points of the pixel centroids

Parameters

Label	名称	Type	Description
Raster layer	INPUT_RASTER	[raster]	Input raster layer
Vector layer	INPUT_VECTOR	[vector: polygon]	Input polygon vector layer
Points inside polygons	OUTPUT	[vector: point] Default: [Create temporary layer]	Resulting point layer of pixel centroids. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Points inside polygons	OUTPUT	[vector: point]	Resulting point layer of pixel centroids

Python code

Algorithm ID: `qgis:generatepointspixelcentroidsinsidepolygons`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Import geotagged photos

Creates a point layer corresponding to the geotagged locations from JPEG images from a source folder.

The point layer will contain a single PointZ feature per input file from which the geotags could be read. Any altitude information from the geotags will be used to set the point's Z value.

Besides longitude and latitude also altitude, direction and timestamp information, if present in the photo, will be added to the point as attributes.

Parameters

Label	名称	Type	Description
Input folder	FOLDER	[folder]	Path to the source folder containing the geotagged photos
Scan recursively	RECURSIVE	[boolean] Default: False	If checked, the folder and its subfolders will be scanned

下页继续

表 24.67 - 续上页

Label	名称	Type	Description
Photos	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the point vector layer for the geotagged photos. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Invalid photos table Optional	INVALID	[table] Default: [Skip output]	Specify the table of unreadable or non-geotagged photos. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Photos	OUTPUT	[vector: point]	Point vector layer with geotagged photos. The form of the layer is automatically filled with paths and photo previews settings.
Invalid photos table Optional	INVALID	[table]	Table of unreadable or non-geotagged photos can also be created.

Python code

Algorithm ID: qgis:importphotos

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Points to path

Converts a point layer to a line layer, by joining points in an order defined by a field in the input point layer (if the order field is a date/time field, the format must be specified).

Points can be grouped by a field to distinguish line features.

In addition to the line vector layer, a text file is output that describes the resulting line as a start point and a sequence of bearings / directions (relative to azimuth) and distances.

Parameters

Label	名称	Type	Description
Input point layer	INPUT	[vector: point]	Input point vector layer
Close path	CLOSE_PATH	[boolean] Default: False	If checked, the first and last points of the line will be connected and close the generated path
Order field	ORDER_FIELD	[tablefield: any]	Field containing the order to connect the points in the path
Group field Optional	GROUP_FIELD	[tablefield: any]	Point features of the same value in the field will be grouped in the same line. If not set, a single path is drawn with all the input points.
Date format (if order field is Date-Time) Optional	DATE_FORMAT	[string]	The format to use for the Order field parameter. Specify this only if the Order field is of type Date/Time.
Paths	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the line vector layer of the path. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Directory for text output	OUTPUT_TEXT_DIR	[folder] Default: [Skip output]	Specify the directory that will contain the description files of points and paths. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary Directory • Save to Directory... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Paths	OUTPUT	[vector: line]	Line vector layer of the path
Directory for text output	OUTPUT_TEXT_DIR	[folder]	Directory containing description files of points and paths

Python code

Algorithm ID: qgis:pointstopath

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random points along line

Creates a new point layer, with points placed on the lines of another layer.

For each line in the input layer, a given number of points is added to the resulting layer. The procedure for adding a point is to:

1. randomly select a line feature from the input layer
2. if the feature is multi-part, randomly select a part of it
3. randomly select a segment of that line
4. randomly select a position on that segment.

The procedure means that curved parts of the lines (with relatively short segments) will get more points than straight parts (with relatively long segments), as demonstrated in the illustration below, where the output of the *Random points along lines* algorithm can be compared with the output of the *Random points on lines* algorithm (that produces points with an, on average, even distribution along the lines).

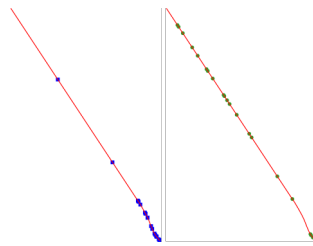


图 24.35: Example algorithm output. Left: *Random points along line*, right: *Random points on lines*

A minimum distance can be specified, to avoid points being too close to each other.

参见:

[Random points on lines](#)

Parameters

Label	名称	Type	Description
Input point layer	INPUT	[vector: line]	Input line vector layer
Number of points	POINTS_NUMBER	[number] Default: 1	Number of points to create
Minimum distance between points	MIN_DISTANCE	[number] Default: 0.0	The minimum distance between points
Random points	OUTPUT	[vector: point] Default: [Create temporary layer]	The output random points. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Random points	OUTPUT	[vector: point]	The output random points layer.

Python code

Algorithm ID: qgis:qgisrandompointsalongline

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random points in extent

Creates a new point layer with a given number of random points, all of them within a given extent.

A distance factor can be specified, to avoid points being too close to each other. If the minimum distance between points makes it impossible to create new points, either distance can be decreased or the maximum number of attempts may be increased.

Default menu: *Vector ► Research Tools*

Parameters

Label	名称	Type	Description
Input extent	EXTENT	[extent]	Map extent for the random points
Number of points	POINTS_NUMBER	[number] Default: 1	Number of point to create
Minimum distance between points	MIN_DISTANCE	[number] Default: 0.0	The minimum distance between points
Target CRS	TARGET_CRIS	[crs] Default: <i>Project CRS</i>	CRS of the random points layer
Maximum number of search attempts given the minimum distance	MAX_ATTEMPTS	[number] Default: 200	Maximum number of attempts to place the points
Random points	OUTPUT	[vector: point] Default: [Create temporary layer]	The output random points. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Random points	OUTPUT	[vector: point]	The output random points layer.

Python code

Algorithm ID: native:randompointsinextent

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random points in layer bounds

Creates a new point layer with a given number of random points, all of them within the extent of a given layer.

A minimum distance can be specified, to avoid points being too close to each other.

Default menu: *Vector ► Research Tools*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Input polygon layer defining the area
Number of points	POINTS_NUMBER	[number] Default: 1	Number of points to create
Minimum distance between points	MIN_DISTANCE	[number] Default: 0.0	The minimum distance between points
Random points	OUTPUT	[vector: point] Default: [Create temporary layer]	The output random points. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Random points	OUTPUT	[vector: point]	The output random points layer.

Python code

Algorithm ID: qgis:randompointsinlayerbounds

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random points in polygons

Creates a point layer with points placed inside the polygons of another layer.

For each feature (polygon / multi-polygon) geometry in the input layer, the given number of points is added to the result layer.

Per feature and global minimum distances can be specified in order to avoid points being too close in the output point layer. If a minimum distance is specified, it may not be possible to generate the specified number of points for each feature. The total number of generated points and missed points are available as output from the algorithm.

The illustration below shows the effect of per feature and global minimum distances and zero/non-zero minimum distances (generated with the same seed, so at least the first point generated will be the same).

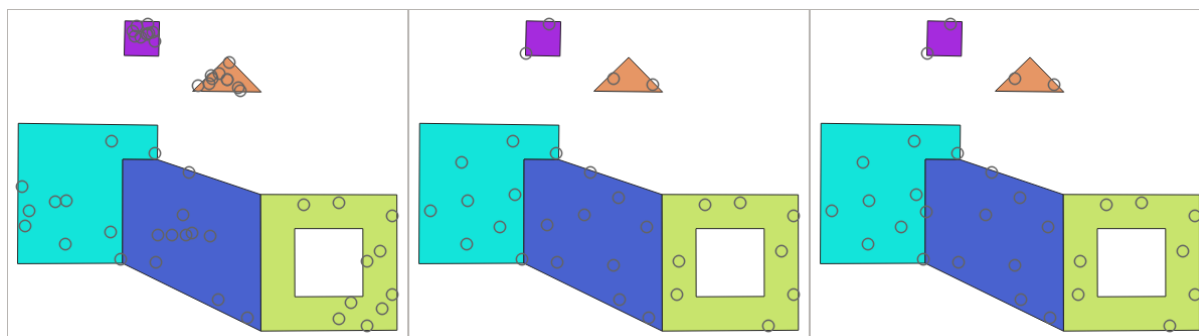


图 24.36: Ten points per polygon feature, *left*: min. distances = 0, *middle*: min.distances = 1, *right*: min. distance = 1, global min. distance = 0

The maximum number of tries per point can be specified. This is only relevant for non-zero minimum distance.

A seed for the random number generator can be provided, making it possible to get identical random number sequences for different runs of the algorithm.





The attributes of the polygon feature on which a point was generated can be included (*Include polygon attributes*).

If you want approximately the same point density for all the features, you can data-define the number of points using the area of the polygon feature geometry.

参见:

Random points inside polygons

Parameters

Label	名称	Type	Description
Input polygon layer	INPUT	[vector: line]	Input polygon vector layer
Number of points for each feature	POINTS_NUMBER	[number ] Default: 1	Number of points to create
Minimum distance between points Optional	MIN_DISTANCE	[number ] Default: 0.0	The minimum distance between points within one polygon feature
Global minimum distance between points Optional	MIN_DISTANCE_GLOBAL	[number ] Default: 0.0	The global minimum distance between points. Should be smaller than the <i>Minimum distance between points (per feature)</i> for that parameter to have an effect.
Maximum number of search attempts (for Min. dist. > 0) Optional	MAX_TRIES_PER_POINT	[number ] Default: 10	The maximum number of tries per point. Only relevant if the minimum distance between points is set (and greater than 0).
Random seed Optional	SEED	[number] Default: Not set	The seed to use for the random number generator.
Include polygon attributes	INCLUDE_POLYGON_ATTRIBUTES	[boolean] Default: True	If set, a point will get the attributes from the line on which it is placed.
Random points in polygons	OUTPUT	[vector: point] Default: [Create temporary layer]	The output random points. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Random points in polygons	OUTPUT	[vector: point]	The output random points layer.
Number of features with empty or no geometry	FEATURES_WITH_EMPTY_OR_NO_GEOMETRY	[number]	
Total number of points generated	OUTPUT_POINTS	[number]	
Number of missed points	POINTS_MISSED	[number]	The number of points that could not be generated due to the minimum distance constraint.
Number of features with missed points	POLYGONS_WITH_MISSED_POINTS	[number]	Not including features with empty or no geometry

Python code

Algorithm ID: qgis:randompointsinpolygons

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random points inside polygons

Creates a new point layer with a given number of random points inside each polygon of the input polygon layer.

Two sampling strategies are available:

- Points count: number of points for each feature
- Points density: density of points for each feature


A minimum distance can be specified, to avoid points being too close to each other.

Default menu: *Vector ► Research Tools*

参见:

Random points in polygons

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Input polygon vector layer
Sampling strategy	STRATEGY	[enumeration] Default: 0	Sampling strategy to use. One of: <ul style="list-style-type: none"> • 0 --- Points count: number of points for each feature • 1 --- Points density: density of points for each feature
Point count or density	VALUE	[number  Default: 1.0	The number or density of points, depending on the chosen <i>Sampling strategy</i> .
Minimum distance between points	MIN_DISTANCE	[number] Default: 0.0	The minimum distance between points
Random points	OUTPUT	[vector: point] Default: [Create temporary layer]	The output random points. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Random points	OUTPUT	[vector: point]	The output random points layer.

Python code

Algorithm ID: qgis:randompointsinsidepolygons

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random points on lines

Creates a point layer with points placed on the lines of another layer.

For each feature (line / multi-line) geometry in the input layer, the given number of points is added to the result layer.

Per feature and global minimum distances can be specified in order to avoid points being too close in the output point layer. If a minimum distance is specified, it may not be possible to generate the specified number of points for each feature. The total number of generated points and missed points are available as output from the algorithm.

The illustration below shows the effect of per feature and global minimum distances and zero/non-zero minimum distances (generated with the same seed, so at least the first point generated will be the same).

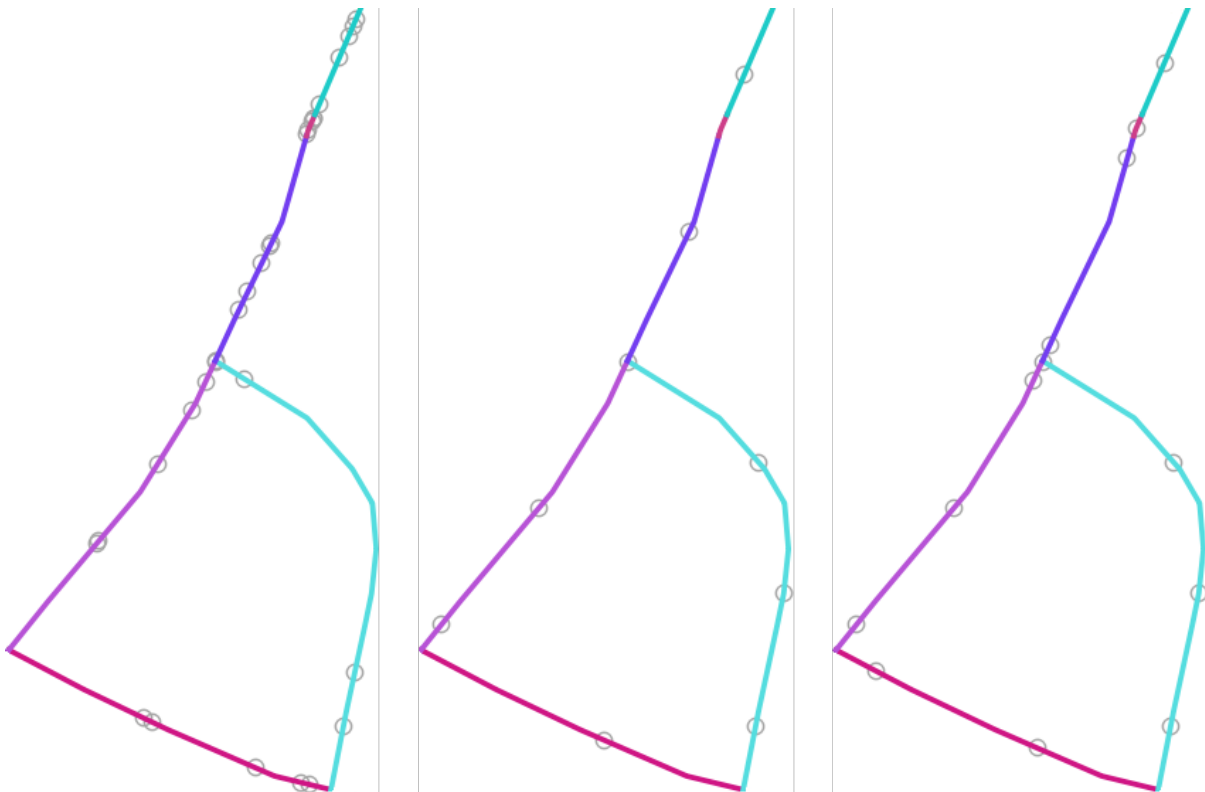


图 24.37: Five points per line feature, *left*: min. distances = 0, *middle*: min.distances != 0, *right*: min. distance != 0, global min. distance = 0

The maximum number of tries per point can be specified. This is only relevant for non-zero minimum distance.

A seed for the random number generator can be provided, making it possible to get identical random number sequences for different runs of the algorithm.





The attributes of the line feature on which a point was generated can be included (*Include line attributes*).

If you want approximately the same point density for all the line features, you can data-define the number of points using the length of the line feature geometry.

参见:

[Random points along line](#)

Parameters

Label	名称	Type	Description
Input line layer	INPUT	[vector: line]	Input line vector layer
Number of points for each feature	POINTS_NUMBER	[number ] Default: 1	Number of points to create
Minimum distance between points (per feature) Optional	MIN_DISTANCE	[number ] Default: 0.0	The minimum distance between points within one line feature
Global minimum distance between points Optional	MIN_DISTANCE_GLOBAL	[number ] Default: 0.0	The global minimum distance between points. Should be smaller than the <i>Minimum distance between points (per feature)</i> for that parameter to have an effect.
Maximum number of search attempts (for Min. dist. > 0) Optional	MAX_TRIES_PER_POINT	[number ] Default: 10	The maximum number of tries per point. Only relevant if the minimum distance between points is set (and greater than 0).
Random seed Optional	SEED	[number] Default: Not set	The seed to use for the random number generator.
Include line attributes	INCLUDE_LINE_ATTRIBUTES	[boolean] Default: True	If set, a point will get the attributes from the line on which it is placed.
Random points on lines	OUTPUT	[vector: point] Default: [Create temporary layer]	The output random points. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Random points on lines	OUTPUT	[vector: point]	The output random points layer.
Number of features with empty or no geometry	FEATURES_WITH_EMPTY_OR_NO_GEOMETRY	[number]	
Number of features with missed points	LINES_WITH_MISSED_POINTS	[number]	Not including features with empty or no geometry
Total number of points generated	OUTPUT_POINTS	[number]	
Number of missed points	POINTS_MISSED	[number]	The number of points that could not be generated due to the minimum distance constraint.

Python code

Algorithm ID: qgis:randompointsonlines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster pixels to points

Creates a vector layer of points corresponding to each pixel in a raster layer.

Converts a raster layer to a vector layer, by creating point features for each individual pixel's center in the raster layer. Any nodata pixels are skipped in the output.

Parameters

Label	名称	Type	Description
Raster layer	INPUT_RASTER	[raster]	Input raster layer
Band number	RASTER_BAND	[raster band]	Raster band to extract data from
Field name	FIELD_NAME	[string] Default: 'VALUE'	Name of the field to store the raster band value
Vector points	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the resulting point layer of pixels centroids. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Vector points	OUTPUT	[vector: point]	Resulting point layer with pixels centroids

Python code

Algorithm ID: qgis:pixelstopoints

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster pixels to polygons

Creates a vector layer of polygons corresponding to each pixel in a raster layer.

Converts a raster layer to a vector layer, by creating polygon features for each individual pixel's extent in the raster layer. Any nodata pixels are skipped in the output.

Parameters

Label	名称	Type	Description
Raster layer	INPUT_RASTER	[raster]	Input raster layer
Band number	RASTER_BAND	[raster band]	Raster band to extract data from
Field name	FIELD_NAME	[string] Default: 'VALUE'	Name of the field to store the raster band value
Vector polygons	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the resulting polygon layer of pixel extents. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Vector polygons	OUTPUT	[vector: polygon]	Resulting polygon layer of pixel extents

Python code

Algorithm ID: qgis:pixelstopolygons

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Regular points

Creates a new point layer with its points placed in a regular grid within a given extent.

The grid is specified either by the spacing between the points (same spacing for all dimensions) or by the number of points to generate. In the latter case, the spacing will be determined from the extent. In order to generate a full rectangular grid, at least the number of points specified by the user is generated for the latter case.

Random offsets to the point spacing can be applied, resulting in a non-regular point pattern.

Default menu: *Vector ► Research Tools*

Parameters

Label	名称	Type	Description
Input extent (xmin, xmax, ymin, ymax)	EXTENT	[extent]	Map extent for the random points
Point spacing/count	SPACING	[number] Default: 100	Spacing between the points, or the number of points, depending on whether <code>Use point spacing</code> is checked or not.
Initial inset from corner (LH side)	INSET	[number] Default: 0.0	Offsets the points relative to the upper left corner. The value is used for both the X and Y axis.
Apply random offset to point spacing	RANDOMIZE	[boolean] Default: False	If checked the points will have a random spacing
Use point spacing	IS_SPACING	[boolean] Default: True	If unchecked the point spacing is not taken into account
Output layer CRS	CRS	[crs] Default: <i>Project CRS</i>	CRS of the random points layer
Regular points	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output regular point layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Regular points	OUTPUT	[vector: point]	The output regular point layer.

Python code

Algorithm ID: qgis:regularpoints

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.15 Vector general

Assign projection

Assigns a new projection to a vector layer.

It creates a new layer with the exact same features and geometries as the input one, but assigned to a new CRS. The geometries are **not** reprojected, they are just assigned to a different CRS.

This algorithm can be used to repair layers which have been assigned an incorrect projection.

Attributes are not modified by this algorithm.

参见:

Define Shapefile projection, Find projection, Reproject layer

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Vector layer with wrong or missing CRS
Assigned CRS	CRS	[crs] Default: EPSG:4326 - WGS84	Select the new CRS to assign to the vector layer
Assigned CRS Optional	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output layer containing only the duplicates. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Assigned CRS	OUTPUT	[same as input]	Vector layer with assigned projection

Python code

Algorithm ID: native:assignprojection

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Convert layer to spatial bookmarks

Creates spatial bookmarks corresponding to the extent of features contained in a layer.

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: line, polygon]	The input vector layer
Bookmark destination	DESTINATION	[enumeration] Default: 0	Select the destination for the bookmarks. One of: <ul style="list-style-type: none"> • 0 --- Project bookmarks • 1 --- User bookmarks
Name field	NAME_EXPRESSION	[expression]	Field or expression that will give names to the generated bookmarks
Group field	GROUP_EXPRESSION	[expression]	Field or expression that will provide groups for the generated bookmarks

Outputs

Label	名称	Type	Description
Count of bookmarks added	COUNT	[number]	

Python code

Algorithm ID: native:layertobookmarks

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Convert spatial bookmarks to layer

Creates a new layer containing polygon features for stored spatial bookmarks. The export can be filtered to only bookmarks belonging to the current project, to all user bookmarks, or a combination of both.

Parameters

Label	名称	Type	Description
Bookmark source	SOURCE	[enumeration] [list] Default: [0,1]	Select the source(s) of the bookmarks. One or more of: <ul style="list-style-type: none"> • 0 --- Project bookmarks • 1 --- User bookmarks
Output CRS	CRS	[crs] Default: EPSG:4326 - WGS 84	The CRS of the output layer
Output	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output	OUTPUT	[vector: polygon]	The output (bookmarks) vector layer

Python code

Algorithm ID: native:bookmarkstolayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create attribute index

Creates an index against a field of the attribute table to speed up queries. The support for index creation depends on both the layer's data provider and the field type.

No outputs are created: the index is stored on the layer itself.

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Select the vector layer you want to create an attribute index for
Attribute to index	FIELD	[tablefield: any]	Field of the vector layer

Outputs

Label	名称	Type	Description
Indexed layer	OUTPUT	[same as input]	A copy of the input vector layer with an index for the specified field

Python code

Algorithm ID: native:createattributeindex

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Create spatial index

Creates an index to speed up access to the features in a layer based on their spatial location. Support for spatial index creation is dependent on the layer's data provider.

No new output layers are created.

Default menu: *Vector ► Data Management Tools*

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer

Outputs

Label	名称	Type	Description
Indexed layer	OUTPUT	[same as input]	A copy of the input vector layer with a spatial index

Python code

Algorithm ID: native:createspatialindex

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Define Shapefile projection

Sets the CRS (projection) of an existing Shapefile format dataset to the provided CRS. It is very useful when a Shapefile format dataset is missing the `prj` file and you know the correct projection.

Contrary to the *Assign projection* algorithm, it modifies the current layer and will not output a new layer.

注解: For Shapefile datasets, the `.prj` and `.qpj` files will be overwritten - or created if missing - to match the provided CRS.

Default menu: *Vector ► Data Management Tools*

参见:

Assign projection, Find projection, Reproject layer

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Vector layer with missing projection information
CRS	CRS	[crs]	Select the CRS to assign to the vector layer

Outputs

Label	名称	Type	Description
	INPUT	[same as input]	The input vector layer with the defined projection

Python code

Algorithm ID: qgis:definecurrentprojection

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Delete duplicate geometries

Finds and removes duplicated geometries.

Attributes are not checked, so in case two features have identical geometries but different attributes, only one of them will be added to the result layer.

参见:

Drop geometries, Remove null geometries, Delete duplicates by attribute

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The layer with duplicate geometries you want to clean
Cleaned	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Count of discarded duplicate records	DUPLICATE_COUNT	[number]	Count of discarded duplicate records
Cleaned	OUTPUT	[same as input]	The output layer without any duplicated geometries
Count of retained records	RETAINED_COUNT	[number]	Count of unique records

Python code

Algorithm ID: native:deleteduplicategeometries

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Delete duplicates by attribute

Deletes duplicate rows by only considering the specified field / fields. The first matching row will be retained, and duplicates will be discarded.

Optionally, these duplicate records can be saved to a separate output for analysis.

参见:

Delete duplicate geometries

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input layer
Fields to match duplicates by	FIELDS	[tablefield: any] [list]	Fields defining duplicates. Features with identical values for all these fields are considered duplicates.
Filtered (no duplicates)	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output layer containing the unique features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Filtered (duplicates) Optional	DUPLICATES	[same as input] Default: [Skip output]	Specify the output layer containing only the duplicates. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Filtered (duplicates) Optional	DUPLICATES	[same as input] Default: [Skip output]	Vector layer containing the removed features. Will not be produced if not specified (left as [Skip output]).
Count of discarded duplicate records	DUPLICATE_COUNT	[number]	Count of discarded duplicate records
Filtered (no duplicates)	OUTPUT	[same as input]	Vector layer containing the unique features.
Count of retained records	RETAINED_COUNT	[number]	Count of unique records

Python code

Algorithm ID: native:removeduplicatesbyattribute

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Detect dataset changes

Compares two vector layers, and determines which features are unchanged, added or deleted between the two. It is designed for comparing two different versions of the same dataset.

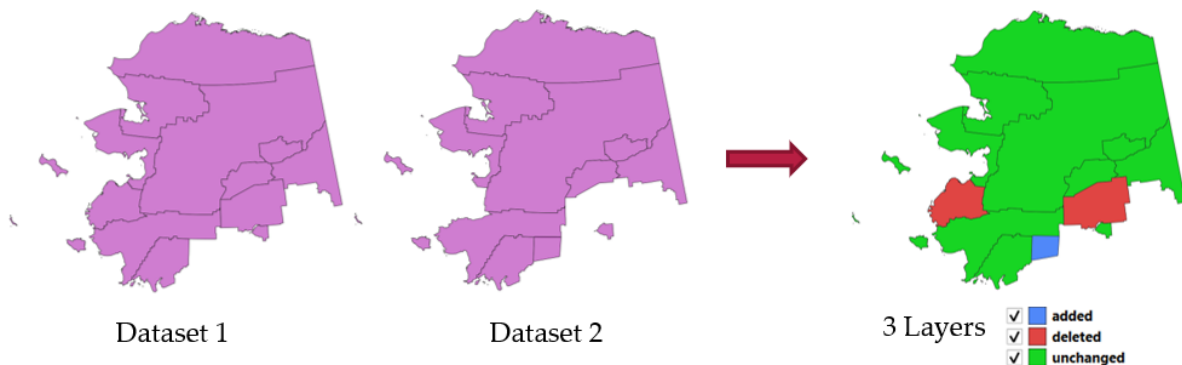


图 24.38: Detect dataset change example

Parameters

Label	名称	Type	Description
Original layer	ORIGINAL	[vector: any]	The vector layer considered as the original version
Revised layer	REVISED	[vector: any]	The revised or modified vector layer
Attributes to consider for match Optional	COMPARE_ATTRIBUTES	[tablefield: any] [list]	Attributes to consider for match. By default, all attributes are compared.
Geometry comparison behavior Optional	MATCH_TYPE	[enumeration] Default: 1	Defines the criteria for comparison. Options: <ul style="list-style-type: none"> • 0 --- Exact Match: includes the order and vertices count of geometries • 1 --- Tolerant Match (Topological Equality): geometries are considered equal
Unchanged features	UNCHANGED	[vector: same as Original layer]	Specify the output vector layer containing the unchanged features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Added features	ADDED	[vector: same as Original layer]	Specify the output vector layer containing the added features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Deleted features	DELETED	[vector: same as Original layer]	Specify the output vector layer containing the deleted features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Unchanged features	UNCHANGED	[vector: same as Original layer]	Vector layer containing the unchanged features.
Added features	ADDED	[vector: same as Original layer]	Vector layer containing the added features.
Deleted features	DELETED	[vector: same as Original layer]	Vector layer containing the deleted features.
Count of unchanged features	UNCHANGED_COUNT	[number]	Count of unchanged features.

下页继续

表 24.78 - 续上页

Label	名称	Type	Description
Count of features added in revised layer	ADDED_COUNT	[number]	Count of features added in revised layer.
Count of features deleted from original layer	DELETED_COUNT	[number]	Count of features deleted from original layer.

Python code

Algorithm ID: native:detectvectorchanges

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Drop geometries

Creates a simple *geometryless* copy of the input layer attribute table. It keeps the attribute table of the source layer.

If the file is saved in a local folder, you can choose between many file formats.

Allows *features in-place modification*

参见:

Delete duplicate geometries, Remove null geometries

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer
Dropped geometries	OUTPUT	[table]	Specify the output geometryless layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Dropped geometries	OUTPUT	[table]	The output geometryless layer. A copy of the original attribute table.

Python code

Algorithm ID: native:dropgeometries

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Execute SQL

Runs a simple or complex query with SQL syntax on the source layer.

Input datasources are identified with input1, input2... inputN and a simple query will look like SELECT * FROM input1.

Beside a simple query, you can add expressions or variables within the SQL query parameter itself. This is particularly useful if this algorithm is executed within a Processing model and you want to use a model input as a parameter of the query. An example of a query will then be SELECT * FROM [% @table %] where @table is the variable that identifies the model input.

The result of the query will be added as a new layer.

参见:

SpatiaLite execute SQL, PostgreSQL execute SQL

Parameters

Label	名称	Type	Description
Additional input datasources (called input1, ..., inputN in the query)	INPUT_DATASOURCES	Vector: any] [list]	List of layers to query. In the SQL editor you can refer these layers with their real name or also with input1, input2, inputN depending on how many layers have been chosen.
SQL query	INPUT_QUERY	[string]	Type the string of your SQL query, e.g. SELECT * FROM input1.
Unique identifier field Optional	INPUT_UID_FIELD	[string]	Specify the column with unique ID
Geometry field Optional	INPUT_GEOMETRY_FIELD	[string]	Specify the geometry field

下页继续

表 24.79 - 续上页

Label	名称	Type	Description
Geometry type Optional	INPUT_GEOMETRY_	[enumeration] Default: 0	Choose the geometry of the result. By default the algorithm will autodetect it. One of: <ul style="list-style-type: none"> • 0 --- Autodetect • 1 --- No geometry • 2 --- Point • 3 --- LineString • 4 --- Polygon • 5 --- MultiPoint • 6 --- MultiLineString • 7 --- MultiPolygon
CRS Optional	INPUT_GEOMETRY_	[crs]	The CRS to assign to the output layer
SQL Output	OUTPUT	[vector: any] Default: [Create temporary layer]	Specify the output layer created by the query. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
SQL Output	OUTPUT	[vector: any]	Vector layer created by the query

Python code

Algorithm ID: qgis:executesql

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extract selected features

Saves the selected features as a new layer.

注解: If the selected layer has no selected features, the newly created layer will be empty.

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Layer to save the selection from
Selected features	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the vector layer for the selected features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Selected features	OUTPUT	[same as input]	Vector layer with only the selected features, or no feature if none was selected.

Python code

Algorithm ID: native:savesselectedfeatures

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extract Shapefile encoding

Extracts the attribute encoding information embedded in a Shapefile. Both the encoding specified by an optional .cpg file and any encoding details present in the .dbf LDID header block are considered.

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	ESRI Shapefile (.SHP) Layer to extract the encoding information.

Outputs

Label	名称	Type	Description
Shapefile encoding	ENCODING	[string]	Encoding information specified in the input file
CPG encoding	CPG_ENCODING	[string]	Encoding information specified in any optional .CPG file
LDID encoding	LDID_ENCODING	[string]	Encoding information specified in .dbf LDID header block

Python code

Algorithm ID: native:shpencodinginfo

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Find projection

Creates a shortlist of candidate coordinate reference systems, for instance for a layer with an unknown projection.

The area that the layer is expected to cover must be specified via the target area parameter. The coordinate reference system for this target area must be known to QGIS.

The algorithm operates by testing the layer's extent in every known reference system and then listing any for which the bounds would be near the target area if the layer was in this projection.

参见:

Assign projection, Define Shapefile projection, Reproject layer

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Layer with unknown projection
Target area for layer (xmin, xmax, ymin, ymax)	TARGET_AREA	[extent]	The area that the layer covers. The options for specifying the extent are: <ul style="list-style-type: none"> • Use Canvas Extent • Select Extent on Canvas • Use Layer Extent It is also possible to provide the extent coordinates directly (xmin, xmax, ymin, ymax).

下页继续

表 24.81 - 续上页

Label	名称	Type	Description
CRS candidates	OUTPUT	[table] Default: [Create temporary layer]	Specify the table (geometryless layer) for the CRS suggestions (EPSG codes). One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
CRS candidates	OUTPUT	[table]	A table with all the CRS (EPSG codes) of the matching criteria.

Python code

Algorithm ID: qgis:findprojection

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Flatten relationship

Flattens a *relationship* for a vector layer, exporting a single layer containing one parent feature per related child feature. This master feature contains all the attributes for the related features. This allows to have the relation as a plain table that can be e.g. exported to CSV.

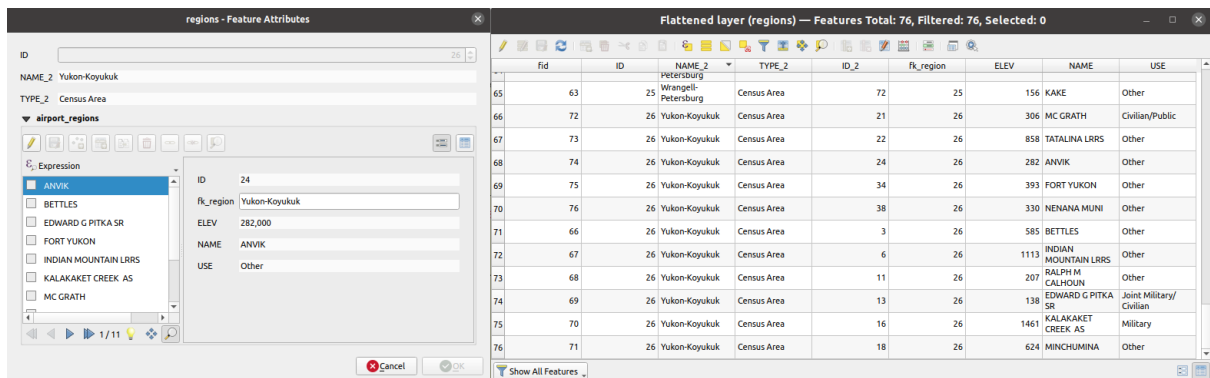


图 24.39: Form of a region with related children (left) - A duplicate region feature for each related child, with joined attributes (right)

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Layer with the relationship that should be de-normalized
Flattened Layer Optional	OUTPUT	[same as input] Default: [Save to temporary file]	Specify the output (flattened) layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... • Save To GeoPackage... • Save to Database Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Flattened layer	OUTPUT	[same as input]	A layer containing master features with all the attributes for the related features

Python code

Algorithm ID: native:flattenrelationships

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Join attributes by field value

Takes an input vector layer and creates a new vector layer that is an extended version of the input one, with additional attributes in its attribute table.

The additional attributes and their values are taken from a second vector layer. An attribute is selected in each of them to define the join criteria.

参见:

Join attributes by nearest, Join attributes by location

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer. The output layer will consist of the features of this layer with attributes from matching features in the second layer.
Table field	FIELD	[tablefield: any]	Field of the source layer to use for the join
Input layer 2	INPUT_2	[vector: any]	Layer with the attribute table to join

下页继续

表 24.83 - 续上页

Label	名称	Type	Description
Table field 2	FIELD_2	[tablefield: any]	Field of the second (join) layer to use for the join. The type of the field must be equal to (or compatible with) the input table field type.
Layer 2 fields to copy Optional	FIELDS_TO_COPY	[tablefield: any] [list]	Select the specific fields you want to add. By default all the fields are added.
Join type	METHOD	[enumeration] Default: 1	The type of the final joined layer. One of: <ul style="list-style-type: none"> • 0 --- Create separate feature for each matching feature (one-to-many) • 1 --- Take attributes of the first matching feature only (one-to-one)
Discard records which could not be joined	DISCARD_NONMATCHING	[boolean] Default: True	Check if you don't want to keep the features that could not be joined
Joined field prefix Optional	PREFIX	[string]	Add a prefix to joined fields in order to easily identify them and avoid field name collision
Joined layer	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer for the join. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Unjoinable features from first layer	NON_MATCHING	[same as input] Default: [Skip output]	Specify the output vector layer for unjoinable features from first layer. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Number of joined features from input table	JOINED_COUNT	[number]	
Unjoinable features from first layer Optional	NON_MATCHING	[same as input]	Vector layer with the non-matched features
Joined layer	OUTPUT	[same as input]	Output vector layer with added attributes from the join
Number of unjoinable features from input table Optional	UNJOINABLE_COUNT	[number]	

Python code

Algorithm ID: native:joinattributetable

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Join attributes by location

Takes an input vector layer and creates a new vector layer that is an extended version of the input one, with additional attributes in its attribute table.

The additional attributes and their values are taken from a second vector layer. A spatial criteria is applied to select the values from the second layer that are added to each feature from the first layer.

Default menu: *Vector ► Data Management Tools*

参见:

Join attributes by nearest, Join attributes by field value, Join attributes by location (summary)

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer. The output layer will consist of the features of this layer with attributes from matching features in the second layer.
Join layer	JOIN	[vector: any]	The attributes of this vector layer will be added to the source layer attribute table.
Geometric predicate	PREDICATE	[enumeration] [list] Default: [0]	Select the geometric criteria. One or more of: <ul style="list-style-type: none"> • 0 --- intersects • 1 --- contains • 2 --- equals • 3 --- touches • 4 --- overlaps • 5 --- within • 6 --- crosses
Fields to add (leave empty to use all fields) Optional	JOIN_FIELDS	[tablefield: any] [list]	Select the specific fields you want to add. By default all the fields are added.

下页继续

表 24.84 - 续上页

Label	名称	Type	Description
Join type	METHOD	[enumeration]	The type of the final joined layer. One of: <ul style="list-style-type: none"> • 0 --- Create separate feature for each matching feature (one-to-many) • 1 --- Take attributes of the first matching feature only (one-to-one) • 2 --- Take attributes of the feature with largest overlap only (one-to-one)
Discard records which could not be joined	DISCARD_NONMATCHING	[boolean] Default: False	Remove from the output the input layer records which could not be joined
Joined field prefix Optional	PREFIX	[string]	Add a prefix to joined fields in order to easily identify them and avoid field name collision
Joined layer	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer for the join. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Unjoinable features from first layer	NON_MATCHING	[same as input] Default: [Skip output]	Specify the output vector layer for unjoinable features from first layer. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Number of joined features from input table	JOINED_COUNT	[number]	
Unjoinable features from first layer Optional	NON_MATCHING	[same as input]	Vector layer of the non-matched features
Joined layer	OUTPUT	[same as input]	Output vector layer with added attributes from the join

Python code

Algorithm ID: native:joinattributesbylocation

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Join attributes by location (summary)

Takes an input vector layer and creates a new vector layer that is an extended version of the input one, with additional attributes in its attribute table.

The additional attributes and their values are taken from a second vector layer. A spatial criteria is applied to select the values from the second layer that are added to each feature from the first layer.

The algorithm calculates a statistical summary for the values from matching features in the second layer (e.g. maximum value, mean value, etc).

参见:

Join attributes by location

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer. The output layer will consist of the features of this layer with attributes from matching features in the second layer.
Join layer	JOIN	[vector: any]	The attributes of this vector layer will be added to the source layer attribute table.
Geometric predicate	PREDICATE	[enumeration] [list] Default: [0]	Select the geometric criteria. One or more of: <ul style="list-style-type: none"> • 0 --- intersects • 1 --- contains • 2 --- equals • 3 --- touches • 4 --- overlaps • 5 --- within • 6 --- crosses
Fields to summarize (leave empty to use all fields) Optional	JOIN_FIELDS	[tablefield: any] [list]	Select the specific fields you want to add and summarize. By default all the fields are added.

下页继续

表 24.85 - 续上页

Label	名称	Type	Description
Summaries to calculate (leave empty to use all fields) Optional	SUMMARIES	[enumeration] [list] Default: []	Choose which type of summary you want to add to each field and for each feature. One or more of: <ul style="list-style-type: none"> • 0 --- count • 1 --- unique • 2 --- min • 3 --- max • 4 --- range • 5 --- sum • 6 --- mean • 7 --- median • 8 --- stddev • 9 --- minority • 10 --- majority • 11 --- q1 • 12 --- q3 • 13 --- iqr • 14 --- empty • 15 --- filled • 16 --- min_length • 17 --- max_length • 18 --- mean_length
Discard records which could not be joined	DISCARD_NONMATCHING	Boolean Default: False	Remove from the output the input layer records which could not be joined
Joined layer	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer for the join. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Joined layer	OUTPUT	[same as input]	Output vector layer with summarized attributes from the join

Python code

Algorithm ID: qgis:joinbylocationsummary

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Join attributes by nearest

Takes an input vector layer and creates a new vector layer with additional fields in its attribute table. The additional attributes and their values are taken from a second vector layer. Features are joined by finding the closest features from each layer.

By default only the nearest feature is joined, but the join can also join to the k-nearest neighboring features.

If a maximum distance is specified, only features which are closer than this distance will be matched.

参见:

Nearest neighbour analysis, Join attributes by field value, Join attributes by location, Distance matrix

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input layer.
Input layer 2	INPUT_2	[vector: any]	The join layer.
Layer 2 fields to copy (leave empty to copy all fields)	FIELDS_TO_COPY	[fields]	Join layer fields to copy (if empty, all fields will be copied).
Discard records which could not be joined	DISCARD_NONMATCHING	[boolean] Default: False	Remove from the output the input layer records which could not be joined
Joined field prefix	PREFIX	[string]	Joined field prefix
Maximum nearest neighbors	NEIGHBORS	[number] Default: 1	Maximum number of nearest neighbors
Maximum distance	MAX_DISTANCE	[number]	Maximum search distance
Joined layer	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the vector layer containing the joined features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Unjoinable features from first layer	NON_MATCHING	[same as input] Default: [Skip output]	Specify the vector layer containing the features that could not be joined. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Joined layer	OUTPUT	[same as input]	The output joined layer.
Unjoinable features from first layer	NON_MATCHING	[same as input]	Layer containing the features from first layer that could not be joined to any features in the join layer.
Number of joined features from input table	JOINED_COUNT	[number]	Number of features from the input table that have been joined.
Number of unjoinable features from input table	UNJOINABLE_COUNT	[number]	Number of features from the input table that could not be joined.

Python code

Algorithm ID: native:joinbynearest

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Merge vector layers

Combines multiple vector layers of the **same geometry** type into a single one.

The attribute table of the resulting layer will contain the fields from all input layers. If fields with the same name but different types are found then the exported field will be automatically converted into a string type field. New fields storing the original layer name and source are also added.

If any input layers contain Z or M values, then the output layer will also contain these values. Similarly, if any of the input layers are multi-part, the output layer will also be a multi-part layer.

Optionally, the destination coordinate reference system (CRS) for the merged layer can be set. If it is not set, the CRS will be taken from the first input layer. All layers will be reprojected to match this CRS.



Default menu: *Vector ► Data Management Tools*

参见:

Split vector layer

Parameters

Label	名称	Type	Description
Input Layers	LAYERS	[vector: any] [list]	The layers that are to be merged into a single layer. Layers should be of the same geometry type.
Destination CRS Optional	CRS	[crs]	Choose the CRS for the output layer. If not specified, the CRS of the first input layer is used.
Merged	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Merged	OUTPUT	[same as input]	Output vector layer containing all the features and attributes from the input layers.

Python code

Algorithm ID: native:mergevectorlayers

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Order by expression

Sorts a vector layer according to an expression: changes the feature index according to an expression.

Be careful, it might not work as expected with some providers, the order might not be kept every time.

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer to sort
Expression	EXPRESSION	[expression]	Expression to use for the sorting
Sort ascending	ASCENDING	[boolean] Default: True	If checked the vector layer will be sorted from small to large values.
Sort nulls first	NULLS_FIRST	[boolean] Default: False	If checked, Null values are placed first
Ordered	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Ordered	OUTPUT	[same as input]	Output (sorted) vector layer

Python code

Algorithm ID: native:orderbyexpression

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Repair Shapefile

Repairs a broken ESRI Shapefile dataset by (re)creating the SHX file.

Parameters

Label	名称	Type	Description
Input Shapefile	INPUT	[file]	Full path to the ESRI Shapefile dataset with a missing or broken SHX file

Outputs

Label	名称	Type	Description
Repaired layer	OUTPUT	[vector: any]	The input vector layer with the SHX file repaired

Python code

Algorithm ID: native:repairshapefile

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Reproject layer

Reprojects a vector layer in a different CRS. The reprojected layer will have the same features and attributes of the input layer.

Allows *features in-place modification*

参见:

Assign projection, Define Shapefile projection, Find projection

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer to reproject
Target CRS	TARGET_CRS	[crs] Default: EPSG:4326 - WGS 84	Destination coordinate reference system
Coordinate Operation Optional	OPERATION	[string]	Specific operation to use for a particular re-projection task, instead of always forcing use of the current project's transformation settings. Useful when reprojecting a particular layer and control over the exact transformation pipeline is required. Requires proj version >= 6. Read more at <i>Datum Transformations</i> .
Reprojected	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Reprojected	OUTPUT	[same as input]	Output (reprojected) vector layer

Python code

Algorithm ID: native:reprojectlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Save vector features to file

Saves vector features to a specified file dataset.

For dataset formats supporting layers, an optional layer name parameter can be used to specify a custom string. Optional GDAL-defined dataset and layer options can be specified. For more information on this, read the online [GDAL documentation](#) on the format.

Parameters

Basic parameters

Label	名称	Type	Description
Vector features	INPUT	[vector: any]	Input vector layer.
Saved features	OUTPUT	[same as input] Default: [Save to temporary file]	Specify the file to save the features to. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Layer name Optional	LAYER_NAME	[string]	Name to use for the output layer
GDAL dataset options Optional	DATASOURCE_OPTIONS	[string]	GDAL dataset creation options of the output format. Separate individual options with semicolons.
GDAL layer options Optional	LAYER_OPTIONS	[string]	GDAL layer creation options of the output format. Separate individual options with semicolons.

Outputs

Label	名称	Type	Description
Saved features	OUTPUT	[same as input]	Vector layer with the saved features.
File name and path	FILE_PATH	[string]	Output file name and path.
Layer name	LAYER_NAME	[string]	Name of the layer, if any.

Python code

Algorithm ID: native:savefeatures

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Set layer encoding

Sets the encoding used for reading a layer's attributes. No permanent changes are made to the layer, rather it affects only how the layer is read during the current session.

注解: Changing the encoding is only supported for some vector layer data sources.

Parameters

Label	名称	Type	Description
Saved features	INPUT	[vector: any]	Vector layer to set the encoding.
Encoding	ENCODING	[string]	Text encoding to assign to the layer in the current QGIS session.

Outputs

Label	名称	Type	Description
Output layer	OUTPUT	[same as input]	Input vector layer with the set encoding.

Python code

Algorithm ID: native:setlayerencoding

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Split features by character

Features are split into multiple output features by splitting a field's value at a specified character. For instance, if a layer contains features with multiple comma separated values contained in a single field, this algorithm can be used to split these values up across multiple output features. Geometries and other attributes remain unchanged in the output. Optionally, the separator string can be a regular expression for added flexibility.

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer
Split using values in the field	FIELD	[tablefield: any]	Field to use for splitting
Split value using character	CHAR	[string]	Character to use for splitting
Use regular expression separator	REGEX	[boolean] Default: False	
Split	OUTPUT	[same as input] Default: Create temporary layer	Specify output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Split	OUTPUT	[same as input]	The output vector layer.

Python code

Algorithm ID: native:splitfeaturesbycharacter

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Split vector layer

Creates a set of vectors in an output folder based on an input layer and an attribute. The output folder will contain as many layers as the unique values found in the desired field.

The number of files generated is equal to the number of different values found for the specified attribute.

It is the opposite operation of *merging*.

Default menu: *Vector ► Data Management Tools*

参见:

[Merge vector layers](#)

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer
Unique ID field	FIELD	[tablefield: any]	Field to use for splitting
Output directory	OUTPUT	[folder] Default: [Save to temporary folder]	Specify the directory for the output layers. One of: <ul style="list-style-type: none"> • Save to a Temporary Directory • Save to Directory... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output directory	OUTPUT	[folder]	The directory for the output layers
Output layers	OUTPUT_LAYERS	[same as input] [list]	The output vector layers resulting from the split.

Python code

Algorithm ID: native:splitvectorlayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Truncate table

Truncates a layer, by deleting all features from within the layer.

警告: This algorithm modifies the layer in place, and deleted features cannot be restored!

Parameters

Label	名称	Type	Description
Input Layer	INPUT	[vector: any]	Input vector layer

Outputs

Label	名称	Type	Description
Truncated layer	OUTPUT	[folder]	The truncated (empty) layer

Python code

Algorithm ID: native:truncatetable

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.16 Vector geometry

Add geometry attributes

Computes geometric properties of the features in a vector layer and includes them in the output layer.

It generates a new vector layer with the same content as the input one, but with additional attributes, containing geometric measurements based on a selected CRS.

The attributes added to the table depend on the geometry type and dimension of the input layer:

- for **point** layers: X (*xcoord*), Y (*ycoord*), Z (*zcoord*) coordinates and/or M value (*mvalue*)
- for **line** layers: *length* and, for the LineString and CompoundCurve geometry types, the feature *sinuosity* and *straight distance* (*straightdis*)
- for **polygon** layers: *perimeter* and *area*

Default menu: *Vector ► Geometry Tools*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Calculate using	CALC_METHOD	[enumeration] Default: 0	Calculation parameters to use for the geometric properties. One of: <ul style="list-style-type: none"> • 0 --- Layer CRS • 1 --- Project CRS • 2 --- Ellipsoidal
Added geom info	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output (input copy with geometry) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Added geom info	OUTPUT	[same as input]	Copy of the input vector layer with the addition of the geometry fields

Python code

Algorithm ID: qgis:exportaddgeometrycolumns

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Affine transform

Applies an affine transformation to the layer geometries. Affine transformations can include translation, scaling and rotation. The operations are performed in the following order: scale, rotation, and translation.

Z and M values (if present) can be translated and scaled.

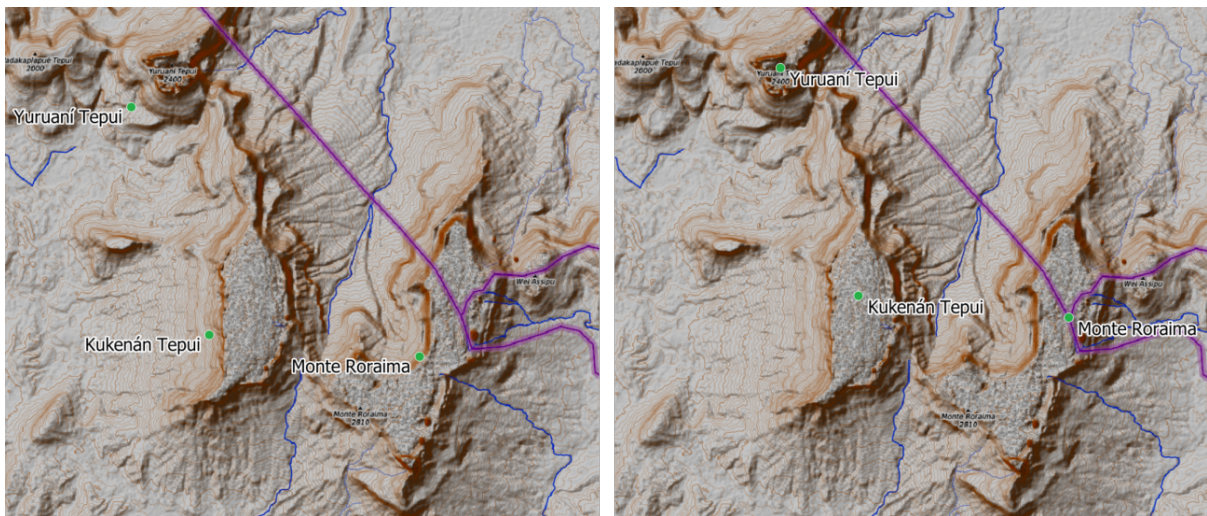











图 24.40: Vector point layer (green dots) before (left), and after (right) an affine transformation (translation).

参见:

Translate

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Translation (x-axis)	DELTA_X	[number ] Default: 0	Displacement to apply on the X axis.
Translation (y-axis)	DELTA_Y	[number ] Default: 0	Displacement to apply on the Y axis.
Translation (z-axis)	DELTA_Z	[number ] Default: 0	Displacement to apply on the Z axis.
Translation (m-values)	DELTA_M	[number ] Default: 0	Offset to apply on m values.
Scale factor (x-axis)	SCALE_X	[number ] Default: 1	Scaling value (expansion or contraction) to apply on the X axis.
Scale factor (y-axis)	SCALE_Y	[number ] Default: 1	Scaling value (expansion or contraction) to apply on the Y axis.
Scale factor (z-axis)	SCALE_Z	[number ] Default: 1	Scaling value (expansion or contraction) to apply on the Z axis.
Scale factor (m-values)	SCALE_M	[number ] Default: 1	Scaling value (expansion or contraction) to apply on m values.
Rotation around z-axis (degrees counter-clockwise)	ROTATION_Z	[number ] Default: 0	Angle of the rotation in degrees.
Transformed	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Transformed	OUTPUT	[same as input]	Output (transformed) vector layer.

Python code

Algorithm ID: native:affinetransform

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Aggregate

Takes a vector or table layer and creates a new layer by aggregating features based on a `group by` expression.

Features for which `group by` expression returns the same value are grouped together.

It is possible to group all source features together using constant value in `group by` parameter, example: `NULL`.

It is also possible to group features by multiple fields using Array function, example: `Array("Field1", "Field2")`.

Geometries (if present) are combined into one multipart geometry for each group. Output attributes are computed depending on each given aggregate definition.

This algorithm allows to use the default *aggregates functions* of the QGIS Expression engine.

参见:






Collect geometries, 融合 (*dissolve*)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Group by expression	GROUP_BY	[tablefield: any] Default: 'NULL'	Choose the grouping field. If <i>NULL</i> all features will be grouped.

下页继续

表 24.93 - 续上页

Label	名称	Type	Description
Aggregates	AGGREGATES	[list]	<p>List of output layer field definitions. Example of a field definition: <code>{'aggregate': 'sum', 'delimiter': ',', 'input': '\$area', 'length': 10, 'name': 'totarea', 'precision': 0, 'type': 6}</code></p> <p>By default, the list contains all the fields of the input layer. In the GUI, you can edit these fields and their definitions, and you can also:</p> <ul style="list-style-type: none"> • Click the  button to add a new field. • Click  to delete the selected field. • Use  and  to change order of the fields. • Click  to reset to the default (the fields of the input layer). <p>For each of the fields you'd like to retrieve information from, you need to define the following:</p> <p>Input expression [expression] (input) Field or expression from the input layer.</p> <p>Aggregate function [enumeration] (aggregate) <i>Function</i> to use on the input expression to return the aggregated value. Default: <i>concatenate</i> (for string data type), <i>sum</i> (for numeric data type)</p> <p>Delimiter [string] (delimiter) Text string to separate aggregated values, for example in case of concatenation. Default: ,</p> <p>Output field name [string] (name) Name of the aggregated field in the output layer. By default input field name is kept.</p> <p>Type [enumeration] (type) Data type of the output field. One of:</p> <ul style="list-style-type: none"> • 1 --- Boolean • 2 --- Integer • 4 --- Integer64 • 6 --- Double • 10 --- String • 14 --- Date • 16 --- DateTime <p>Length [number] (length) Length of the output field.</p> <p>Precision [number] (precision) Precision of the output field.</p>
Load fields from layer	GUI only	[vector: any]	You can load fields from another layer and use them for the aggregation

下页继续

表 24.93 - 续上页

Label	名称	Type	Description
Aggregated	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output (aggregate) layer One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Aggregated	OUTPUT	[same as input]	Multigeometry vector layer with the aggregated values

Python code

Algorithm ID: native:aggregate

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Boundary

Returns the closure of the combinatorial boundary of the input geometries (i.e. the topological boundary of the geometry).

Only for polygon and line layers.

For **polygon geometries** , the boundary consists of all the lines making up the rings of the polygon.

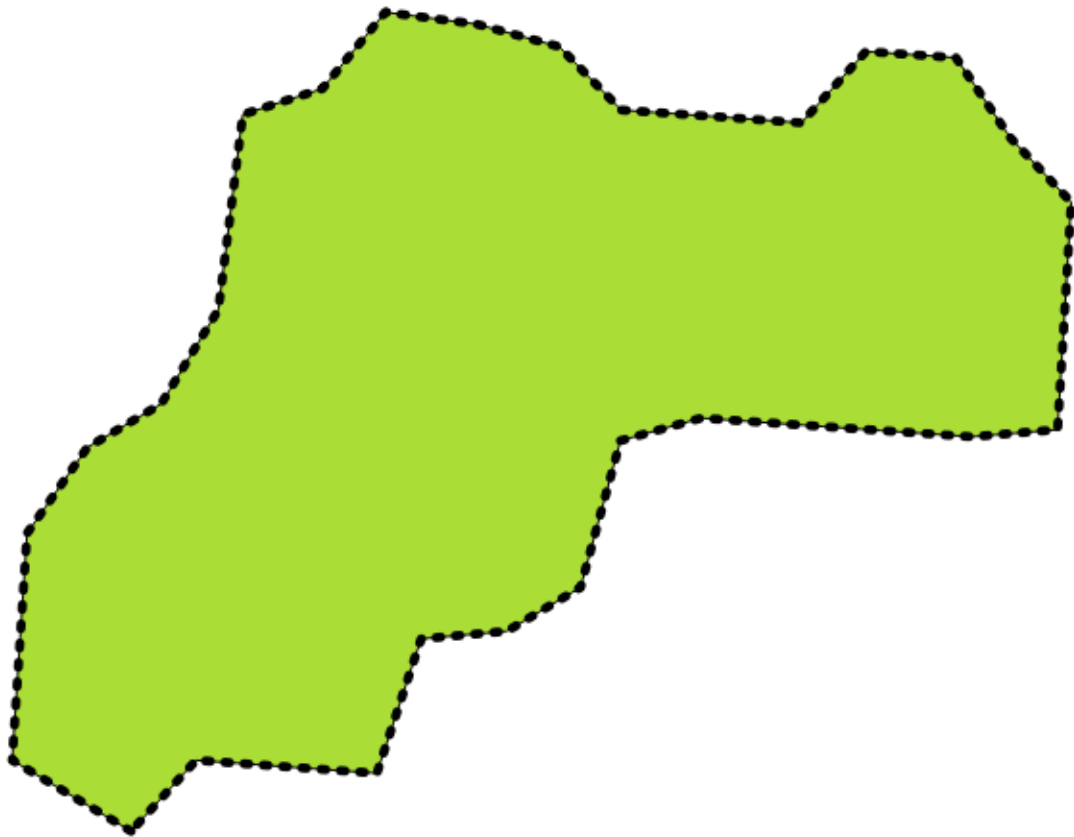


图 24.41: Boundaries (black dashed line) of the source polygon layer

For **lines geometries**, the boundaries are their end points.

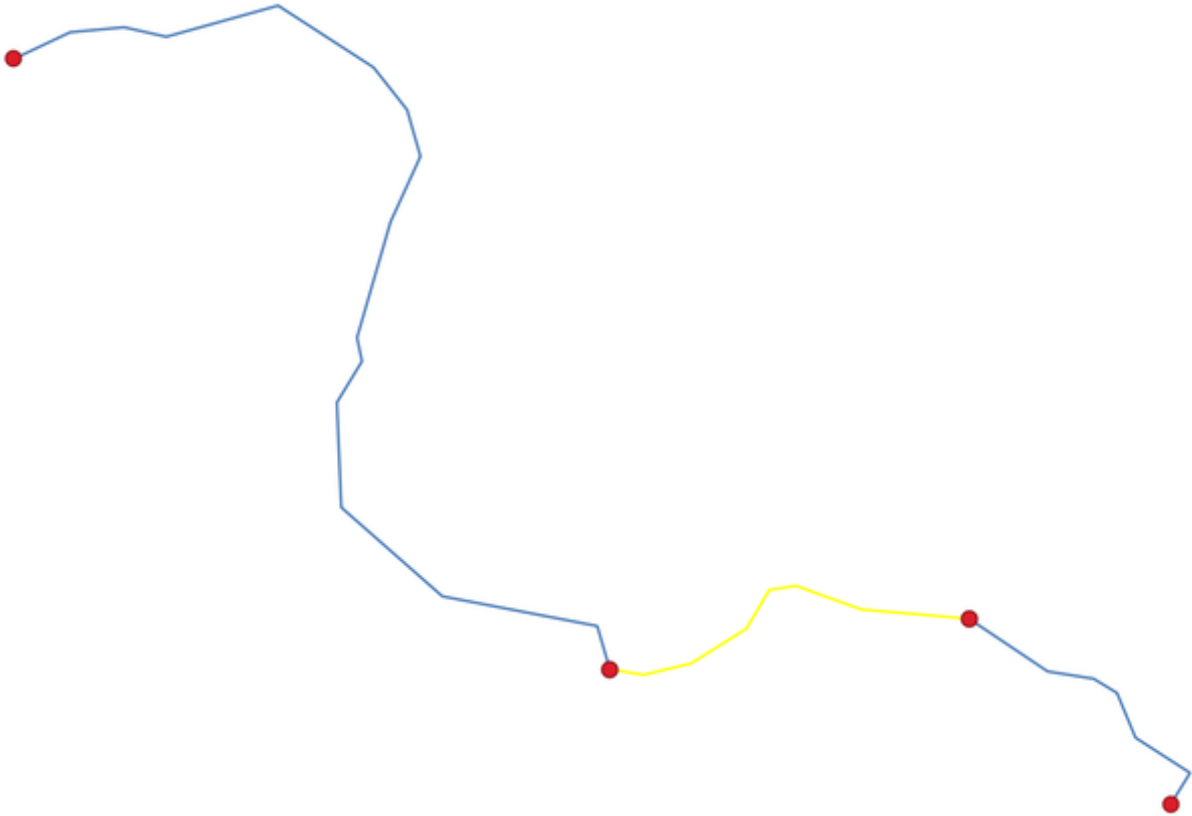


图 24.42: Boundary layer (red points) for lines. In yellow a selected feature.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Boundary	OUTPUT	[vector: point, line] Default: [Create temporary layer]	Specify the output (boundary) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Boundary	OUTPUT	[vector: point, line]	Boundaries from the input layer (point for line, and line for polygon)

Python code

Algorithm ID: native:boundary

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Bounding boxes

Calculates the bounding box (envelope) of each feature in an input layer. Polygon and line geometries are supported.

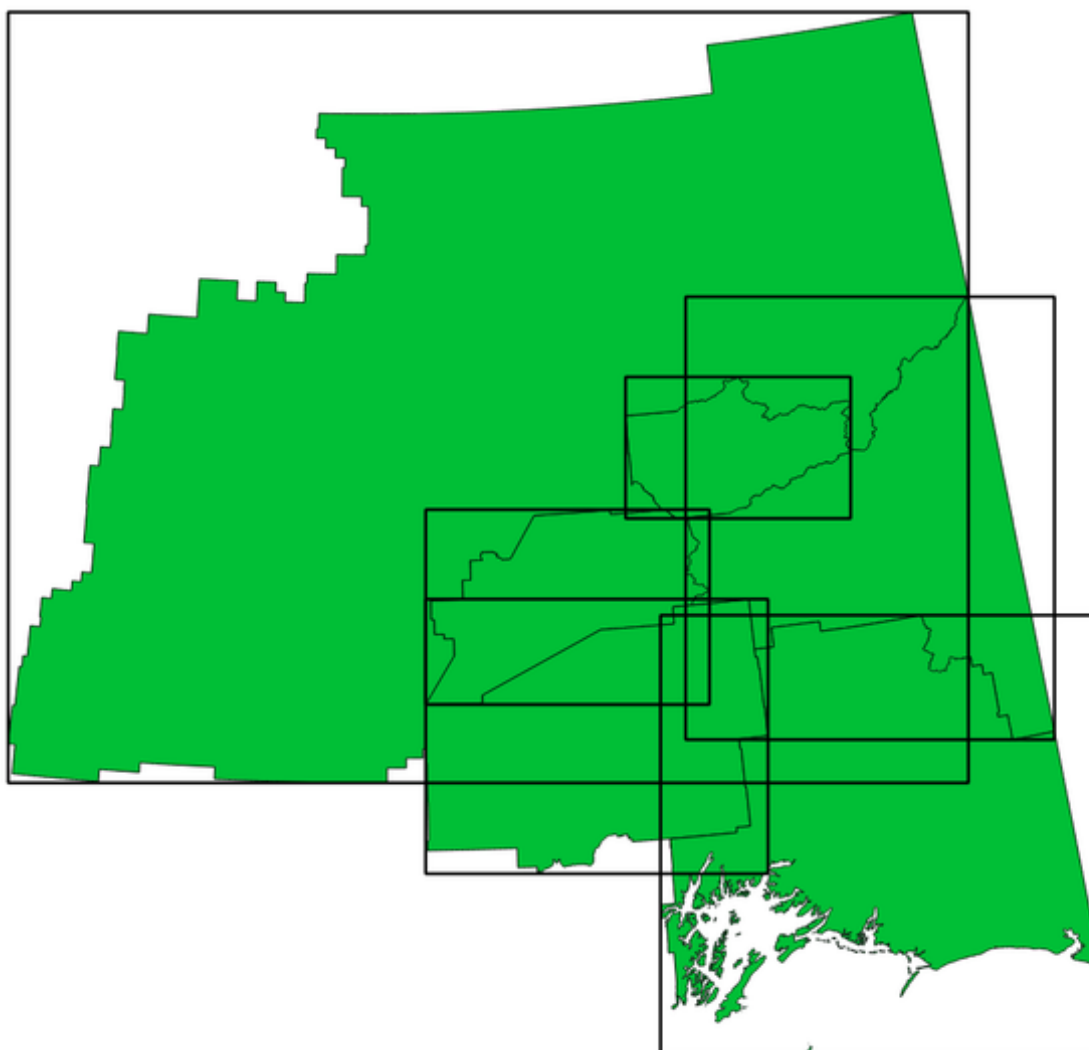


图 24.43: Black lines represent the bounding boxes of each polygon feature

Allows *features in-place modification*

参见:

Minimum bounding geometry

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Bounds	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output (bounding box) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Bounds	OUTPUT	[vector: polygon]	Bounding boxes of input layer

Python code

Algorithm ID: native:boundingboxes

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Buffer

Computes a buffer area for all the features in an input layer, using a fixed distance.

It is possible to use a negative distance for polygon input layers. In this case the buffer will result in a smaller polygon (setback).

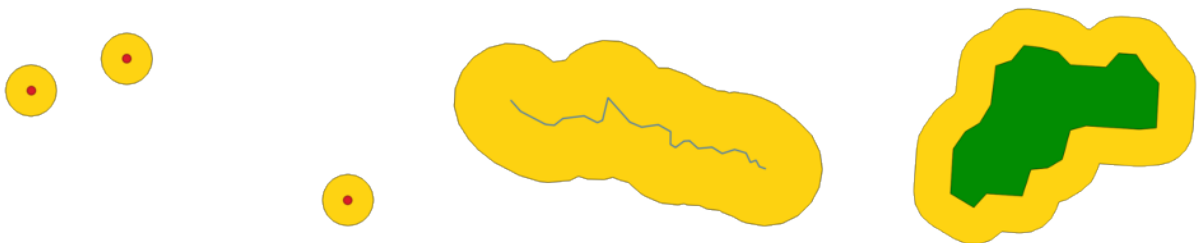


图 24.44: Buffer (in yellow) of points, line and polygon


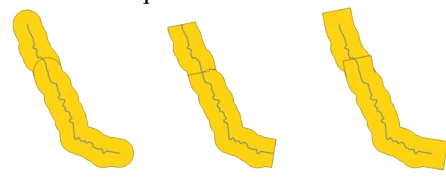
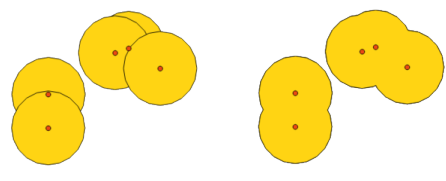
Allows *features in-place modification*

Default menu: Vector ► Geoprocessing Tools

参见:

Variable distance buffer, *Multi-ring buffer (constant distance)*, *Variable width buffer (by M value)*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Distance	DISTANCE	[number ] Default: 10.0	Buffer distance (from the boundary of each feature). You can use the Data Defined button on the right to choose a field from which the radius will be calculated. This way you can have different radius for each feature (see <i>Variable distance buffer</i>).
Segments	SEGMENTS	[number] Default: 5	Controls the number of line segments to use to approximate a quarter circle when creating rounded offsets.
End cap style	END_CAP_STYLE	[enumeration] Default: 0	Controls how line endings are handled in the buffer. One of: <ul style="list-style-type: none"> • 0 --- Round • 1 --- Flat • 2 --- Square  <p>图 24.45: Round, flat and square cap styles</p>
Join style	JOIN_STYLE	[enumeration] Default: 0	Specifies whether round, miter or beveled joins should be used when offsetting corners in a line. Options are: <ul style="list-style-type: none"> • 0 --- Round • 1 --- Miter • 2 --- Bevel
Miter limit	MITER_LIMIT	[number] Default: 2.0	Controls the maximum distance from the offset curve to use when creating a mitered join (only applicable for miter join styles). Minimum: 1.
Dissolve result	DISSOLVE	[boolean] Default: False	Dissolve the final buffer. If True (checked), overlapping buffers will be dissolved (combined) into a new feature.  <p>图 24.46: Standard and dissolved buffer</p>
Buffered	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output (buffer) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Buffered	OUTPUT	[vector: polygon]	Output (buffer) polygon layer

Python code

Algorithm ID: native:buffer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Centroids

Creates a new point layer, with points representing the centroids of the geometries of the input layer.

The centroid is a single point representing the barycenter (of all parts) of the feature, so it can be outside the feature borders. But can also be a point on each part of the feature.

The attributes of the points in the output layer are the same as for the original features.

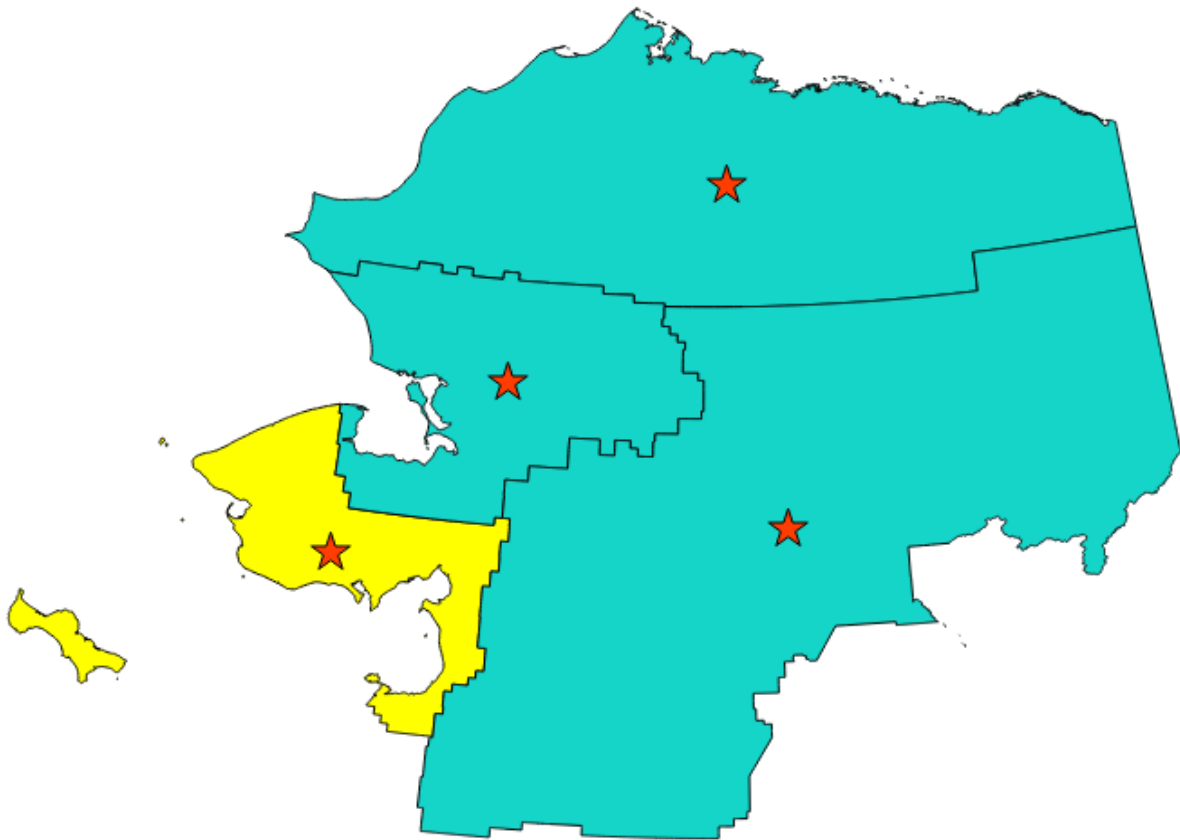


图 24.47: The red stars represent the centroids of the features of the input layer.


 Allows *features in-place modification*

Default menu: *Vector ► Geometry Tools*

参见:

Point on Surface

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Create centroid for each part	ALL_PARTS	[boolean  Default: False	If True (checked), a centroid will be created for each part of the geometry
Centroids	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output (centroid) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Centroids	OUTPUT	[vector: point]	Output point vector layer (centroids)

Python code

Algorithm ID: `native:centroids`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Check validity

Performs a validity check on the geometries of a vector layer.

The geometries are classified in three groups (valid, invalid and error) and for each group, a vector layer with its features is generated:

- The **Valid output** layer contains only the valid features (without topological errors).
- The **Invalid output** layer contains all the invalid features found by the algorithm.
- The **Error output** layer is a point layer that points to where the invalid features were found.

The attribute tables of the generated layers will contain some additional information ("message" for the **error** layer, "FID" and "_errors" for the **invalid** layer and only "FID" for the **valid** layer):

The attribute table of each generated vector layer will contain some additional information (number of errors found and types of error):

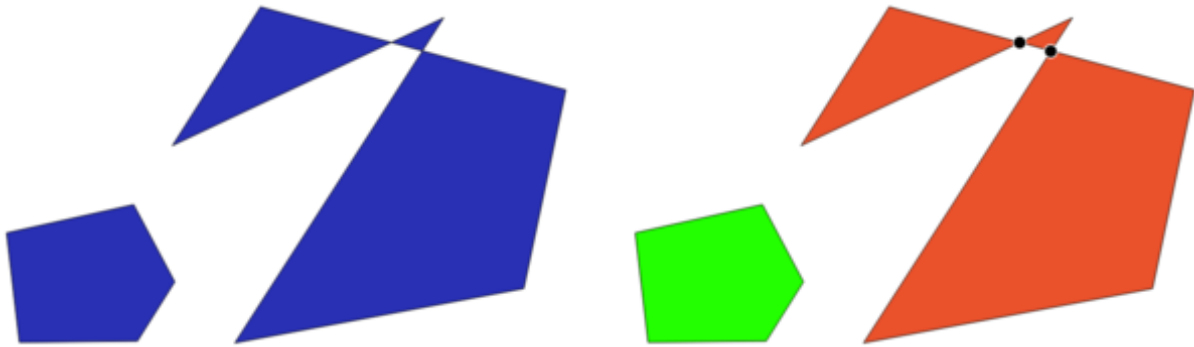


图 24.48: Left: the input layer. Right: the valid layer (green), the invalid layer (orange)

Default menu: *Vector ► Geometry Tools*

参见:

Fix geometries and the core plugin *Geometry Checker Plugin*

Parameters

Label	名称	Type	Description
Input layer	INPUT_LAYER	[vector: any]	Input vector layer
Method	METHOD	[enumeration] Default: 2	Method to use to check validity. Options: <ul style="list-style-type: none"> • 0: The one selected in digitizing settings • 1: QGIS • 2: GEOS
Ignore ring self intersection	IGNORE_RING_SELF_INTERSECTION	[boolean] Default: False	Ignore self intersecting rings when checking for validity.
Valid output	VALID_OUTPUT	[same as input] Default: [Create temporary layer]	Specify the vector layer to contain a copy of the valid features of the source layer. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Invalid output	INVALID_OUTPUT	[same as input] Default: [Create temporary layer]	Vector layer containing copy of the invalid features of the source layer with the field <code>_errors</code> listing the summary of the error(s) found. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

下页继续

表 24.95 - 续上页

Label	名称	Type	Description
Error output	ERROR_OUTPUT	[vector: point] Default: [Create temporary layer]	Point layer of the exact position of the validity problems detected with the message field describing the error(s) found. One of: <ul style="list-style-type: none"> • Skip output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Count of errors	ERROR_COUNT	[number]	The number of geometries that caused errors.
Error output	ERROR_OUTPUT	[vector: point]	Point layer of the exact position of the validity problems detected with the message field describing the error(s) found.
Count of invalid features	INVALID_COUNT	[number]	The number of invalid geometries.
Invalid output	INVALID_OUTPUT	[same as input]	Vector layer containing copy of the invalid features of the source layer with the field <code>_errors</code> listing the summary of the error(s) found.
Count of valid features	VALID_COUNT	[number]	The number of valid geometries.
Valid output	VALID_OUTPUT	[same as input]	Vector layer containing a copy of the valid features of the source layer.

Python code

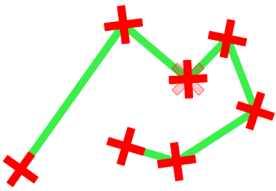
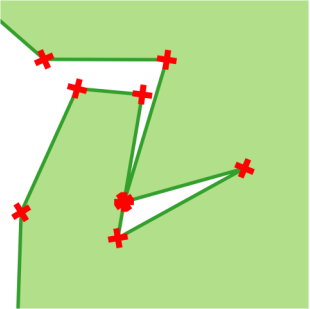
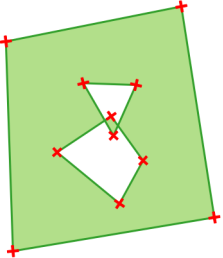
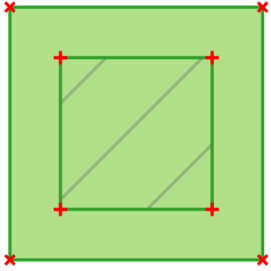
Algorithm ID: qgis:checkvalidity

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Types of error messages and their meanings

表 24.97: If the GEOS method is used the following error messages can occur:

Error message	Explanation	Example
Repeated point	This error happens when a given vertex is repeated.	
Ring self-intersection	This error happens when a geometry touches itself and generates a ring.	
Self-intersection	This error happens when a geometry touches itself.	
Topology validation error		
Hole lies outside shell		
Holes are nested		
Interior is disconnected		
Nested shells	This error happens when a polygon geometry is on top of another polygon geometry.	

下页继续

表 24.97 - 续上页

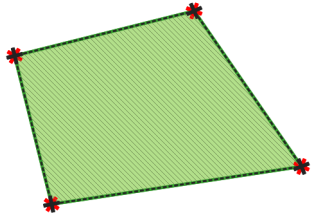
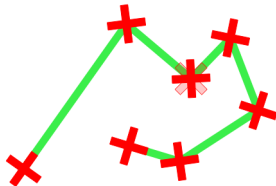
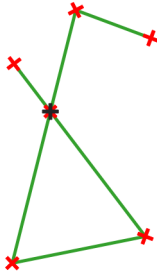
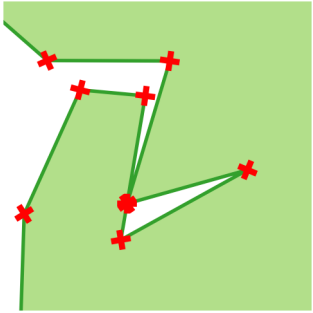
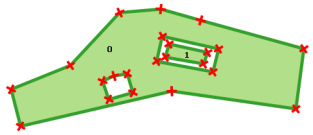
Error message	Explanation	Example
Duplicate rings	This error happens when two rings (exterior or interior) of a polygon geometry are identical	
Too few points in geometry component		
Invalid coordinate	For a point geometry, this error happens when the geometry does not have a proper coordinate pair. The coordinate pair does not contain a latitude value and a longitude value in that order.	
Ring is not closed		

表 24.98: If the QGIS method is used the following error messages can occur:

Error message	Explanation	Example
Segment %1 of ring %2 of polygon %3 intersects segment %4 of ring %5 of polygon %6 at %7		
Ring %1 with less than four points		
Ring %1 not closed		
Line %1 with less than two points		
Line %1 contains %n duplicate node(s) at %2	This error happens when consecutive points on a line have the same coordinates.	
Segments %1 and %2 of line %3 intersect at %4	This error happens when a line self intersects (two segments of the line intersect each other).	

下页继续

表 24.98 - 续上页

Error message	Explanation	Example
Ring self-intersection	This error happens when an outer or inner (island) ring / boundary of a polygon geometry intersects itself.	
Ring %1 of polygon %2 not in exterior ring		
Polygon %1 lies inside polygon %2	This error happens when a part of a MultiPolygon geometry is inside a hole of a MultiPolygon geometry.	

Collect geometries

Takes a vector layer and collects its geometries into new multipart geometries.

One or more attributes can be specified to collect only geometries belonging to the same class (having the same value for the specified attributes), alternatively all geometries can be collected.

All output geometries will be converted to multi geometries, even those with just a single part. This algorithm does not dissolve overlapping geometries - they will be collected together without modifying the shape of each geometry part.

See the 'Promote to multipart' or 'Aggregate' algorithms for alternative options.

Default menu: *Vector ► Geometry Tools*

参见:

Aggregate, Promote to multipart, 融合 (dissolve)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Unique ID fields	FIELD	[tablefield: any] [list]	Choose one or more attributes to collect the geometries
Collected	OUTPUT	[same as input]	Vector layer with collected geometries

Outputs

Label	名称	Type	Description
Collected	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer for the collected geometries. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Python code

Algorithm ID: native:collect

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Concave hull (alpha shapes)

Computes the concave hull of the features in an input point layer.

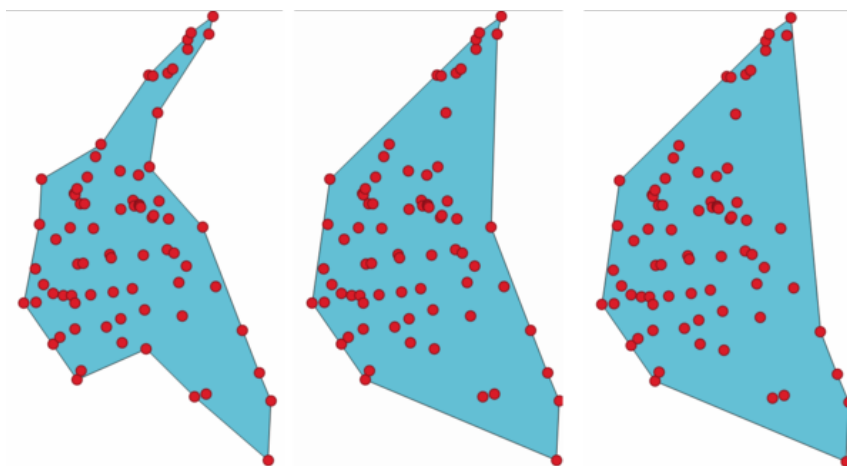


图 24.49: Concave hulls with different thresholds (0.3, 0.6, 0.9)

参见:

Convex hull, Concave hull (k-nearest neighbor)

Parameters

Label	名称	Type	Description
Input point layer	INPUT	[vector: point]	Input point vector layer
Threshold	ALPHA	[number] Default: 0.3	Number from 0 (maximum concave hull) to 1 (convex hull).
Allow holes	HOLES	[boolean] Default: True	Choose whether to allow holes in the final concave hull
Split multipart geometry into singlepart geometries	NO_MULTIGEOMETRY	[boolean] Default: True	Check if you want to have singlepart geometries instead of multipart ones.
Concave hull	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Concave hull	OUTPUT	[vector: polygon]	The output vector layer

Python code

Algorithm ID: qgis:concavehull

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Concave hull (k-nearest neighbor)

Generates a concave hull polygon from a set of points. If the input layer is a line or polygon layer, it will use the vertices.

The number of neighbors to consider determines the concaveness of the output polygon. A lower number will result in a concave hull that follows the points very closely, while a higher number will have a smoother shape. The minimum number of neighbor points to consider is 3. A value equal to or greater than the number of points will result in a convex hull.

If a field is selected, the algorithm will group the features in the input layer using unique values in that field and generate individual polygons in the output layer for each group.

参见:

Concave hull (alpha shapes)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Number of neighboring points to consider (a lower number is more concave, a higher number is smoother)	KNEIGHBORS	[number] Default: 3	Determines the concaveness of the output polygon. A small number will result in a concave hull that follows the points very closely, while a high number will make the polygon look more like the convex hull (if the number is equal to or larger than the number of features, the result will be the convex hull). Minimum value: 3.
Field Optional	FIELD	[tablefield: any] Default: None	If specified, one concave hull polygon is generated for each unique value of the field (by selecting features using this value).
Concave hull	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Concave hull	OUTPUT	[vector: polygon]	The output vector layer

Python code

Algorithm ID: qgis:knearestconcavehull

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Convert geometry type

Generates a new layer based on an existing one, with a different type of geometry.

The attribute table of the output layer is the same as the one of the input layer.

Not all conversions are possible. For instance, a line layer can be converted to a point layer, but a point layer cannot be converted to a line layer.

参见:

Polygonize, Lines to polygons, Polygons to lines, Points to path

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
New geometry type	TYPE	[enumeration] Default: 0	Geometry type to apply to the output features. One of: <ul style="list-style-type: none"> • 0 --- Centroids • 1 --- Nodes • 2 --- Linestrings • 3 --- Multilinestrings • 4 --- Polygons
Converted	OUTPUT	[vector: any] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Converted	OUTPUT	[vector: any]	Output vector layer - the type depends on the parameters

Python code

Algorithm ID: qgis:convertgeometrytype

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Convert to curved geometries

Converts a geometry into its curved geometry equivalent.

Already curved geometries will be retained without change.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line or polygon]	Input vector layer
Maximum distance tolerance	DISTANCE	[number] Default: 0.000001	The maximum distance allowed between the original location of vertices and where they would fall on the converted curved geometries
Maximum angle tolerance	ANGLE	[number] Default: 0.000001	Segments are considered as suitable for replacing with an arc if the points are all regularly spaced on the candidate arc. This parameter specifies the maximum angular deviation (in degrees) allowed when testing for regular point spacing. Between 0 and 45°.
Curves	OUTPUT	[vector: compound-curve or curvopolygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to Database Table... • Append to Layer... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Curves	OUTPUT	[vector: compound-curve or curvopolygon]	Output vector layer with curved geometries

Python code

Algorithm ID: native:converttocurves

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Convex hull

Calculates the convex hull for each feature in an input layer.

See the 'Minimum bounding geometry' algorithm for a convex hull calculation which covers the whole layer or grouped subsets of features.

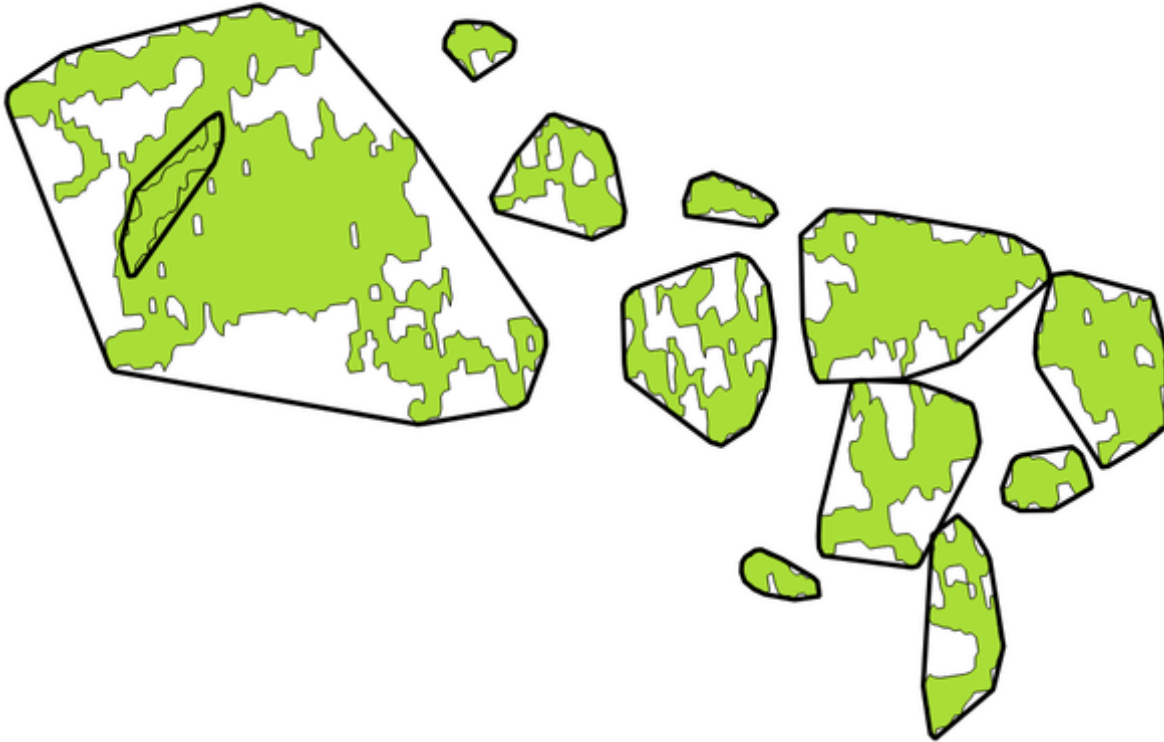


图 24.50: Black lines identify the convex hull for each layer feature

Allows *features in-place modification*

Default menu: *Vector ► Geoprocessing Tools*

参见:

Minimum bounding geometry, Concave hull (alpha shapes)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Convex hull	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Convex hull	OUTPUT	[vector: polygon]	The output (convex hull) vector layer

Python code

Algorithm ID: native:convexhull

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create layer from extent

Creates a new vector layer that contains a single feature with geometry matching the extent of the input layer.

It can be used in models to convert a literal extent (xmin, xmax, ymin, ymax format) into a layer which can be used for other algorithms which require a layer based input.

参见:

Create layer from point

Parameters

Label	名称	Type	Description
Extent (xmin, xmax, ymin, ymax)	INPUT	[extent]	Input extent
Extent	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Extent	OUTPUT	[vector: polygon]	The output (extent) vector layer

Python code

Algorithm ID: native:extenttolayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create layer from point

Creates a new vector layer that contains a single feature with geometry matching a point parameter. It can be used in models to convert a point into a point layer for algorithms which require a layer based input.

参见:

Create layer from extent

Parameters

Label	名称	Type	Description
Point	INPUT	[coordinates]	Input point, including CRS info (example: 397254, 6214446 [EPSG:32632]). If the CRS is not provided, the Project CRS will be used. The point can be specified by clicking on the map canvas.
Point	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Point	OUTPUT	[vector: point]	The output point vector layer containing the input point.

Python code

Algorithm ID: native:pointtolayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Create wedge buffers

Creates wedge shaped buffers from input points.




图 24.51: Wedge buffers

The native output from this algorithm are CurvePolygon geometries, but these may be automatically segmented to Polygons depending on the output format.

参见:

Buffer, Variable width buffer (by M value), Tapered buffers

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Input point vector layer
Azimuth (degrees from North)	AZIMUTH	[number ] Default: 0.0	Angle (in degrees) as the middle value of the wedge

下页继续

表 24.101 - 续上页


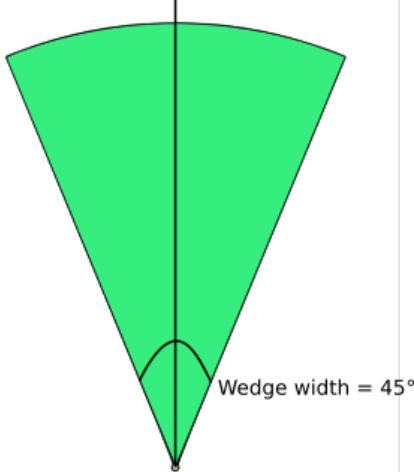


Label	名称	Type	Description
Wedge width (in degrees)	WIDTH	[number ] Default: 45.0	Width (in degrees) of the buffer. The wedge will extend to half of the angular width either side of the azimuth direction. 
Outer radius	OUTER_RADIUS	[number ] Default: 1.0	The outer <i>size</i> (length) of the wedge: the size is meant from the source point to the edge of the wedge shape.
Inner radius Optional	INNER_RADIUS	[number ] Default: 0.0	Inner radius value. If 0 the wedge will begin from the source point.
Buffers	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

图 24.52: Azimuth and width values of the wedge buffer

Outputs

Label	名称	Type	Description
Buffers	OUTPUT	[vector: polygon]	The output (wedge buffer) vector layer

Python code

Algorithm ID: native:wedgebuffers

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Delaunay triangulation

Creates a polygon layer with the Delaunay triangulation corresponding to the input point layer.

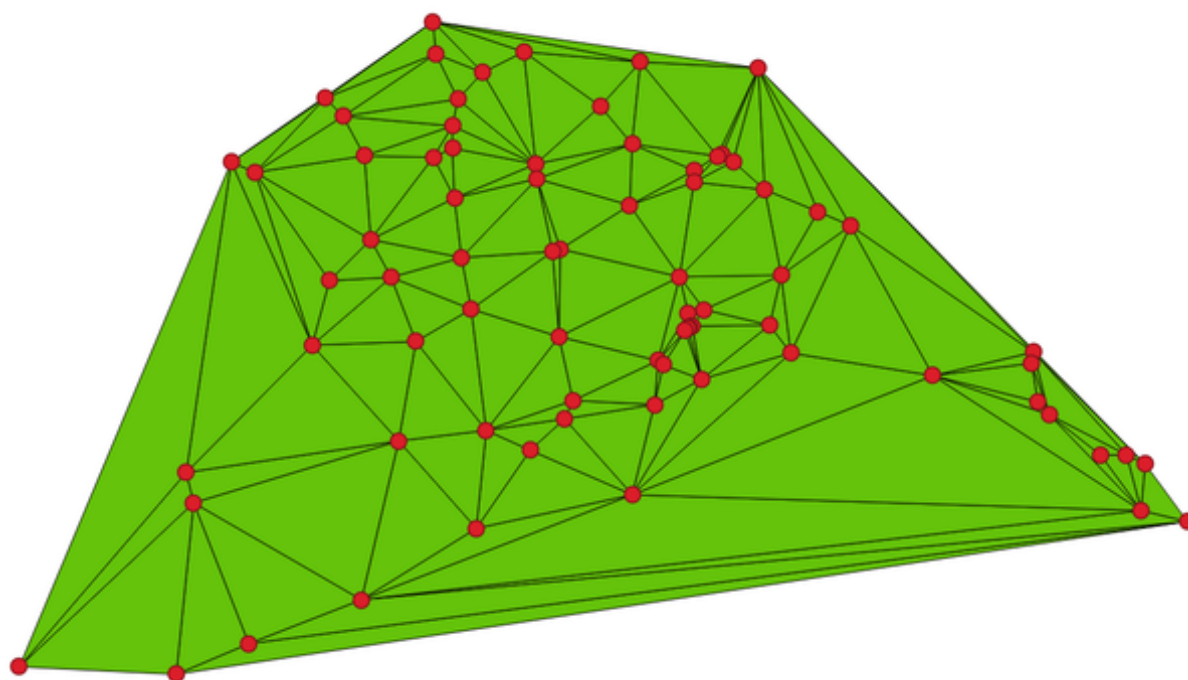


图 24.53: Delaunay triangulation on points

Default menu: *Vector ► Geometry Tools*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Input point vector layer
Delaunay triangulation	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Delaunay triangulation	OUTPUT	[vector: polygon]	The output (Delaunay triangulation) vector layer

Python code

Algorithm ID: qgis:delatunaytriangulation

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Delete holes

Takes a polygon layer and removes holes in polygons. It creates a new vector layer in which polygons with holes have been replaced by polygons with only their external ring. Attributes are not modified.

An optional minimum area parameter allows removing only holes which are smaller than a specified area threshold. Leaving this parameter at 0.0 results in all holes being removed.

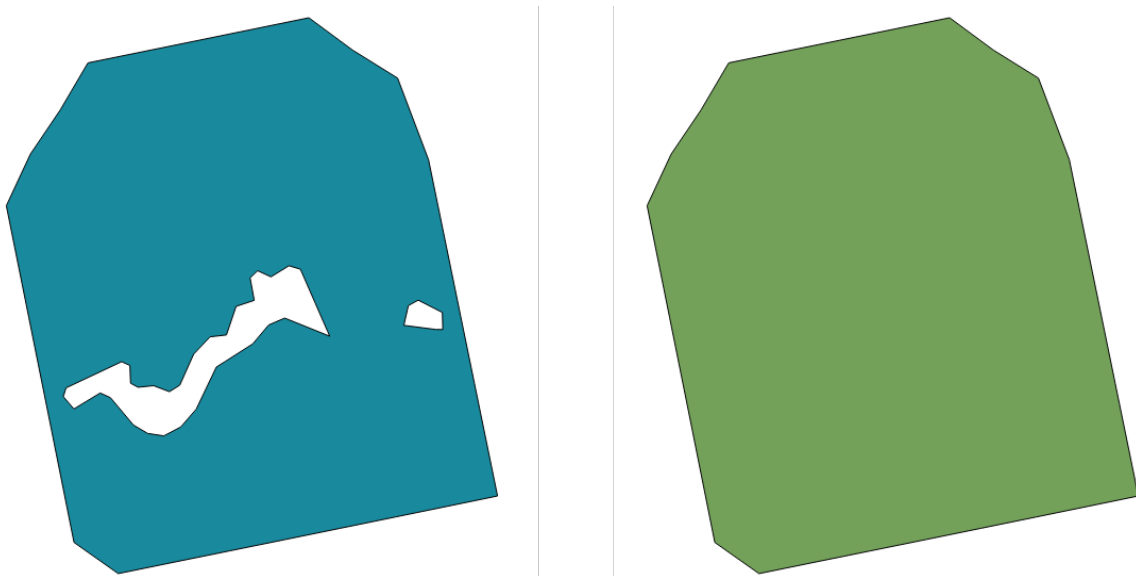



图 24.54: Before and after the cleaning

Allows *features in-place modification*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Input polygon vector layer
Remove holes with area less than Optional	MIN_AREA	[number ] Default: 0.0	Only holes with an area less than this threshold will be deleted. With a value of 0.0, all the holes will be deleted.
Cleaned	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Cleaned	OUTPUT	[same as input]	The output (cleaned) vector layer

Python code

Algorithm ID: native:deleteholes

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Densify by count

Takes a polygon or line layer and generates a new one in which the geometries have a larger number of vertices than the original one.

If the geometries have Z or M values present then these will be linearly interpolated at the added vertices.

The number of new vertices to add to each segment is specified as an input parameter.

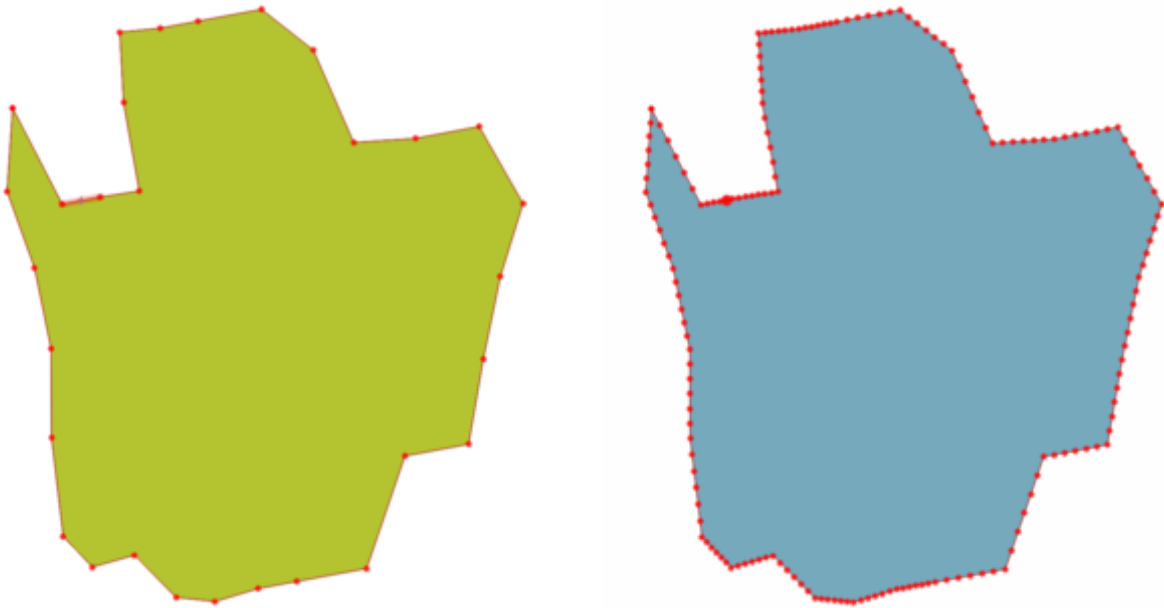


图 24.55: Red points show the vertices before and after the densify

Allows *features in-place modification*

Default menu: *Vector ► Geometry Tools*

参见:

Densify by interval

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Vertices to add	VERTICES	[number] Default: 1	Number of vertices to add to each segment
Densified	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Densified	OUTPUT	[same as input]	The output (densified) vector layer

Python code

Algorithm ID: native:densifygeometries

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Densify by interval

Takes a polygon or line layer and generates a new one in which the geometries have a larger number of vertices than the original one.

The geometries are densified by adding regularly placed extra vertices inside each segment so that the maximum distance between any two vertices does not exceed the specified distance.

If the geometries have Z or M values present then these will be linearly interpolated at the added vertices.

Example

Specifying a distance of 3 would cause the segment $[0\ 0] \rightarrow [10\ 0]$ to be converted to $[0\ 0] \rightarrow [2.5\ 0] \rightarrow [5\ 0] \rightarrow [7.5\ 0] \rightarrow [10\ 0]$, since 3 extra vertices are required on the segment and spacing these at 2.5 increments allows them to be evenly spaced over the segment.

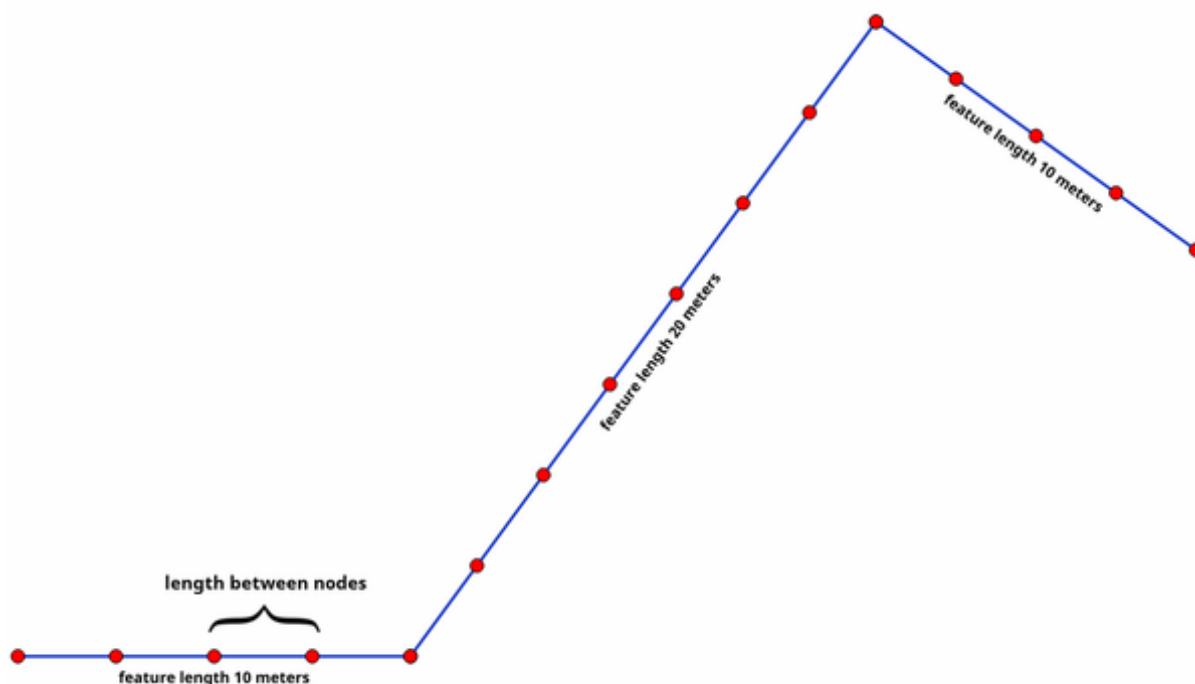



图 24.56: Densify geometry at a given interval

Allows *features in-place modification*

参见:

Densify by count

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Interval between vertices to add	INTERVAL	[number ] Default: 1.0	Maximum distance between two consecutive vertices
Densified	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Densified	OUTPUT	[same as input]	The output (densified) vector layer

Python code

Algorithm ID: native:densifygeometriesgivenaninterval

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

融合 (dissolve)

Takes a vector layer and combines its features into new features. One or more attributes can be specified to dissolve features belonging to the same class (having the same value for the specified attributes), alternatively all features can be dissolved to a single feature.

All output geometries will be converted to multi geometries. In case the input is a polygon layer, common boundaries of adjacent polygons being dissolved will get erased.

The resulting attribute table will have the same fields as the input layer. The values in the output layer's fields are the ones of the first input feature that happens to be processed.

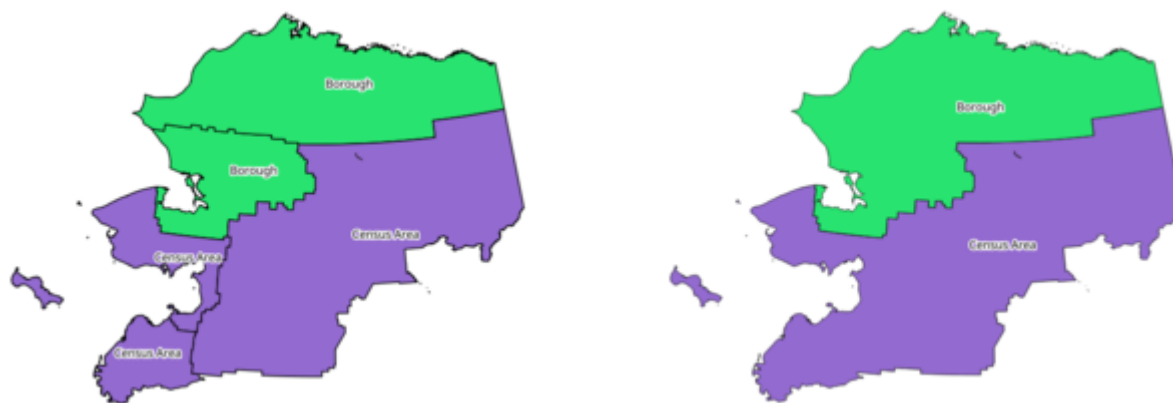


图 24.57: Dissolve the polygon layer on a common attribute

Default menu: *Vector ► Geoprocessing Tools*

参见:

Aggregate, Collect geometries

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Dissolve field(s) Optional	FIELD	[tablefield: any] [list] Default: []	Features having the same value for the selected field(s) will be replaced with a single one and their geometries are merged. If no field is provided then all the features are dissolved, resulting in a single (multi-part) feature.
Dissolved	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Dissolved	OUTPUT	[same as input]	The output vector layer with dissolved geometries

Python code

Algorithm ID: native:dissolve

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Drape (set Z value from raster)



Uses values sampled from a band within a raster layer to set the Z value for every overlapping vertex in the feature geometry. The raster values can optionally be scaled by a preset amount.

If Z values already exist in the layer, they will be overwritten with the new value. If no Z values exist, the geometry will be upgraded to include the Z dimension.

参见:

Set M value from raster, Set Z value

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Raster layer	RASTER	[raster]	Raster layer with Z values
Band number	BAND	[raster band] Default: 1	The raster band to take the Z values from
Value for no-data or non-intersecting vertices	NODATA	[number  Default: 0	Value to use in case the vertex does not intersect (a valid pixel of) the raster
Scale factor	SCALE	[number  Default: 1.0	Scaling value: the band values are multiplied by this value.
Updated	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer (with Z values from the raster layer). One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Updated	OUTPUT	[same as input]	The output vector layer with Z values from the raster layer

Python code

Algorithm ID: native:setzfromraster

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Drop M/Z values

Removes M (measure) or Z (altitude) values from input geometries.

参见:

Set M value, Set Z value

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer with M or Z values
Drop M Values	DROP_M_VALUES	[boolean] Default: False	Removes the M values from the geometries
Drop Z Values	DROP_Z_VALUES	[boolean] Default: False	Removes the Z values from the geometries
Z/M Dropped	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Z/M Dropped	OUTPUT	[same as input]	The output vector layer (identical to the input layer, except that the M and/or Z dimensions have been removed from the geometries).

Python code

Algorithm ID: native:dropmzvalues

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Eliminate selected polygons

Combines selected polygons of the input layer with certain adjacent polygons by erasing their common boundary. The adjacent polygon can be either the one with the largest or smallest area or the one sharing the largest common boundary with the polygon to be eliminated.

Eliminate is normally used to get rid of sliver polygons, i.e. tiny polygons that are a result of polygon intersection processes where boundaries of the inputs are similar but not identical.

Default menu: *Vector ► Geoprocessing Tools*

参见:

Fix geometries

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Input polygon vector layer
Merge selection with the neighboring polygon with the	MODE	[enumeration] Default: None	Choose the parameter to use in order to get rid of the selected polygons: <ul style="list-style-type: none"> • 0 --- Largest Area • 1 --- Smallest Area • 2 --- Largest Common Boundary
Eliminated	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Eliminated	OUTPUT	[vector: polygon]	The output polygon vector layer.

Python code

Algorithm ID: qgis:eliminateselectedpolygons

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Explode lines

Takes a lines layer and creates a new one in which each line layer is replaced by a set of lines representing the segments in the original line.

Each line in the resulting layer contains only a start and an end point, with no intermediate vertices between them.



图 24.58: The original line layer and the exploded one

Allows *features in-place modification*

参见:

Subdivide, Line substring

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Exploded	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Exploded	OUTPUT	[vector: line]	The output line vector layer with features representing each segment of the input layer.

Python code

Algorithm ID: native:explodelines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extend lines

Extends line geometry by a specified amount at the start and end of the line.

Lines are extended using the bearing of the first and last segment in the line.

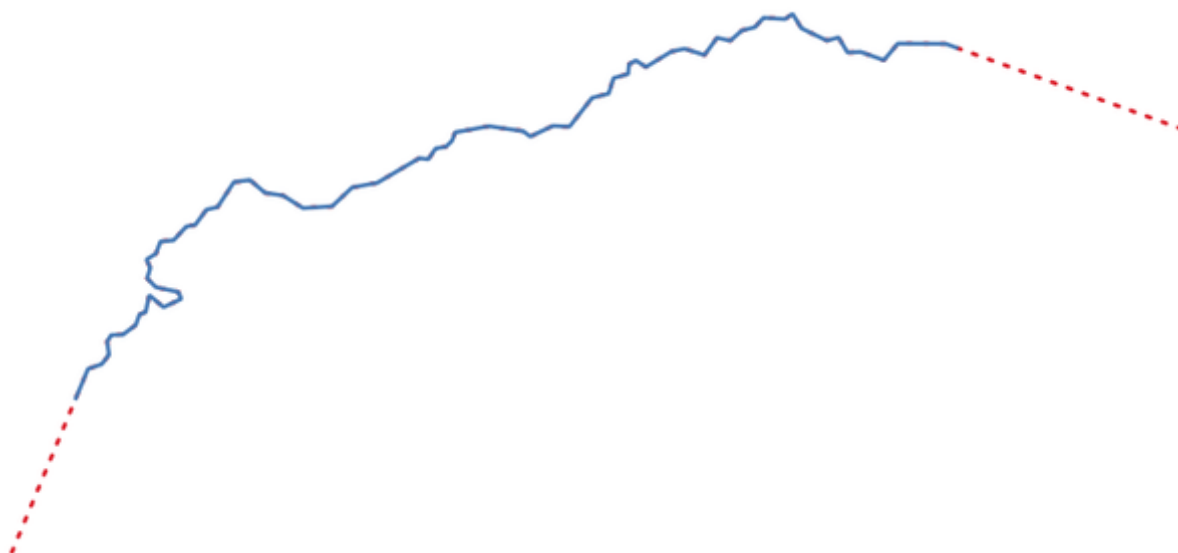




图 24.59: The red dashes represent the initial and final extension of the original layer

Allows *features in-place modification*

参见:

Line substring

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Start distance	START_DISTANCE	[number ]	Distance by which to extend the first segment of the line (starting point)
End distance	END_DISTANCE	[number ]	Distance by which to extend the last segment of the line (ending point)
Extended	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Extended	OUTPUT	[vector: line]	The output (extended) line vector layer.

Python code

Algorithm ID: native:extendlines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extract M values

Extracts M values from geometries into feature attributes.

By default only the M value from the first vertex of each feature is extracted, however the algorithm can optionally calculate statistics on all of the geometry's M values, including sum, mean, minimum and maximum.

参见:

Extract Z values, Set M value, Drop M/Z values

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Summaries to calculate	SUMMARIES	[enumeration] Default: [0]	Statistics on the M values of a geometry. One or more of: <ul style="list-style-type: none"> • 0 --- First • 1 --- Last • 2 --- Count • 3 --- Sum • 4 --- Mean • 5 --- Median • 6 --- St.dev (pop) • 7 --- Minimum • 8 --- Maximum • 9 --- Range • 10 --- Minority • 11 --- Majority • 12 --- Variety • 13 --- Q1 • 14 --- Q3 • 15 --- IQR
Output column prefix	COLUMN_PREFIX	[string] Default: 'm_'	The prefix for the output (M) column

下页继续

表 24.103 – 续上页

Label	名称	Type	Description
Extracted	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Extracted	OUTPUT	[same as input]	The output vector layer (with M values)

Python code

Algorithm ID: native:extractmvalues

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Extract specific vertices

Takes a vector layer and generates a point layer with points representing specific vertices in the input geometries.

For instance, this algorithm can be used to extract the first or last vertices in the geometry. The attributes associated to each point are the same ones associated to the feature that the vertex belongs to.

The vertex indices parameter accepts a comma separated string specifying the indices of the vertices to extract. The first vertex corresponds to an index of 0, the second vertex has an index of 1, etc. Negative indices can be used to find vertices at the end of the geometry, e.g., an index of -1 corresponds to the last vertex, -2 corresponds to the second last vertex, etc.

Additional fields are added to the vertices indicating the specific vertex position (e.g., 0, -1, etc), the original vertex index, the vertex' s part and its index within the part (as well as its ring for polygons), distance along the original geometry and bisector angle of vertex for the original geometry.

参见:

[Extract vertices](#), [Filter vertices by M value](#), [Filter vertices by Z value](#)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Vertex indices	VERTICES	[string] Default: '0'	Comma-separated string of the indices of the vertices to extract.
Vertices	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Vertices	OUTPUT	[vector: point]	The output (point) vector layer containing the specified vertices from the input layer geometries.

Python code

Algorithm ID: native:extractspecificvertices

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extract vertices

Takes a vector layer and generates a point layer with points representing the vertices in the input geometries.

The attributes associated to each point are the same ones associated to the feature that the vertex belongs to.

Additional fields are added to the vertices indicating the vertex index (beginning at 0), the feature's part and its index within the part (as well as its ring for polygons), distance along original geometry and bisector angle of vertex for original geometry.

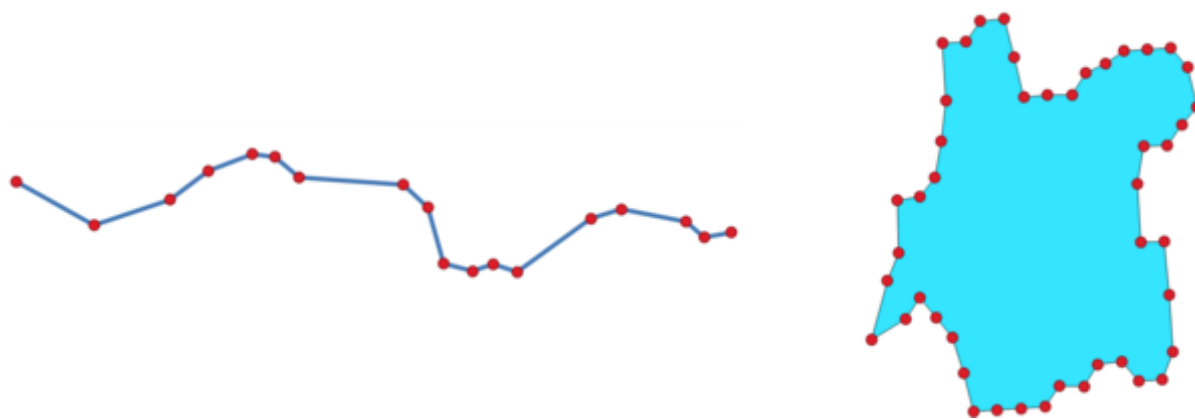


图 24.60: Vertices extracted for line and polygon layer

Default menu: *Vector ► Geometry Tools*

参见:

Extract specific vertices, Filter vertices by M value, Filter vertices by Z value

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Vertices	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Vertices	OUTPUT	[vector: point]	The output (point) vector layer containing the vertices from the input layer geometries.

Python code

Algorithm ID: native:extractvertices

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extract Z values

Extracts Z values from geometries into feature attributes.

By default only the Z value from the first vertex of each feature is extracted, however the algorithm can optionally calculate statistics on all of the geometry's Z values, including sum, mean, minimum and maximum.

参见:

Extract M values, Set Z value, Drop M/Z values

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Summaries to calculate	SUMMARIES	[enumeration] Default: [0]	Statistics on the Z values of a geometry. One or more of: <ul style="list-style-type: none"> • 0 --- First • 1 --- Last • 2 --- Count • 3 --- Sum • 4 --- Mean • 5 --- Median • 6 --- St.dev (pop) • 7 --- Minimum • 8 --- Maximum • 9 --- Range • 10 --- Minority • 11 --- Majority • 12 --- Variety • 13 --- Q1 • 14 --- Q3 • 15 --- IQR
Output column prefix	COLUMN_PREFIX	[string] Default: 'z_'	The prefix for the output (Z) column
Extracted	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Extracted	OUTPUT	[same as input]	The output vector layer (with Z values)

Python code

Algorithm ID: native:extractzvalues

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Filter vertices by M value

Filters away vertices based on their M value, returning geometries with only vertex points that have a M value greater than or equal to the specified minimum value and/or less than or equal to the maximum value.

If the minimum value is not specified then only the maximum value is tested, and similarly if the maximum value is not specified then only the minimum value is tested.

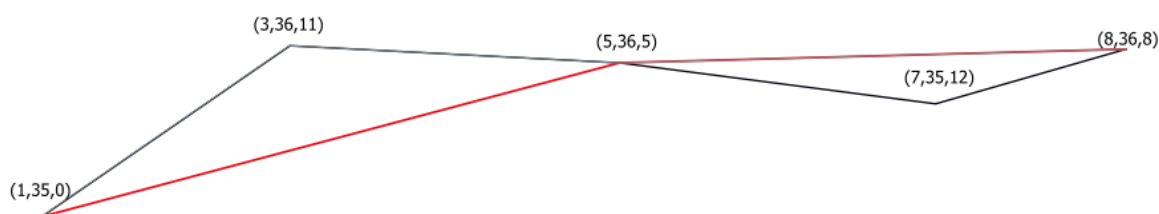




图 24.61: The red line represents the black line with only vertices whose M value is ≤ 10 .

注解: Depending on the input geometry attributes and the filters used, the resultant geometries created by this algorithm may no longer be valid.

参见:

Filter vertices by Z value, Extract vertices, Extract specific vertices

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer to remove vertices from
Minimum Optional	MIN	[number ] Default: <i>Not set</i>	Minimum of M values allowed
Maximum Optional	MAX	[number ] Default: <i>Not set</i>	Maximum of M values allowed
Filtered	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Filtered	OUTPUT	[same as input]	The output vector layer of features with only the filtered vertices.

Python code

Algorithm ID: native:filterverticesbym

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Filter vertices by Z value

Filters away vertices based on their Z value, returning geometries with only vertex points that have a Z value greater than or equal to the specified minimum value and/or less than or equal to the maximum value.

If the minimum value is not specified then only the maximum value is tested, and similarly if the maximum value is not specified then only the minimum value is tested.

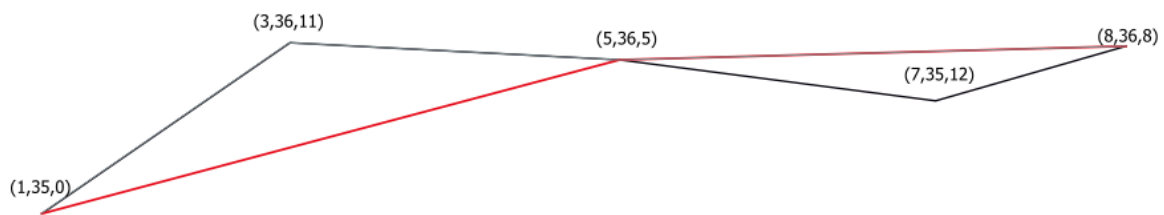




图 24.62: The red line represents the black line with only vertices whose Z value is ≤ 10 .

注解: Depending on the input geometry attributes and the filters used, the resultant geometries created by this algorithm may no longer be valid. You may need to run the *Fix geometries* algorithm to ensure their validity.

参见:

Filter vertices by M value, Extract vertices, Extract specific vertices

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer to remove vertices from
Minimum Optional	MIN	[number ] Default: <i>Not set</i>	Minimum of Z values allowed
Maximum Optional	MAX	[number ] Default: <i>Not set</i>	Maximum of Z values allowed
Filtered	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Filtered	OUTPUT	[same as input]	The output vector layer of features with only the filtered vertices.

Python code

Algorithm ID: native:filterverticesbyz

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Fix geometries

Attempts to create a valid representation of a given invalid geometry without losing any of the input vertices. Already valid geometries are returned without further intervention. Always outputs multi-geometry layer.

注解: M values will be dropped from the output.

 Allows *features in-place modification*

参见:

[Check validity](#)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Fixed geometries	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Fixed geometries	OUTPUT	[same as input]	The output vector layer with fixed geometries.

Python code

Algorithm ID: native:fixgeometries

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Force right-hand-rule

Forces polygon geometries to respect the Right-Hand-Rule, in which the area that is bounded by a polygon is to the right of the boundary. In particular, the exterior ring is oriented in a clockwise direction and any interior rings in a counter-clockwise direction.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Input vector layer
Reoriented	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Reoriented	OUTPUT	[vector: polygon]	The output vector layer with reoriented geometries.

Python code

Algorithm ID: native:forcerhr

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Geodesic line split at antimeridian

Splits a line into multiple geodesic segments, whenever the line crosses the antimeridian (± 180 degrees longitude).

Splitting at the antimeridian helps the visual display of the lines in some projections. The returned geometry will always be a multi-part geometry.

Whenever line segments in the input geometry cross the antimeridian, they will be split into two segments, with the latitude of the breakpoint being determined using a geodesic line connecting the points either side of this segment. The current project ellipsoid setting will be used when calculating this breakpoint.

If the input geometry contains M or Z values, these will be linearly interpolated for the new vertices created at the antimeridian.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Split	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output line vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Split	OUTPUT	[vector: line]	The output line vector layer split at the antimeridian.

Python code

Algorithm ID: native:antimeridiansplit

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Geometry by expression

Updates existing geometries (or creates new geometries) for input features by use of a QGIS expression.

This allows complex geometry modifications which can utilize all the flexibility of the QGIS expression engine to manipulate and create geometries for output features.

For help with QGIS expression functions, see the inbuilt help available in the *expression builder*.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Output geometry type	OUTPUT_GEOMETRY	[enumeration] Default: 0	The output geometry strongly depends on the expression: for instance, if you create a buffer the geometry type has to be polygon. One of: <ul style="list-style-type: none"> • 0 --- Polygon • 1 --- Line • 2 --- Point
Output geometry has z values	WITH_Z	[boolean] Default: False	Choose if the output geometry should include the Z dimension
Output geometry has m values	WITH_M	[boolean] Default: False	Choose if the output geometry should include the M dimension
Geometry expression	EXPRESSION	[expression] Default: '\$geometry'	Add the geometry expression you want to use. You can use the button to open the Expression Dialog. The dialog lists all the relevant expressions, together with their help and guide.
Modified geometry	OUTPUT	[vector: any] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Modified geometry	OUTPUT	[vector: any]	The output vector layer

Python code

Algorithm ID: native:geometrybyexpression

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Interpolate point on line

Creates a point geometry interpolated at a set distance along line or curve geometries.

Z and M values are linearly interpolated from existing values.

If a multipart geometry is encountered, only the first part is considered when calculating the substring.

If the specified distance is greater than the input feature's length, the resultant feature will have a null geometry.




图 24.63: Interpolated point at 500m of the beginning of the line

参见:

Points along geometry

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Distance	DISTANCE	[number ] Default: 0.0	Distance from the beginning of the line
Interpolated points	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Interpolated points	OUTPUT	[vector: point]	The output point vector layer with features at a set distance along the line or polygon boundary

Python code

Algorithm ID: native:interpolatepoint

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Keep N biggest parts

Takes a layer with polygons or multipolygons and returns a new layer in which only the *n* largest polygons of each multipolygon feature are kept. If a feature has *n* or fewer parts, the feature will just be copied.

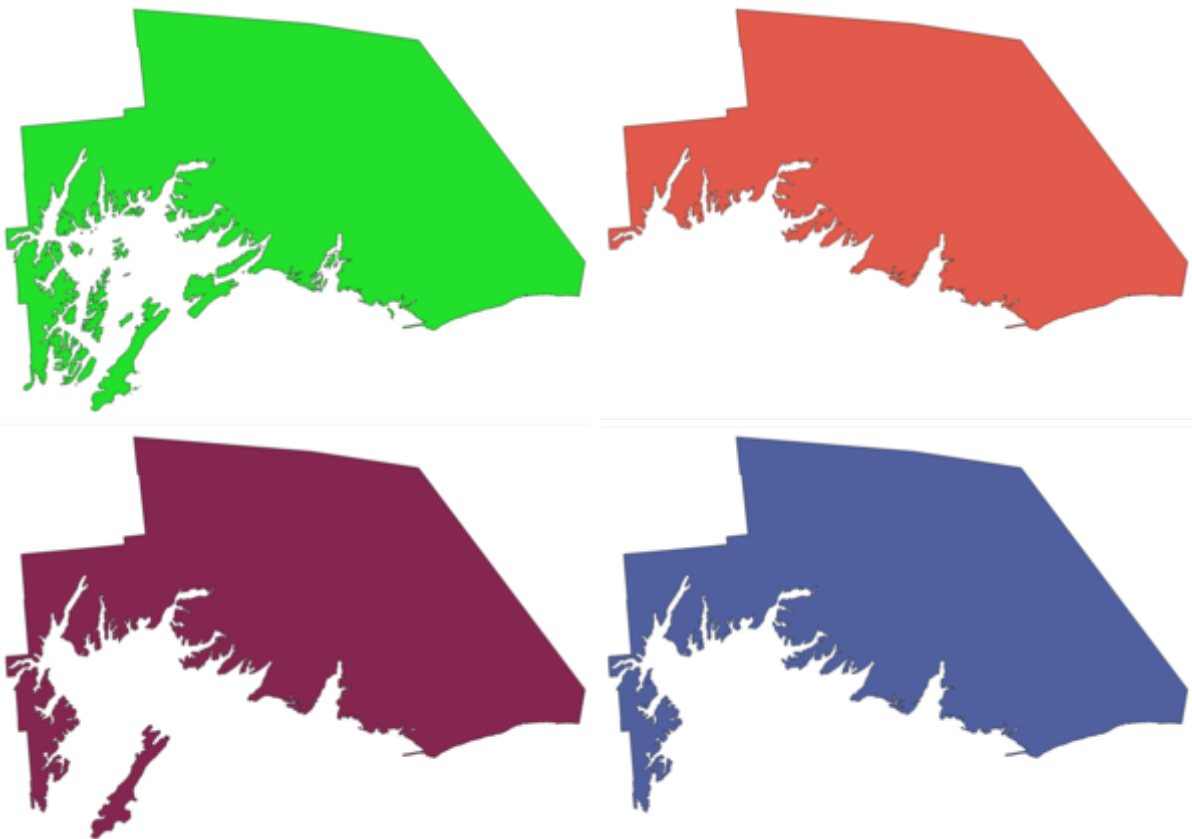


图 24.64: Clockwise from top left: original multipart feature, one, two and three biggest parts kept

Parameters

Label	名称	Type	Description
Polygons	INPUT	[vector: polygon]	Input polygon vector layer
Parts to keep	PARTS	[number] Default: 1	Number of parts to keep. If 1, only the biggest part of the feature will be kept.
Parts	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Parts	OUTPUT	[vector: polygon]	The output polygon vector layer with the N biggest parts of each feature

Python code

Algorithm ID: qgis:keepnbiggestparts

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Line substring

Returns the portion of a line (or curve) which falls between the specified start and end distances (measured from the beginning of the line).

Z and M values are linearly interpolated from existing values.

If a multipart geometry is encountered, only the first part is considered when calculating the substring.

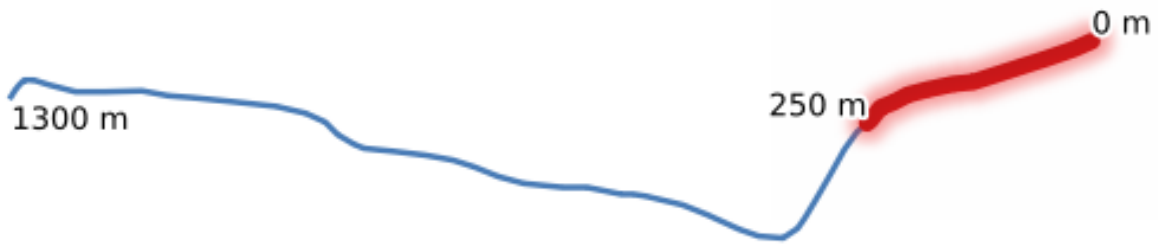




图 24.65: Substring line with starting distance set at 0 meters and the ending distance at 250 meters.

Allows *features in-place modification*

参见:

Extend lines

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Start distance	START_DISTANCE	[number ]	Distance along the input line to the start point of the output feature
End distance	END_DISTANCE	[number ]	Distance along the input line to the end point of the output feature
Substring	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output line vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Substring	OUTPUT	[vector: line]	The output line vector layer.

Python code

Algorithm ID: native:linesubstring

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Lines to polygons

Generates a polygon layer using as polygon rings the lines from an input line layer.

The attribute table of the output layer is the same as the one of the input layer.

Default menu: *Vector ► Geometry Tools*

参见:

Polygons to lines, Polygonize, Convert geometry type

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Polygons	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Polygons	OUTPUT	[vector: polygon]	The output polygon vector layer.

Python code

Algorithm ID: qgis:linestopolygons

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Merge lines

Joins all connected parts of MultiLineString geometries into single LineString geometries.

If any parts of the input MultiLineString geometries are not connected, the resultant geometry will be a Multi-LineString containing any lines which could be merged and any non-connected line parts.

 Allows *features in-place modification*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Merged	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output line vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Merged	OUTPUT	[vector: line]	The output (merged) line vector layer.

Python code

Algorithm ID: native:mergelines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Minimum bounding geometry

Creates geometries which enclose the features from an input layer. The features can be grouped by a field. The output layer will then contain one feature per group value with a geometry (MBB) that covers the geometries of the features with matching value.

The following enclosing geometry types are supported:

- bounding box (envelope)
- oriented rectangle
- circle
- convex hull

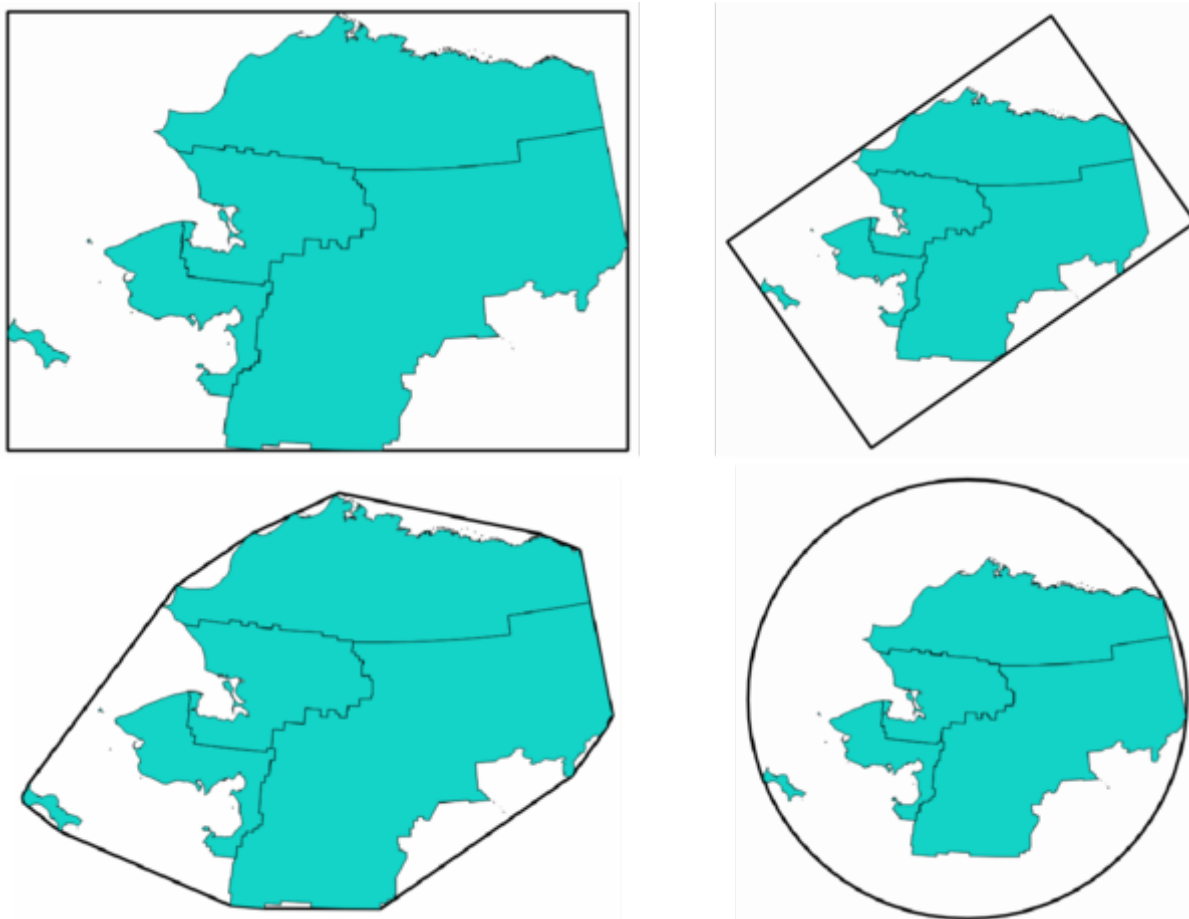


图 24.66: Clockwise from top left: envelope, oriented rectangle, circle, convex hull

参见:

Minimum enclosing circles

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Field Optional	FIELD	[tablefield: any]	Features can be grouped by a field. If set, this causes the output layer to contain one feature per grouped value with a minimal geometry covering only the features with matching values.
Geometry type	TYPE	[enumeration] Default: 0	Enclosing geometry types. One of: <ul style="list-style-type: none"> • 0 --- Envelope (Bounding Box) • 1 --- Minimum Oriented Rectangle • 2 --- Minimum Enclosing Circle • 3 --- Convex Hull

下页继续

表 24.107 – 续上页

Label	名称	Type	Description
Bounding geometry	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Bounding geometry	OUTPUT	[vector: polygon]	The output (bounding) polygon vector layer.

Python code

Algorithm ID: `qgis:minimumboundinggeometry`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Minimum enclosing circles

Calculates the minimum enclosing circles of the features in the input layer.

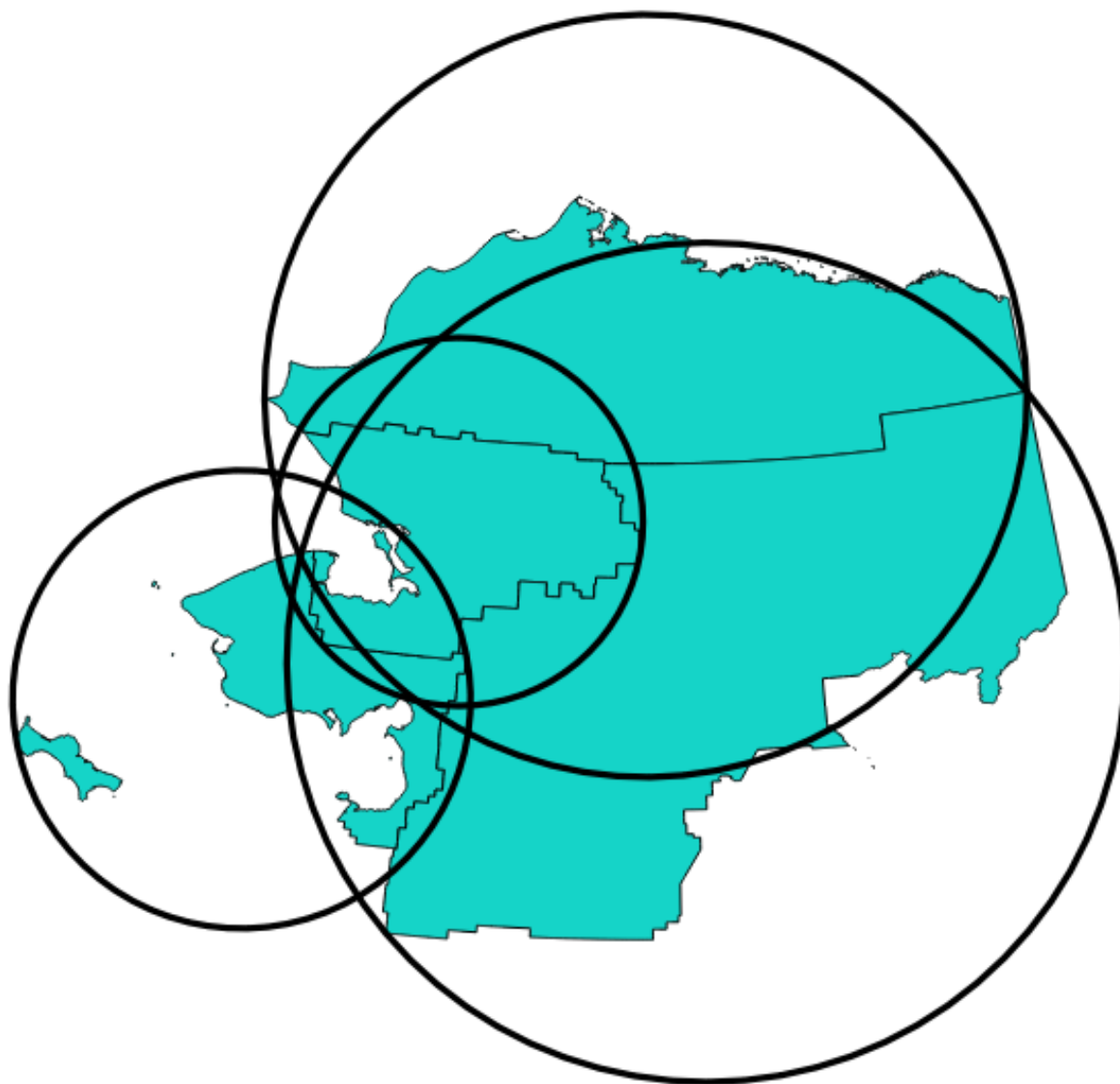


图 24.67: Enclosing circles for each feature

Allows *features in-place modification*

参见:

Minimum bounding geometry

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Number of segment in circles	SEGMENTS	[number] Default: 72	The number of segment used to approximate a circle. Minimum 8, maximum 100000.
Minimum enclosing circles	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Minimum enclosing circles	OUTPUT	[vector: polygon]	The output polygon vector layer.

Python code

Algorithm ID: native:minimumenclosingcircle

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Multi-ring buffer (constant distance)

Computes multi-ring (*donut*) buffer for the features of the input layer, using a fixed or dynamic distance and number of rings.

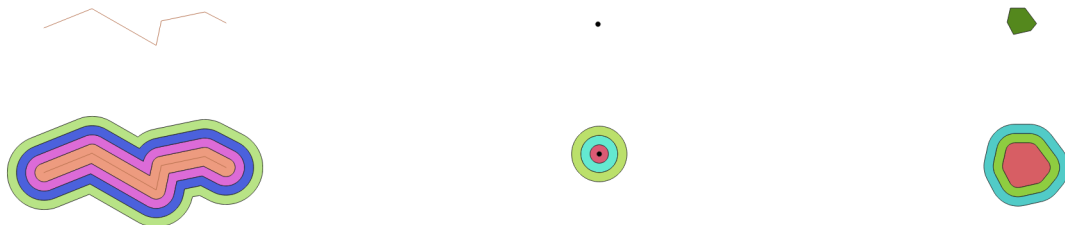




图 24.68: Multi-ring buffer for a line, point and polygon layer

Allows *features in-place modification*

参见:

Buffer, Variable distance buffer, Rectangles, ovals, diamonds, Single sided buffer

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Number of rings	RINGS	[number ] Default: 1	The number of rings. It can be a unique value (same number of rings for all the features) or it can be taken from features data (the number of rings depends on feature values).
Distance between rings	DISTANCE	[number ] Default: 1.0	Distance between the rings. It can be a unique value (same distance for all the features) or it can be taken from features data (the distance depends on feature values).
Multi-ring buffer (constant distance)	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Multi-ring buffer (constant distance)	OUTPUT	[vector: polygon]	The output polygon vector layer.

Python code

Algorithm ID: native:multiringconstantbuffer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Multipart to singleparts

Splits multipart features in the input layer into singlepart features.

The attributes of the output layer are the same as the original ones but divided into single features.



图 24.69: Left the multipart source layer and right the single part output result

Allows *features in-place modification*

Default menu: *Vector ► Geometry Tools*

参见:

Collect geometries, Promote to multipart

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Single parts	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Single parts	OUTPUT	[same as input]	The output vector layer.

Python code

Algorithm ID: `native:multiparttosingleparts`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Offset lines

Offsets lines by a specified distance. Positive distances will offset lines to the left, and negative distances will offset them to the right.

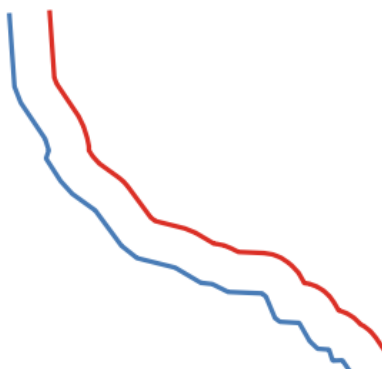



图 24.70: In blue the source layer, in red the offset one

Allows *features in-place modification*

参见:

Array of offset (parallel) lines, Translate

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Distance	DISTANCE	[number ] Default: 10.0	Offset distance. You can use the Data Defined button on the right to choose a field from which the radius will be calculated. This way you can have different radius for each feature (see <i>Variable distance buffer</i>).
Segments	SEGMENTS	[number] Default: 8	Controls the number of line segments to use to approximate a quarter circle when creating rounded offsets.
Join style	JOIN_STYLE	[enumeration] Default: 0	Specifies whether round, miter or beveled joins should be used when offsetting corners in a line. Options are: <ul style="list-style-type: none"> • 0 --- Round • 1 --- Miter • 2 --- Bevel
Miter limit	MITER_LIMIT	[number] Default: 2.0	Controls the maximum distance from the offset curve to use when creating a mitered join (only applicable for miter join styles). Minimum: 1.

下页继续

表 24.109 – 续上页

Label	名称	Type	Description
Offset	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output (offset) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Offset	OUTPUT	[vector: line]	Output (offset) line layer

Python code

Algorithm ID: native:offsetline

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Oriented minimum bounding box

Calculates the minimum area rotated rectangle for each feature in the input layer.

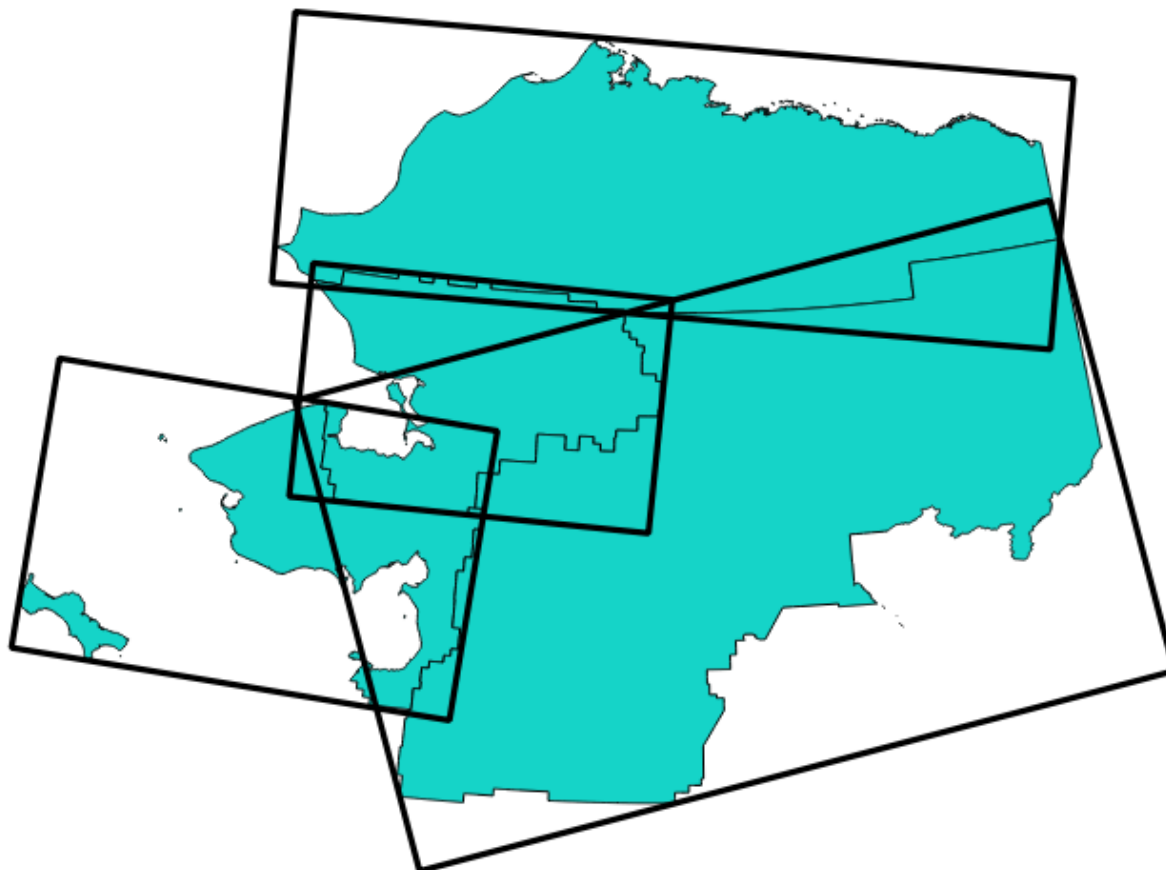


图 24.71: Oriented minimum bounding box

Allows *features in-place modification*

参见:

Minimum bounding geometry

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Bounding boxes	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Bounding boxes	OUTPUT	[vector: polygon]	The output polygon vector layer.

Python code

Algorithm ID: native:orientedminimumboundingbox

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Orthogonalize

Attempts to orthogonalize the geometries of the input line or polygon layer. This process shifts the vertices in the geometries to try to make every angle in the geometry either a right angle or a straight line.

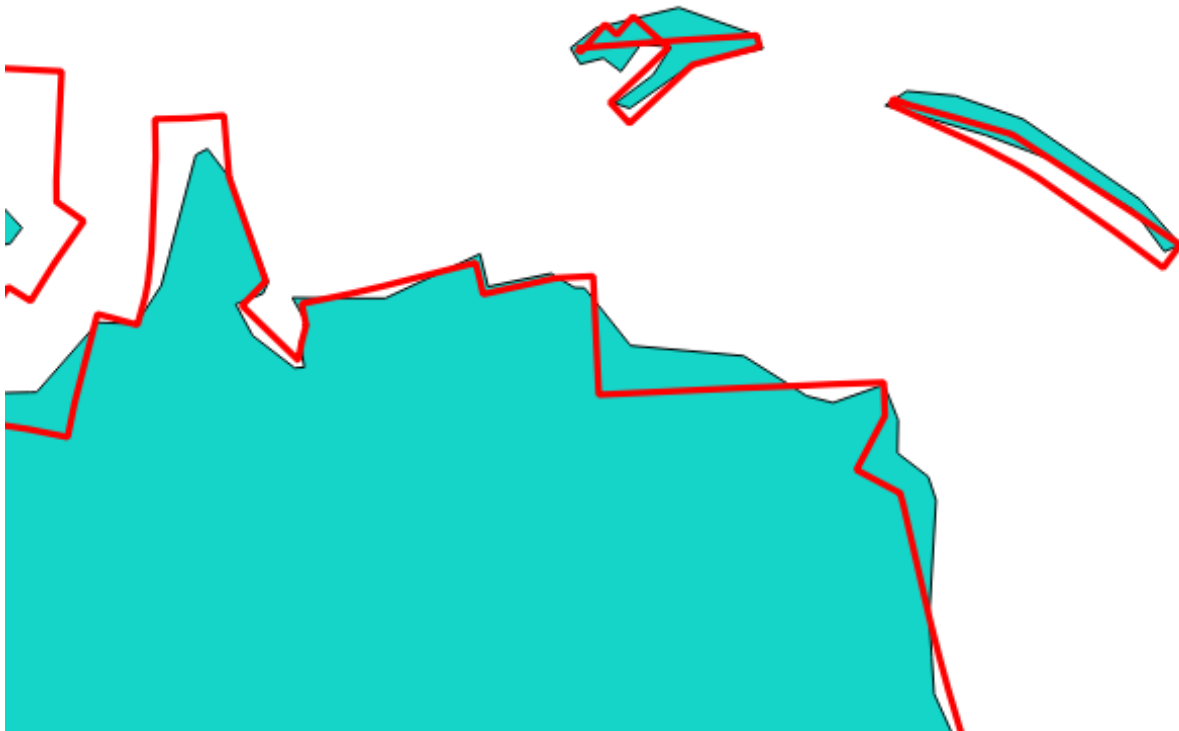


图 24.72: In blue the source layer and in the red orthogonalized result

Allows *features in-place modification*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Maximum angle tolerance (degrees)	ANGLE_TOLERANCE	[number] Default: 15	Specify the maximum deviation from a right angle or straight line a vertex can have for it to be adjusted. Smaller tolerances mean that only vertices which are already closer to right angles will be adjusted, and larger tolerances mean that vertices which deviate further from right angles will also be adjusted.
Maximum algorithm iterations	MAX_ITERATIONS	[number] Default: 1000	Setting a larger number for the maximum number of iterations will result in a more orthogonal geometry at the cost of extra processing time.
Orthogonalized	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Orthogonalized	OUTPUT	[same as input]	The output polygon vector layer with adjusted angles.

Python code

Algorithm ID: native:orthogonalize

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Point on Surface


For each feature of the input layer, returns a point that is guaranteed to lie on the surface of the feature geometry.

 Allows *features in-place modification*

参见:

Centroids

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Create point on surface for each part	ANGLE_TOLERANCE	[boolean ]	If checked, a point will be created for each part of the geometry.
Point	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output point vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Point	OUTPUT	[vector: point]	The output point vector layer.

Python code

Algorithm ID: native:pointonsurface

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Points along geometry

Creates points at regular intervals along line or polygon geometries. Created points will have new attributes added for the distance along the geometry and the angle of the line at the point.

An optional start and end offset can be specified, which controls how far from the start and end of the geometry the points should be created.

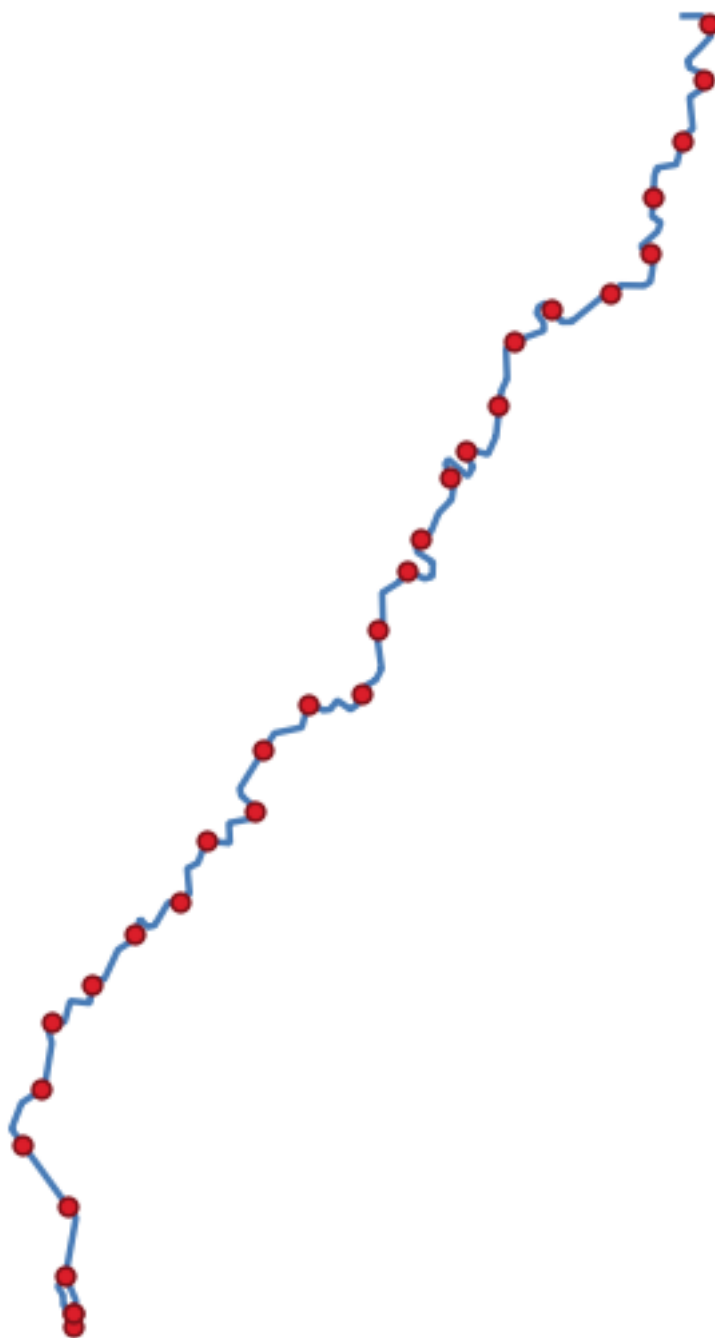





图 24.73: Points created along the source line layer

参见:

Interpolate point on line

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Distance	DISTANCE	[number ] Default: 1.0	Distance between two consecutive points along the line
Start offset	START_OFFSET	[number ] Default: 0.0	Distance from the beginning of the input line, representing the position of the first point.
End offset	END_OFFSET	[number ] Default: 0.0	Distance from the end of the input line, representing the position beyond which no point feature should be created.
Interpolated points	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Interpolated points	OUTPUT	[vector: point]	Point vector layer with features placed along lines or polygon boundaries of the input layer.

Python code

Algorithm ID: native:pointsalonglines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Points displacement

Given a distance of proximity, identifies nearby point features and radially distributes them over a circle whose center represents their barycenter. A convenient tool to scatter overlaid features.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Input point vector layer
Minimum distance to other points	PROXIMITY	[number] Default: 1.0	Distance below which point features are considered close. Close features are distributed altogether.
Displacement distance	DISTANCE	[number] Default: 1.0	Radius of the circle on which close features are placed
Horizontal distribution for two point case	HORIZONTAL	[boolean] Default: False	When only two points are identified as close, aligns them horizontally on the circle instead of vertically.
Displaced	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Displaced	OUTPUT	[vector: point]	Output point vector layer

Python code

Algorithm ID: qgis:pointdisplacement

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Pole of inaccessibility

Calculates the pole of inaccessibility for a polygon layer, which is the most distant internal point from the boundary of the surface.

This algorithm uses the 'polylabel' algorithm (Vladimir Agafonkin, 2016), which is an iterative approach guaranteed to find the true pole of inaccessibility within a specified tolerance. A more precise tolerance (lower value) requires more iterations and will take longer to calculate.

The distance from the calculated pole to the polygon boundary will be stored as a new attribute in the output layer.



图 24.74: Pole of inaccessibility

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Input vector layer
Tolerance	TOLERANCE	[number] Default: 1.0	Set the tolerance for the calculation
Point	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Point	OUTPUT	[vector: point]	The output point vector layer

Python code

Algorithm ID: native:poleofinaccessibility

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Polygonize

Creates a polygon layer whose features boundaries are generated from a line layer of **closed** features.

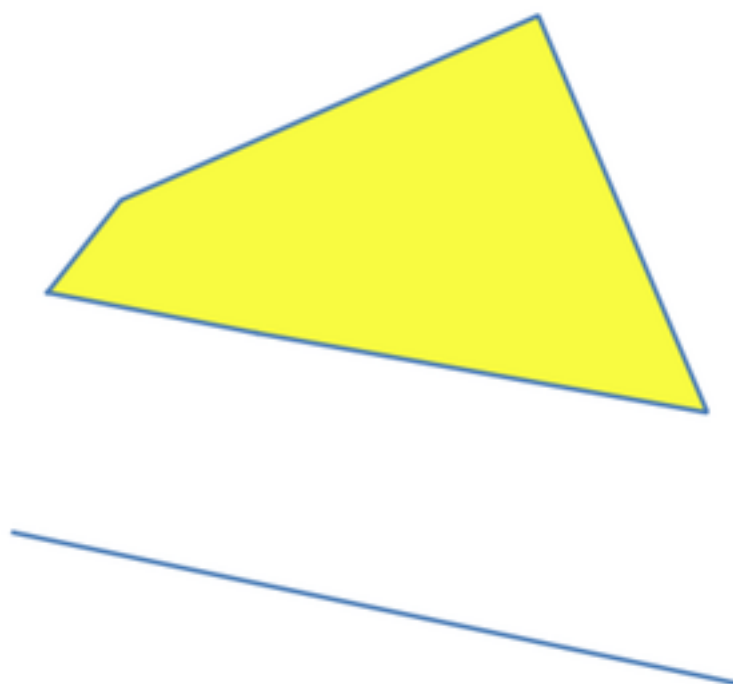


图 24.75: The yellow polygons generated from the closed lines

注解: The line layer must have closed shapes in order to be transformed into a polygon.

参见:

Polygons to lines, Lines to polygons, Convert geometry type

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Keep table structure of line layer Optional	KEEP_FIELDS	[boolean] Default: False	Check to keep the fields (only the table structure, not the values) of the input layer
Polygons from lines	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output polygon vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Polygons from lines	OUTPUT	[vector: polygon]	The output polygon vector layer from lines

Python code

Algorithm ID: native:polygonize

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Polygons to lines

Takes a polygon layer and creates a line layer, with lines representing the boundaries of the polygons in the input layer.

The attribute table of the output layer is the same as the one of the input layer.



图 24.76: Black lines as the result of the algorithm

Default menu: *Vector ► Geometry Tools*

参见:

Lines to polygons, Polygonize, Convert geometry type

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Input polygon vector layer
Lines	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output line vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Lines	OUTPUT	[vector: line]	The output line vector layer from polygons

Python code

Algorithm ID: native:polygonstolines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```



The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Project points (Cartesian)

Projects point geometries by a specified distance and bearing (azimuth).

Allows *features in-place modification*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Input point vector layer
Bearing (degrees from North)	BEARING	[number ] Default: 0.0	Clockwise angle starting from North, in degree (°) unit
Distance	DISTANCE	[number ] Default: 1.0	Distance to offset geometries, in layer units
Projected	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output point vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Projected	OUTPUT	[vector: point]	The output (projected) point vector layer

Python code

Algorithm ID: native:projectpointcartesian

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Promote to multipart

Takes a vector layer with singlepart geometries and generates a new one in which all geometries are multipart.

Input features which are already multipart features will remain unchanged.

This algorithm can be used to force geometries to multipart types in order to be compatible with data providers that require multipart features.

Allows *features in-place modification*

参见:

Aggregate, Collect geometries

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Multiparts	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output multipart vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Multiparts	OUTPUT	[same as input]	The output multipart vector layer

Python code

Algorithm ID: native:promotetomulti

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Rectangles, ovals, diamonds




Creates a buffer area with a rectangle, oval or diamond shape for each feature of the input point layer.

The shape parameters can be fixed for all features or dynamic using a field or an expression.



图 24.77: Different buffer shapes with dynamic parameters

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Input point vector layer
Buffer shape	SHAPE	[enumeration]	The shape to use. One of: <ul style="list-style-type: none"> • 0 --- Rectangles • 1 --- Ovals • 2 --- Diamonds
Width	WIDTH	[number ] Default: 1.0	Width of the buffer shape
Height	HEIGHT	[number ] Default: 1.0	Height of the buffer shape
Rotation Optional	ROTATION	[number ] Default: None	Rotation of the buffer shape
Number of segment	SEGMENTS	[number] Default: 36	Number of segments for a full circle (<i>Ovals</i> shape)

下页继续

表 24.114 – 续上页

Label	名称	Type	Description
Output	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output	OUTPUT	[vector: polygon]	The output vector layer (with the buffer shapes)

Python code

Algorithm ID: native:rectanglesovalsdiamonds

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Remove duplicate vertices

Removes duplicate vertices from features, wherever removing the vertices does not result in a degenerate geometry.

The tolerance parameter specifies the tolerance for coordinates when determining whether vertices are identical.

By default, Z values are not considered when detecting duplicate vertices. E.g. two vertices with the same X and Y coordinate but different Z values will still be considered duplicate and one will be removed. If the *Use Z Value* parameter is true, then the Z values are also tested and vertices with the same X and Y but different Z will be maintained.



注解: Duplicate vertices are not tested between different parts of a multipart geometry, e.g. a multipoint geometry with overlapping points will not be changed by this method.

 Allows *features in-place modification*

参见:

Extract vertices, Extract specific vertices, Delete duplicate geometries

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Tolerance	TOLERANCE	[number  Default: 0.000001]	Vertices closer than the specified distance are considered duplicates
Use Z value	USE_Z_VALUE	[boolean  Default: False]	If the <i>Use Z Value</i> parameter is true, then the Z values are also tested and vertices with the same X and Y but different Z will be maintained.
Cleaned	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Cleaned	OUTPUT	[same as input]	The output vector layer (without duplicate vertices)

Python code

Algorithm ID: native:removeduplicatevertices

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Remove null geometries

Removes any features which do not have a geometry from a vector layer. All other features will be copied unchanged.

The features with null geometries can be saved to a separate layer.

If *Also remove empty geometries* is checked, the algorithm removes features whose geometries have no coordinates, i.e., geometries that are empty. In that case, also the null output will reflect this option, containing both null and empty geometries.

参见:

Delete duplicate geometries

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer (with non-NULL geometries)
Also remove empty geometries	REMOVE_EMPTY	[boolean]	
Non null geometries	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer for the non-NULL (and non-empty) geometries. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Null geometries	NULL_OUTPUT	[same as input] Default: [Skip output]	Specify the output vector layer for the NULL (and empty) geometries. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Null geometries	NULL_OUTPUT	[same as input]	Output vector layer (for NULL and, if chosen, empty geometries)
Non null geometries	OUTPUT	[same as input]	The output vector layer (without NULL and, if chosen, empty geometries)

Python code

Algorithm ID: native:removenullgeometries

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Reverse line direction

Inverts the direction of a line layer.

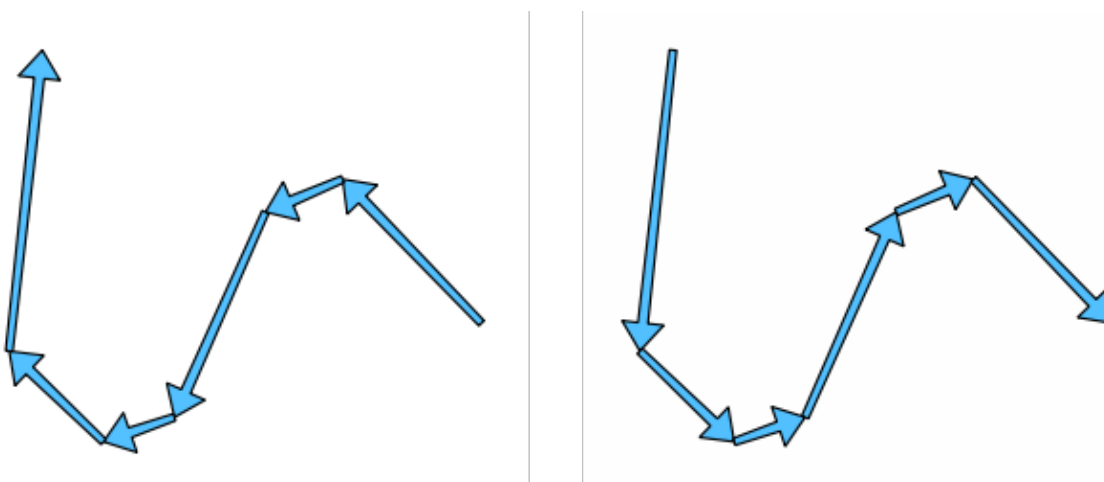


图 24.78: Before and after the direction inversion

Allows *features in-place modification*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Reversed	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output line vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Reversed	OUTPUT	[vector: line]	The output line vector layer (with reversed lines)

Python code

Algorithm ID: native:reverselinedirection

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Rotate


Rotates feature geometries by the specified angle clockwise. The rotation occurs around each feature's centroid, or optionally around a unique preset point.

 Allows *features in-place modification*

参见:

Translate, Swap X and Y coordinates

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Rotation (degrees clockwise)	ANGLE	[number ] Default: 0.0	Angle of the rotation in degrees
Rotation anchor point (x, y) Optional	ANCHOR	[point] Default: None	X,Y coordinates of the point to rotate the features around. If not set the rotation occurs around each feature's centroid.
Rotated	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer (with rotated geometries). One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Rotated	OUTPUT	[same as input]	The output vector layer with rotated geometries

Python code

Algorithm ID: native:rotatefeatures

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Segmentize by maximum angle


Segmentizes a geometry by converting curved sections to linear sections.

The segmentization is performed by specifying the maximum allowed radius angle between vertices on the straightened geometry (e.g the angle of the arc created from the original arc center to consecutive output vertices on the linearized geometry). Non-curved geometries will be retained without change.

参见:

Segmentize by maximum distance, Simplify, Smooth

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Maximum angle between vertices (degrees)	ANGLE	[number ] Default: 5.0	Maximum allowed radius angle between vertices on the straightened geometry
Segmentized	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer (with segmentized geometries). One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Segmentized	OUTPUT	[same as input]	The output vector layer with segmentized geometries

Python code

Algorithm ID: native:segmentizebymaxangle

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Segmentize by maximum distance


Segmentizes a geometry by converting curved sections to linear sections.

The segmentization is performed by specifying the maximum allowed offset distance between the original curve and the segmentized representation. Non-curved geometries will be retained without change.

参见:

Segmentize by maximum angle, Simplify, Smooth

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Maximum offset distance	DISTANCE	[number ] Default: 1.0	Maximum allowed offset distance between the original curve and the segmentized representation, in the layer units.
Segmentized	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer (with segmentized geometries). One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Segmentized	OUTPUT	[same as input]	The output vector layer with segmentized geometries

Python code

Algorithm ID: native:segmentizebymaxdistance


```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Set M value

Sets the M value for geometries in a layer.


If M values already exist in the layer, they will be overwritten with the new value. If no M values exist, the geometry will be upgraded to include M values and the specified value used as the initial M value for all geometries.

小技巧: Use the  Identify Features button to check the added M value: the results are available in the *Identify Results* dialog.

参见:

Set M value from raster, Set Z value, Drop M/Z values

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
M Value	M_VALUE	[number  Default: 0.0	M value to assign to the feature geometries
M Added	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
M Added	OUTPUT	[same as input]	The output vector layer (with M values assigned to the geometries)

Python code

Algorithm ID: native:setmvalue

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Set M value from raster



Uses values sampled from a band within a raster layer to set the M value for every overlapping vertex in the feature geometry. The raster values can optionally be scaled by a preset amount.

If M values already exist in the layer, they will be overwritten with the new value. If no M values exist, the geometry will be upgraded to include M values.

参见:

Drape (set Z value from raster), Set M value

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Raster layer	RASTER	[raster]	Raster layer with M values
Band number	BAND	[raster band] Default: 1	The raster band from which the M values are taken
Value for no-data or non-intersecting vertices	NODATA	[number ] Default: 0.0	Value to use in case the vertex does not intersect (a valid pixel of) the raster
Scale factor	SCALE	[number ] Default: 1.0	Scaling value: the band values are multiplied by this value.
Updated	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer (with updated M values). One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Updated	OUTPUT	[same as input]	The output vector layer (with updated M values)

Python code

Algorithm ID: native:setmfromraster


```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Set Z value

Sets the Z value for geometries in a layer.


If Z values already exist in the layer, they will be overwritten with the new value. If no Z values exist, the geometry will be upgraded to include Z values and the specified value used as the initial Z value for all geometries.

小技巧: Use the  button to check the added Z value: the results are available in the *Identify Results* dialog.

参见:

Drape (set Z value from raster), Set M value, Drop M/Z values

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Z Value	Z_VALUE	[number  Default: 0.0	Z value to assign to the feature geometries
Z Added	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Z Added	OUTPUT	[same as input]	The output vector layer (with Z values assigned)

Python code

Algorithm ID: native:setzvalue

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Simplify

Simplifies the geometries in a line or polygon layer. It creates a new layer with the same features as the ones in the input layer, but with geometries containing a lower number of vertices.

The algorithm gives a choice of simplification methods, including distance based (the "Douglas-Peucker" algorithm), area based ("Visvalingam" algorithm) and snapping geometries to grid.



图 24.79: Clockwise from top left: source layer and increasing simplification tolerances


Allows *features in-place modification*

Default menu: *Vector* ► *Geometry Tools*

参见:

Smooth, Densefy by count, Densefy by interval

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Simplification method	METHOD	[enumeration] Default: 0	Simplification method. One of: <ul style="list-style-type: none"> • 0 --- Distance (Douglas-Peucker) • 1 --- Snap to grid • 2 --- Area (Visvalingam)
Tolerance	TOLERANCE	[number ] Default: 1.0	Threshold tolerance (in units of the layer): if the distance between two nodes is smaller than the tolerance value, the segment will be simplified and vertices will be removed.
Simplified	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output (simplified) vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Simplified	OUTPUT	[same as input]	The output (simplified) vector layer

Python code

Algorithm ID: native:simplifygeometries

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Single sided buffer

Computes a buffer on lines by a specified distance on one side of the line only.

Buffer always results in a polygon layer.



图 24.80: Left versus right side buffer on the same vector line layer

参见:

Buffer

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Distance	DISTANCE	[number] Default: 10.0	Buffer distance.
Side	SIDE	[enumeration]	Which side to create the buffer on. One of: <ul style="list-style-type: none"> • 0 -- Left • 1 -- Right
Segments	SEGMENTS	[number] Default: 8	Controls the number of line segments to use to approximate a quarter circle when creating rounded offsets.
Join style	JOIN_STYLE	[enumeration]	Specifies whether round, miter or beveled joins should be used when offsetting corners in a line. Options are: <ul style="list-style-type: none"> • 0 --- Round • 1 --- Miter • 2 --- Bevel
Miter limit	MITER_LIMIT	[number] Default: 2.0	Controls the maximum distance from the offset curve to use when creating a mitered join (only applicable for miter join styles). Minimum: 1.0
Buffer	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output (buffer) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Buffer	OUTPUT	[vector: polygon]	Output (buffer) polygon layer

Python code

Algorithm ID: native:singlesidedbuffer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Smooth

Smooths the geometries in a line or polygon layer by adding more **vertices and corners** to the feature geometries.

The iterations parameter dictates how many smoothing iterations will be applied to each geometry. A higher number of iterations results in smoother geometries with the cost of greater number of nodes in the geometries.



图 24.81: Increasing number of iterations causes smoother geometries

The offset parameter controls how "tightly" the smoothed geometries follow the original geometries. Smaller values results in a tighter fit, and larger values will create a looser fit.

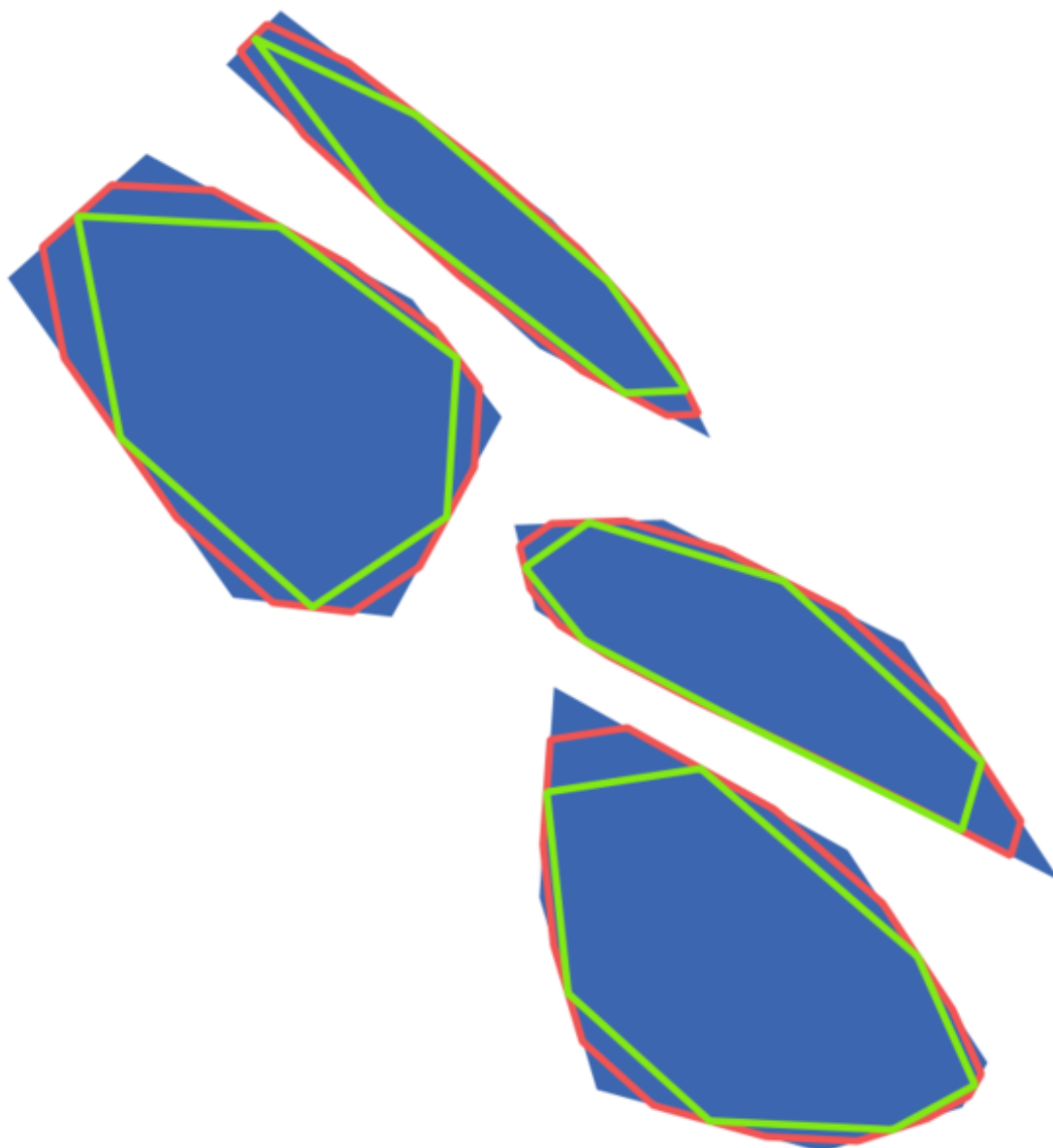


图 24.82: Blue: the input layer. Offset 0.25 gives the red line, while offset 0.50 gives the green line.




The maximum angle parameter can be used to prevent smoothing of nodes with large angles. Any node where the angle of the segments to either side is larger than this will not be smoothed. For example, setting the maximum angle to 90 degrees or lower would preserve right angles in the geometry.

Allows *features in-place modification*

参见:

Simplify, Densify by count, Densify by interval

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Input line or polygon vector layer
Iterations	ITERATIONS	[number ] Default: 1	Increasing the number of iterations will give smoother geometries (and more vertices).
Offset	OFFSET	[number ] Default: 0.25	Increasing values will <i>move</i> the smoothed lines / boundaries further away from the input lines / boundaries.
Maximum node angle to smooth	MAX_ANGLE	[number ] Default: 180.0	Every node below this value will be smoothed
Smoothed	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output (smoothed) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Smoothed	OUTPUT	[same as input]	Output (smoothed) vector layer

Python code

Algorithm ID: native:smoothgeometry

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Snap geometries to layer

Snaps the geometries in a layer either to the geometries from another layer, or to geometries within the same layer.

Matching is done based on a tolerance distance, and vertices will be inserted or removed as required to make the geometries match the reference geometries.

参见:

[Snap points to grid](#)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Reference layer	REFERENCE_LAYER	[vector: any]	Vector layer to snap to
Tolerance	TOLERANCE	[number] Default: 10.0	Control how close input vertices need to be to the reference layer geometries before they are snapped.
Behavior	BEHAVIOR	[enumeration] Default: 0	<p>Snapping can be done to an existing node or a segment (its closest point to the vertex to move). Available snapping options:</p> <ul style="list-style-type: none"> • 0 --- Prefer aligning nodes, insert extra vertices where required Prefer to snap to nodes, even when a segment may be closer than a node. New nodes will be inserted to make geometries follow each other exactly when inside allowable tolerance. • 1 --- Prefer closest point, insert extra vertices where required Snap to closest point, regardless of it is a node or a segment. New nodes will be inserted to make geometries follow each other exactly when inside allowable tolerance. • 2 --- Prefer aligning nodes, don't insert new vertices Prefer to snap to nodes, even when a segment may be closer than a node. No new nodes will be inserted. • 3 --- Prefer closest point, don't insert new vertices Snap to closest point, regardless of it is a node or a segment. No new nodes will be inserted. • 4 --- Move end points only, prefer aligning nodes Only snap start/end points of lines (point features will also be snapped, polygon features will not be modified), prefer to snap to nodes. • 5 --- Move end points only, prefer closest point Only snap start/end points of lines (point features will also be snapped, polygon features will not be modified), snap to closest point • 6 --- Snap end points to end points only Only snap the start/end points of lines to other start/end points of lines • 7 --- Snap to anchor nodes (single layer only)

下页继续

表 24.121 - 续上页

Label	名称	Type	Description
Snapped geometry	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output (snapped) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Snapped geometry	OUTPUT	[same as input]	Output (snapped) vector layer

Python code

Algorithm ID: native:snappedgeometries

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Snap points to grid

Modifies the coordinates of geometries in a vector layer, so that all points or vertices are snapped to the closest point of a grid.

If the snapped geometry cannot be calculated (or is totally collapsed) the feature's geometry will be cleared.

Snapping can be performed on the X, Y, Z or M axis. A grid spacing of 0 for any axis will disable snapping for that axis.





注解: Snapping to grid may generate an invalid geometry in some corner cases.

 Allows *features in-place modification*

参见:

Snap geometries to layer

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
X Grid Spacing	HSPACING	[number ] Default: 1.0	Grid spacing on the X axis
Y Grid Spacing	VSPACING	[number ] Default: 1.0	Grid spacing on the Y axis
Z Grid Spacing	ZSPACING	[number ] Default: 0.0	Grid spacing on the Z axis
M Grid Spacing	MSPACING	[number ] Default: 0.0	Grid spacing on the M axis
Snapped	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output (snapped) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Snapped	OUTPUT	[same as input]	Output (snapped) vector layer

Python code

Algorithm ID: native:snappointstogrid


```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Split lines by maximum length

Takes a line (or curve) layer and splits each feature into multiple parts, where each part is of a specified maximum length. Z and M values at the start and end of the new line substrings are linearly interpolated from existing values.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	The input line vector layer
Maximum line length	LENGTH	[number ] Default: 10.0	The maximum length of a line in the output.
Split	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output line vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Split	OUTPUT	[vector: line]	The new line vector layer - the length of the feature geometries is less than or equal to the length specified in the LENGTH parameter.

Python code

Algorithm ID: native:splitlinesbylength

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Subdivide

Subdivides the geometry. The returned geometry will be a collection containing subdivided parts from the original geometry, where no part has more than the specified maximum number of nodes.

This is useful for dividing a complex geometry into less complex parts, easier to spatially index and faster to perform spatial operations. Curved geometries will be segmented before subdivision.

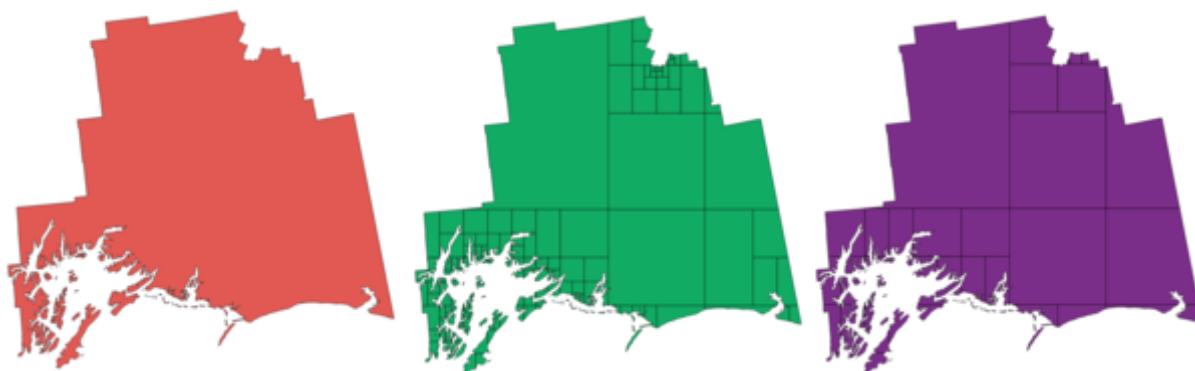


图 24.83: Left the input layer, middle maximum nodes value is 100 and right maximum value is 200


注解: Subdividing a geometry can generate geometry parts that may not be valid and may contain self-intersections.

Allows *features in-place modification*

参见:

Explode lines, Line substring

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer
Maximum nodes in parts	MAX_NODES	[number ] Default: 256	Maximum number of vertices each new geometry part is allowed to have. Fewer <i>sub-parts</i> for higher values.
Subdivided	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output (subdivided) vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Subdivided	OUTPUT	[same as input]	Output vector layer

Python code

Algorithm ID: native:subdivide

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Swap X and Y coordinates

Switches the X and Y coordinate values in input geometries.

It can be used to repair geometries which have accidentally had their latitude and longitude values reversed.

Allows *features in-place modification*

参见:

Translate, Rotate

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer
Swapped	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Swapped	OUTPUT	[same as input]	Output (swapped) vector layer

Python code

Algorithm ID: native:swapxy

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Tapered buffers

Creates tapered buffer along line geometries, using a specified start and end buffer diameter.

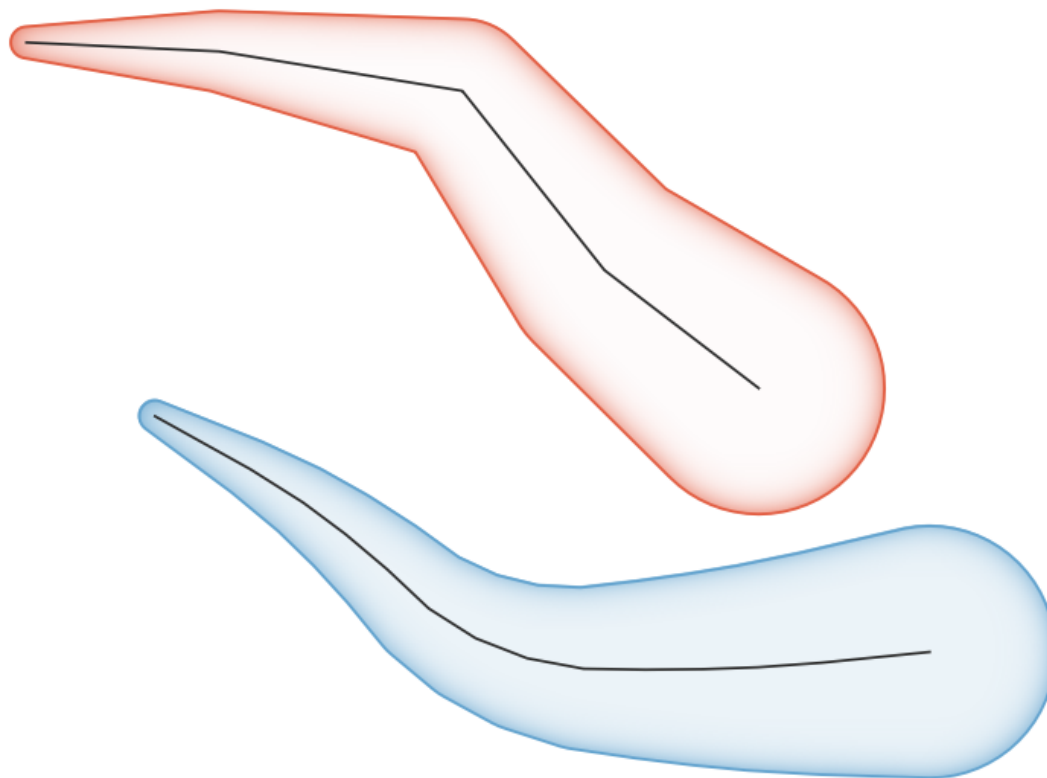





图 24.84: Tapered buffer example

参见:

Variable width buffer (by M value), Buffer, Create wedge buffers

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Start width	START_WIDTH	[number  Default: 0.0	Represents the radius of the buffer applied at the start point of the line feature
End width	END_WIDTH	[number  Default: 0.0	Represents the radius of the buffer applied at the end point of the line feature.
Segments	SEGMENTS	[number  Default: 16	Controls the number of line segments to use to approximate a quarter circle when creating rounded offsets.

下页继续

表 24.123 - 续上页

Label	名称	Type	Description
Buffered	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output (buffer) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Buffered	OUTPUT	[vector: polygon]	Output (buffer) polygon layer

Python code

Algorithm ID: native:taperedbuffer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Tessellate

Tessellates a polygon geometry layer, dividing the geometries into triangular components.

The output layer consists of multipolygon geometries for each input feature, with each multipolygon consisting of multiple triangle component polygons.

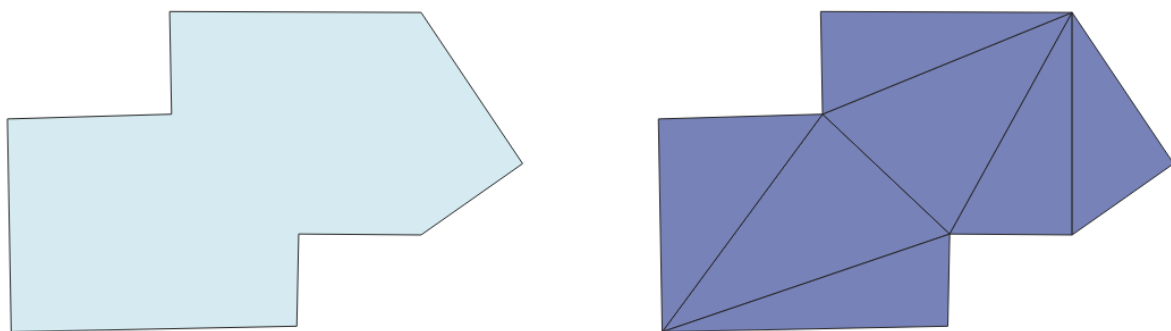


图 24.85: Tessellated polygon (right)

Allows *features in-place modification*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: polygon]	Input polygon vector layer
Tesselated	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Tesselated	OUTPUT	[vector: polygon]	Output multipolygonZ layer

Python code

Algorithm ID: 3d:tessellate

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Transect

Creates transects on vertices for (multi)linestring.

A transect is a line oriented from an angle (by default perpendicular) to the input polylines (at vertices).

Field(s) from feature(s) are returned in the transect with these new fields:

- TR_FID: ID of the original feature
- TR_ID: ID of the transect. Each transect have an unique ID
- TR_SEGMENT: ID of the segment of the linestring
- TR_ANGLE: Angle in degrees from the original line at the vertex
- TR_LENGTH: Total length of the transect returned
- TR_ORIENT: Side of the transect (only on the left or right of the line, or both side)

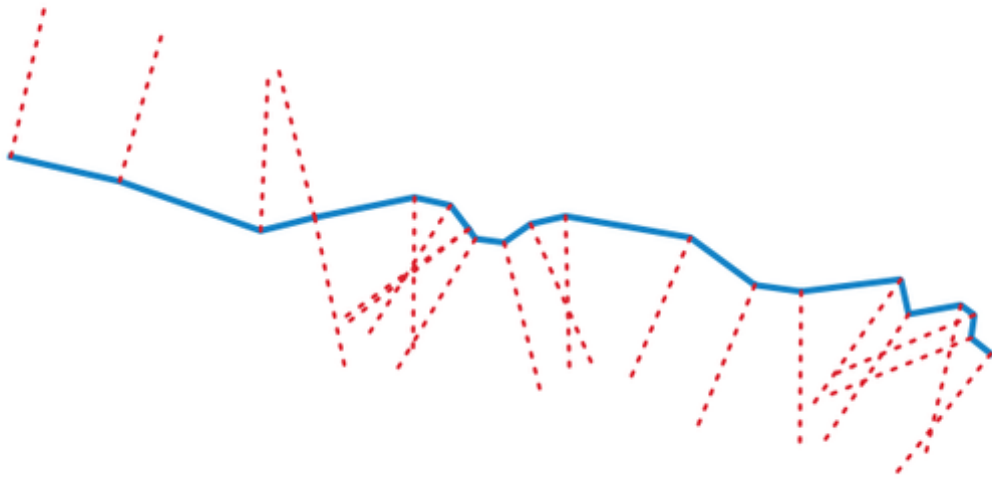




图 24.86: Dashed red lines represent the transect of the input line layer

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Length of the transect	LENGTH	[number ] Default: 5.0	Length in map unit of the transect
Angle in degrees from the original line at the vertices	ANGLE	[number ] Default: 90.0	Change the angle of the transect
Side to create the transect	SIDE	[enumeration]	Choose the side of the transect. Available options are: <ul style="list-style-type: none"> • 0 --- Left • 1 --- Right • 2 --- Both
Transect	OUTPUT	[vector: line] Default: [Create temporary layer]	Specify the output line layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Transect	OUTPUT	[vector: line]	Output line layer

Python code

Algorithm ID: native:transect

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Translate

Moves the geometries within a layer, by offsetting with a predefined X and Y displacement.

Z and M values present in the geometry can also be translated.

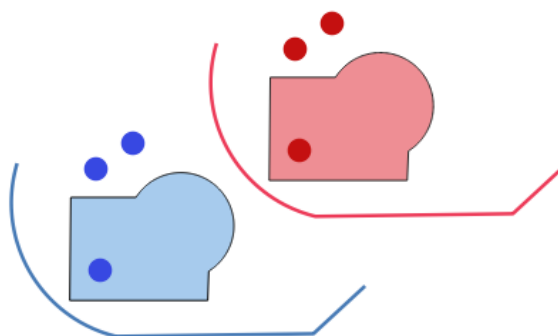




图 24.87: Dashed lines represent the translated geometry of the input layer

Allows *features in-place modification*

参见:



Array of translated features, Offset lines, Rotate, Swap X and Y coordinates

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Offset distance (x-axis)	DELTA_X	[number ] Default: 0.0	Displacement to apply on the X axis
Offset distance (y-axis)	DELTA_Y	[number ] Default: 0.0	Displacement to apply on the Y axis

下页继续

表 24.125 - 续上页

Label	名称	Type	Description
Offset distance (z-axis)	DELTA_Z	[number ] Default: 0.0	Displacement to apply on the Z axis
Offset distance (m values)	DELTA_M	[number ] Default: 0.0	Displacement to apply on the M axis
Translated	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Translated	OUTPUT	[same as input]	Output vector layer

Python code

Algorithm ID: native:translategeometry

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Variable distance buffer

Computes a buffer area for all the features in an input layer.

The size of the buffer for a given feature is defined by an attribute, so it allows different features to have different buffer sizes.

注解: This algorithm is only available from the *Graphical modeler*.

参见:

Buffer

Parameters

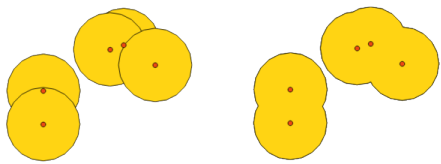
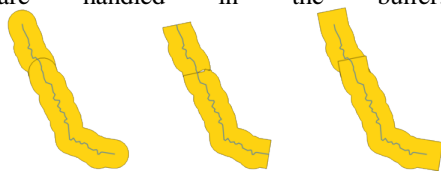
Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Distance field	DISTANCE	[tablefield: numeric]	Attribute for the distance radius of the buffer
Segments	SEGMENTS	[number] Default: 5	Controls the number of line segments to use to approximate a quarter circle when creating rounded offsets.
Dissolve result	DISSOLVE	[boolean] Default: <i>False</i>	Choose to dissolve the final buffer, resulting in a single feature covering all input features. 
End cap style	END_CAP_STYLE	[enumeration]	Controls how line endings are handled in the buffer. 
Join style	JOIN_STYLE	[enumeration]	Specifies whether round, miter or beveled joins should be used when offsetting corners in a line.
Miter limit	MITER_LIMIT	[number] Default: 2.0	Only applicable for mitered join styles, and controls the maximum distance from the offset curve to use when creating a mitered join.

图 24.88: Normal and dissolved buffer

图 24.89: Round, flat and square cap styles

Outputs

Label	名称	Type	Description
Buffer	OUTPUT	[vector: polygon]	Buffer polygon vector layer.

Python code

Algorithm ID: qgis:variabledistancebuffer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Variable width buffer (by M value)

Creates variable width buffers along lines, using the M value of the line geometries as the diameter of the buffer at each vertex.

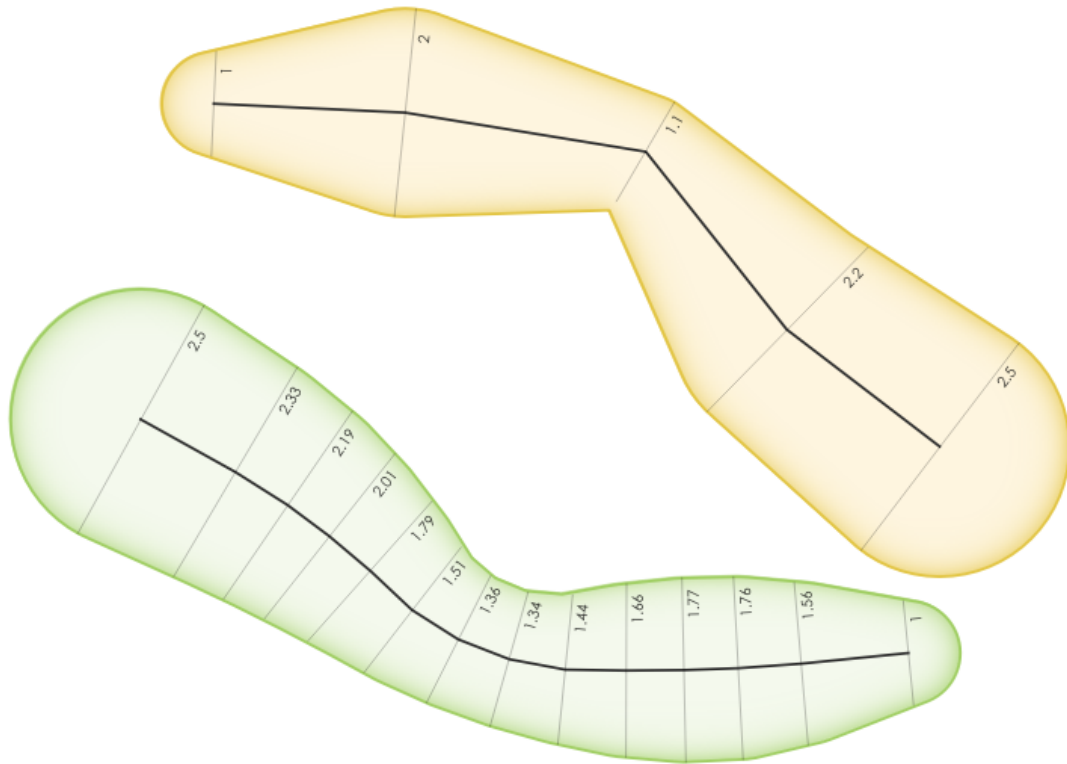



图 24.90: Variable buffer example

参见:

Tapered buffers, Buffer, Set M value

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line vector layer
Segments	SEGMENTS	[number ] Default: 16	Number of the buffer segments per quarter circle. It can be a unique value (same value for all the features), or it can be taken from features data (the value can depend on feature attributes).
Buffered	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output (buffer) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Buffered	OUTPUT	[vector: polygon]	Variable buffer polygon layer

Python code

Algorithm ID: native:bufferbym

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Voronoi polygons

Takes a point layer and generates a polygon layer containing the Voronoi polygons (known also as Thiessen polygons) corresponding to those input points.

Any location within a Voronoi polygon is closer to the associated point than to any other point.

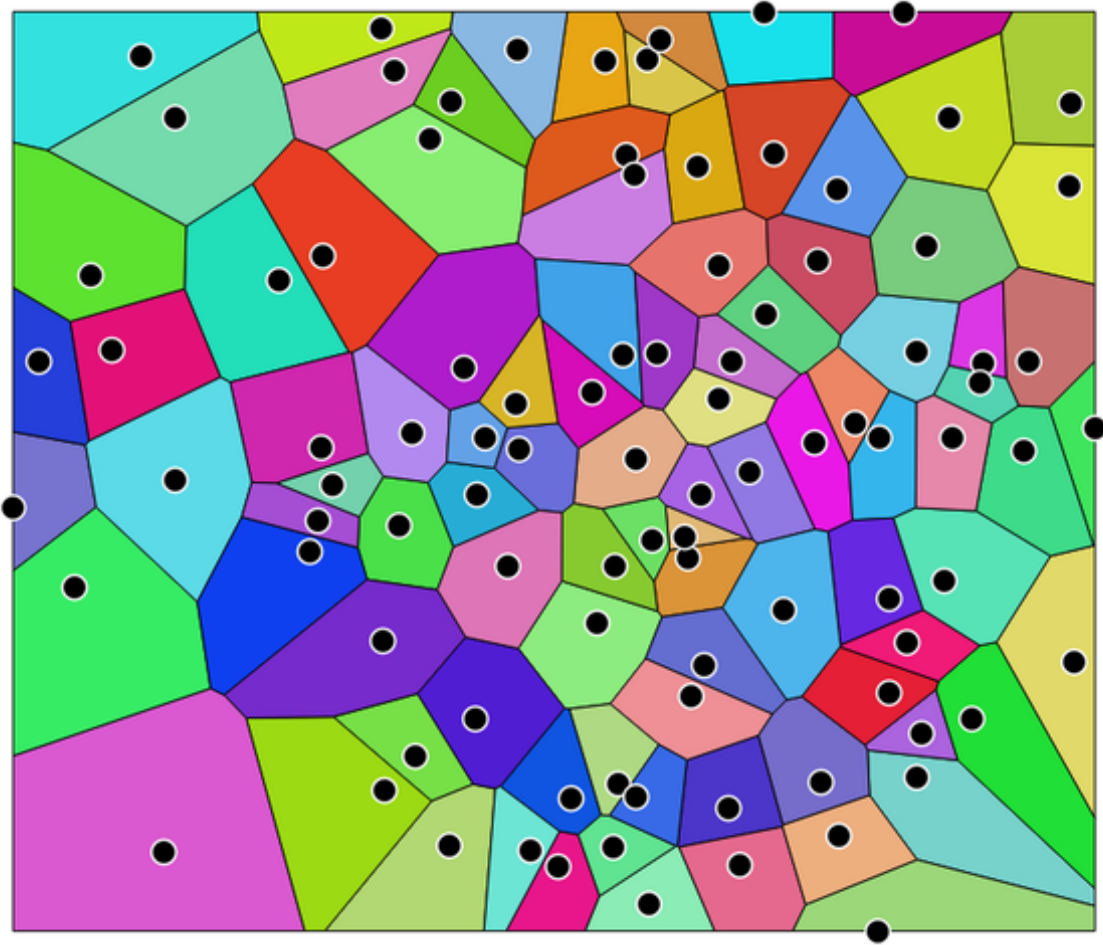


图 24.91: Voronoi polygons

Default menu: *Vector* ► *Geometry Tools*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	Input point vector layer
Buffer region (% of extent)	BUFFER	[number] Default: 0.0	The extent of the output layer will be this much bigger than the extent of the input layer
Voronoi polygons	OUTPUT	[vector: polygon] Default: [Create temporary layer]	Specify the output layer (with the Voronoi polygons). One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Voronoi polygons	OUTPUT	[vector: polygon]	Voronoi polygons of the input point vector layer

Python code

Algorithm ID: qgis:voronoipolygons

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.17 Vector overlay

Clip

Clips a vector layer using the features of an additional polygon layer.

Only the parts of the features in the input layer that fall within the polygons of the overlay layer will be added to the resulting layer.

警告: Feature modification

The attributes of the features are **not modified**, although properties such as area or length of the features will be modified by the clipping operation. If such properties are stored as attributes, those attributes will have to be manually updated.

This algorithm uses spatial indexes on the providers, prepared geometries and apply a clipping operation if the geometry isn't wholly contained by the mask geometry.

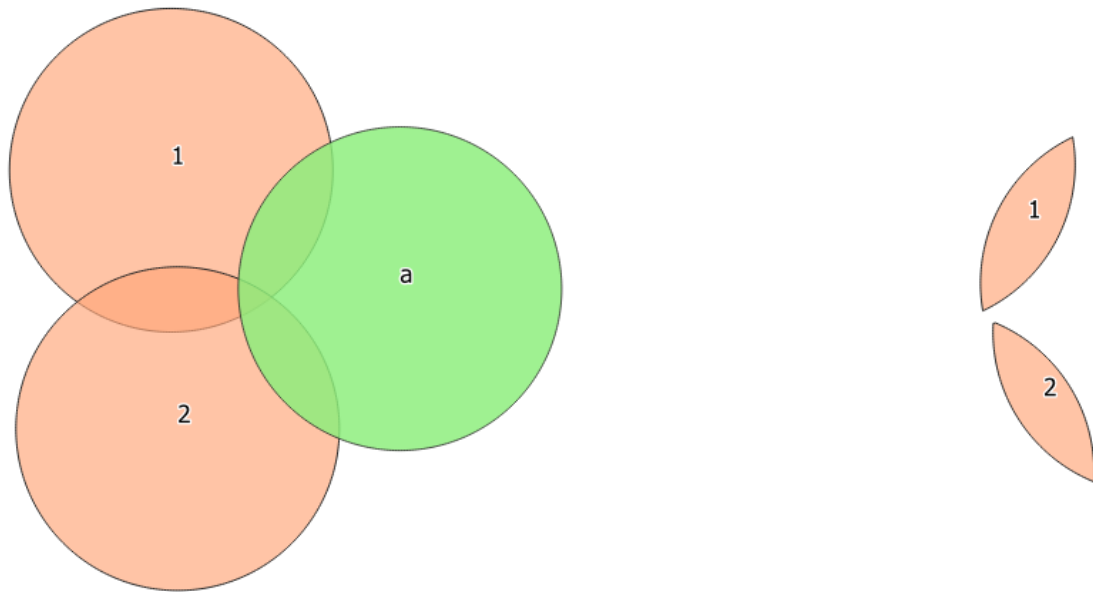


图 24.92: Clipping operation between a two-features input layer and a single feature overlay layer (left) - resulting features are moved for clarity (right)

Allows *features in-place modification*

Default menu: *Vector ► Geoprocessing Tools*

参见:

Intersection, Difference

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Layer containing the features to be clipped
Overlay layer	OVERLAY	[vector: polygon]	Layer containing the clipping features
Clipped	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the layer to contain the features from the input layer that are inside the overlay (clipping) layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Clipped	OUTPUT	[same as input]	Layer containing features from the input layer split by the overlay layer.

Python code

Algorithm ID: qgis:clip

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Difference

Extracts features from the input layer that don't fall within the boundaries of the overlay layer.

Input layer features that partially overlap the overlay layer feature(s) are split along the boundary of those feature(s) and only the portions outside the overlay layer features are retained.

Attributes are not modified (see *warning*).

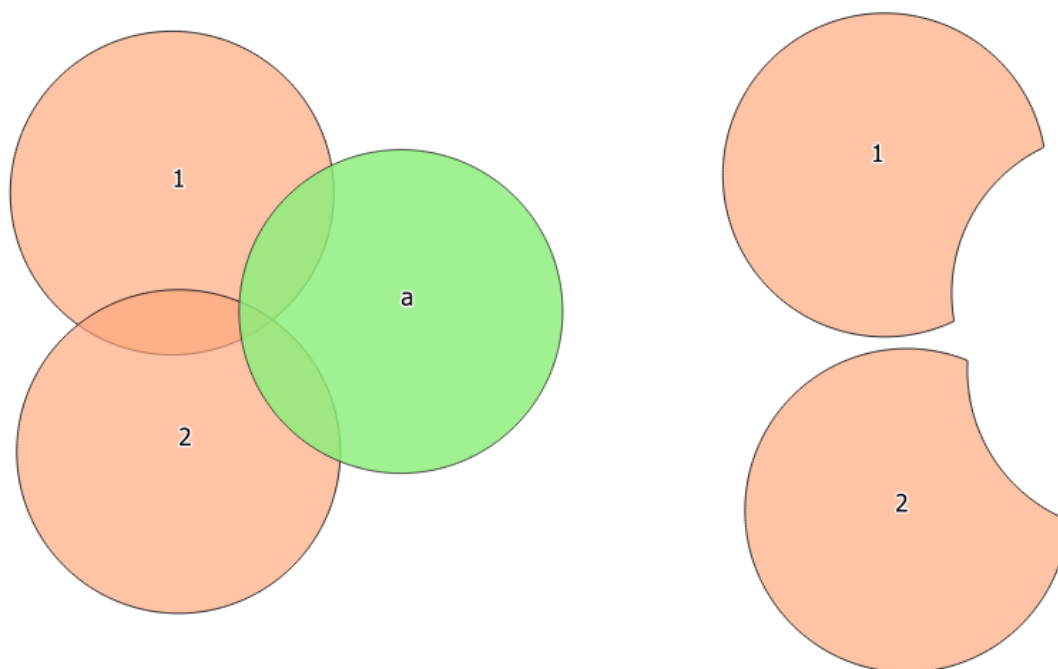


图 24.93: Difference operation between a two-features input layer and a single feature overlay layer (left) - resulting features are moved for clarity (right)

Allows *features in-place modification*

Default menu: *Vector ► Geoprocessing Tools*

参见:

Symmetrical difference, Clip

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Layer to extract (parts of) features from.
Overlay layer	OVERLAY	[vector: any]	Layer containing the geometries that will be subtracted from the input layer geometries. It is expected to have at least as many dimensions (point: 0D, line: 1D, polygon: 2D, volume: 3D) as the input layer geometries.
Difference	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the layer to contain the (parts of) features from the input layer that are not inside the overlay layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Difference	OUTPUT	[same as input]	Layer containing (parts of) features from the input layer not overlapping the overlay layer.

Python code

Algorithm ID: qgis:difference

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extract/clip by extent

Creates a new vector layer that only contains features which fall within a specified extent.

Any features which intersect the extent will be included.

参见:

Clip

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Layer to extract (parts of) features from.
Extent (xmin, xmax, ymin, ymax)	EXTENT	[extent]	Extent for clipping.
Clip features to extent	CLIP	[boolean] Default: False	If checked, output geometries will be automatically converted to multi geometries to ensure uniform output types. Moreover the geometries will be clipped to the extent chosen instead of taking the whole geometry as output.
Extracted	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the layer to contain the features from the input layer that are inside the clip extent. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Extracted	OUTPUT	[same as input]	Layer containing the clipped features.

Python code

Algorithm ID: qgis:extractbyextent

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Intersection

Extracts the portions of features from the input layer that overlap features in the overlay layer.

Features in the intersection layer are assigned the attributes of the overlapping features from both the input and overlay layers.

Attributes are not modified (see *warning*).

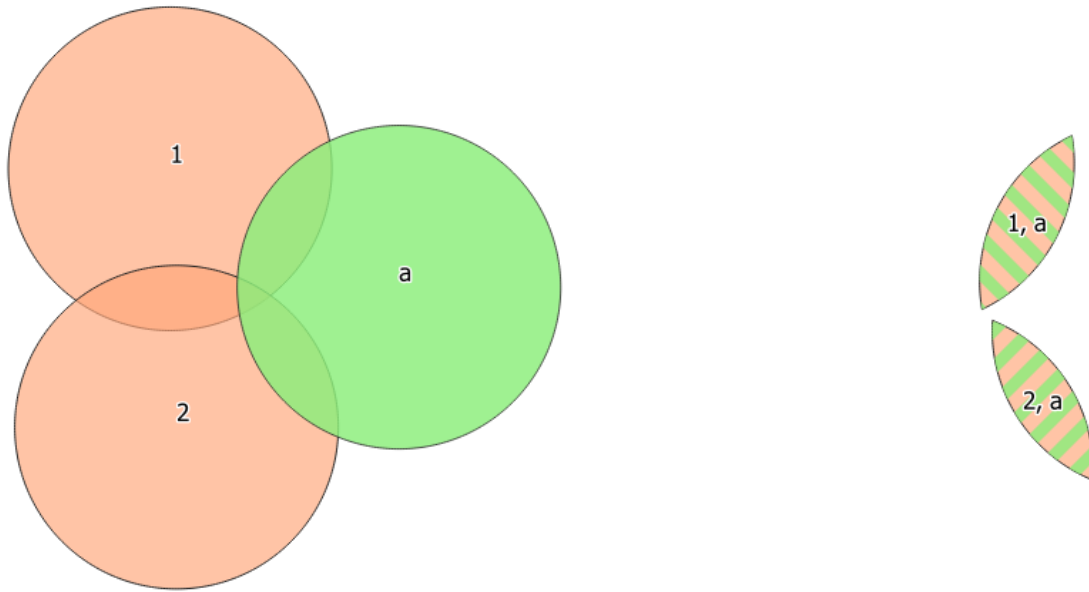


图 24.94: The intersection operation: A two-features input layer and a single feature overlay layer (left) - resulting features are moved for clarity (right)

Default menu: *Vector ► Geoprocessing Tools*

参见:

Clip, Difference

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Layer to extract (parts of) features from.
Overlay layer	OVERLAY	[vector: any]	Layer containing the features to check for overlap. Its features' geometry is expected to have at least as many dimensions (point: 0D, line: 1D, polygon: 2D, volume: 3D) as the input layer's.
Input fields to keep (leave empty to keep all fields) Optional	INPUT_FIELDS	[tablefield: any] [list] Default: None	Field(s) of the input layer to keep in the output. If no fields are chosen all fields are taken.
Overlay fields to keep (leave empty to keep all fields) Optional	OVERLAY_FIELDS	[tablefield: any] [list] Default: None	Field(s) of the overlay layer to keep in the output. If no fields are chosen all fields are taken.
Overlay fields prefix Optional	OVERLAY_FIELDS_PREFIX	string	Prefix to add to the field names of the intersect layer's fields to avoid name collisions with fields in the input layer.

下页继续

表 24.127 - 续上页

Label	名称	Type	Description
Intersection	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the layer to contain (the parts of) the features from the input layer that overlap one or more features from the overlay layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Intersection	OUTPUT	[same as input]	Layer containing (parts of) features from the input layer that overlap the overlay layer.

Python code

Algorithm ID: qgis:intersection

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Line intersections

Creates point features where the lines from the two layers intersect.

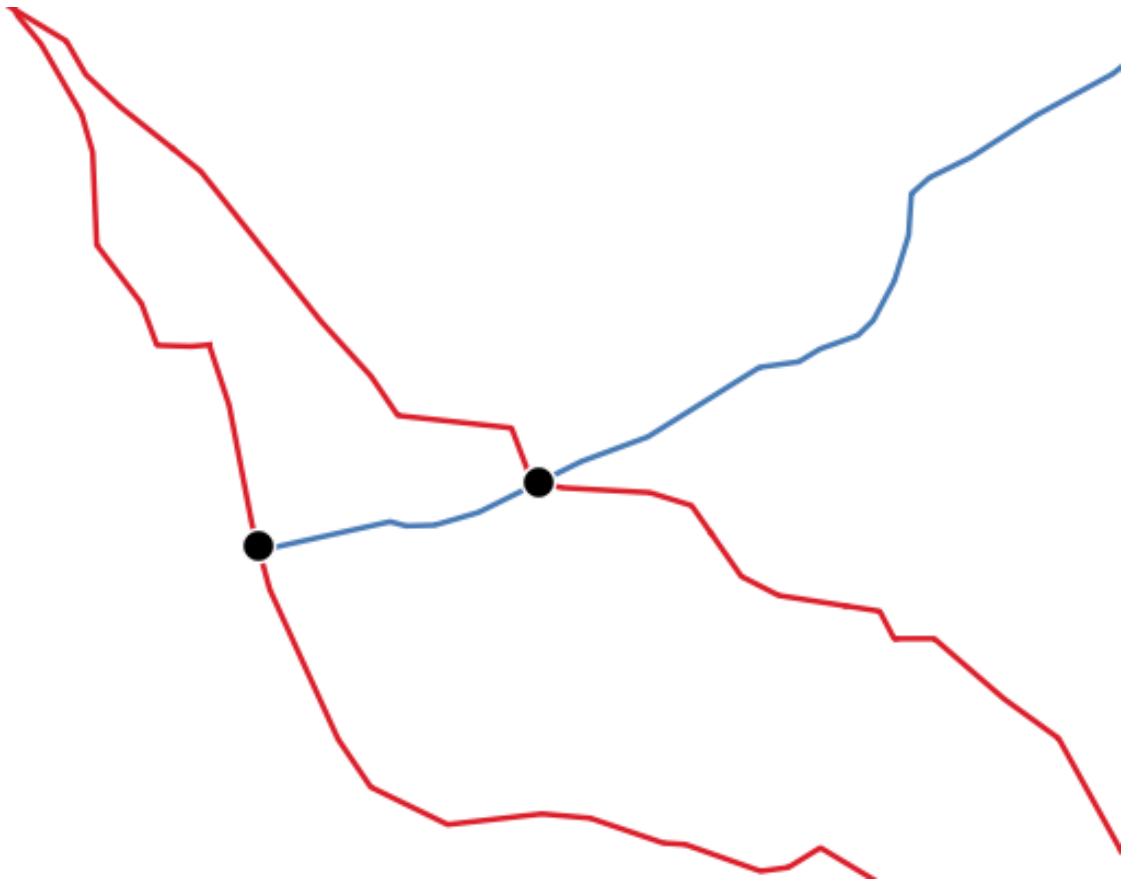


图 24.95: Points of intersection

Default menu: *Vector* ► *Analysis Tools*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	Input line layer.
Intersect layer	INTERSECT	[vector: line]	Layer to use to find line intersections.
Input fields to keep (leave empty to keep all fields) Optional	INPUT_FIELDS	[tablefield: any] [list] Default: None	Field(s) of the input layer to keep in the output. If no fields are chosen all fields are taken.
Intersect fields to keep (leave empty to keep all fields) Optional	INTERSECT_FIELDS	[tablefield: any] [list] Default: None	Field(s) of the intersect layer to keep in the output. If no fields are chosen all fields are taken.
Intersect fields prefix Optional	OVERLAY_FIELDS_PREFIX	[string]	Prefix to add to the field names of the intersect layer's fields to avoid name collisions with fields in the input layer.

下页继续

表 24.129 - 续上页

Label	名称	Type	Description
Intersection	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the layer to contain the intersection points of the lines from the input and overlay layers. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Intersections	OUTPUT	[vector: point]	Point vector layer with the intersections.

Python code

Algorithm ID: qgis:lineintersections

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Split with lines

Splits the lines or polygons in one layer using the lines in another layer to define the breaking points. Intersection between geometries in both layers are considered as split points.

Output will contain multi geometries for split features.

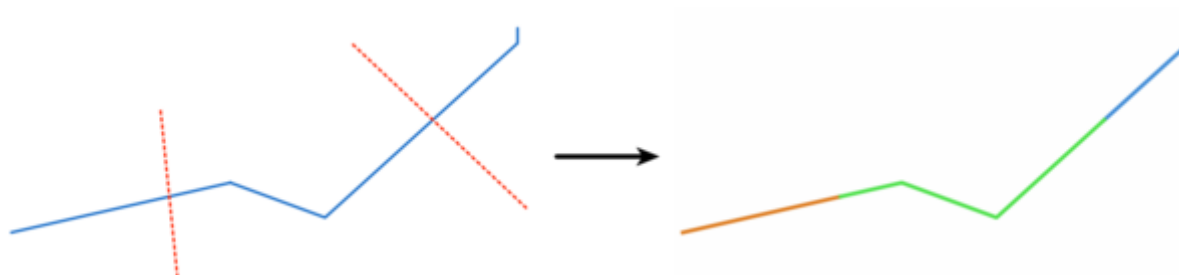


图 24.96: Split lines

Allows *features in-place modification*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line, polygon]	Layer containing the lines or polygons to split.
Split layer	LINES	[vector: line]	Line layer whose lines are used to define the breaking points.
Split	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the layer to contain the splitted (in case they are intersected by a line in the split layer) line/polygon features from the input layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Split	OUTPUT	[same as input]	Output vector layer with split lines or polygons from input layer.

Python code

Algorithm ID: qgis:splitwithlines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Symmetrical difference

Creates a layer containing features from both the input and overlay layers but with the overlapping areas between the two layers removed.

The attribute table of the symmetrical difference layer contains attributes and fields from both the input and overlay layers.

Attributes are not modified (see *warning*).

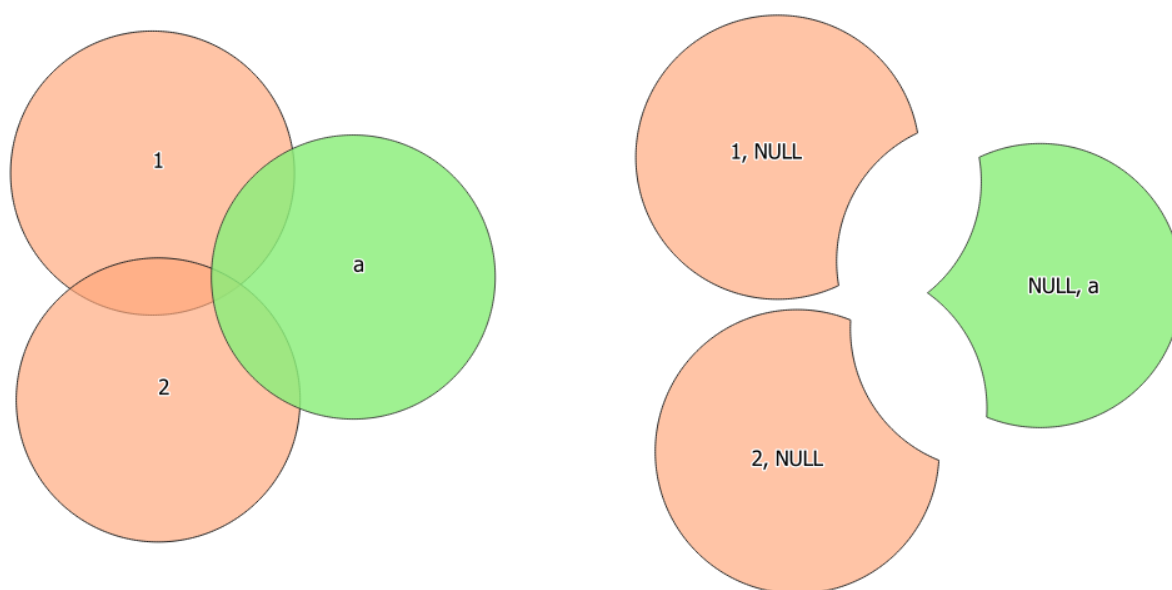


图 24.97: Symmetrical difference operation between a two-features input layer and a single feature overlay layer (left) - resulting features are moved for clarity (right)

Default menu: *Vector ► Geoprocessing Tools*

参见:

Difference, Clip, Intersection

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	First layer to extract (parts of) features from.
Overlay layer	OVERLAY	[vector: any]	Second layer to extract (parts of) features from. Ideally the geometry type should be the same as input layer.
Overlay fields prefix Optional	OVERLAY_FIELDS_PREFIX	[string]	Prefix to add to the field names of the overlay layer's fields to avoid name collisions with fields in the input layer.
Symmetrical difference	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the layer to contain (the parts of) the features from the input and overlay layers that do not overlap features from the other layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Symmetrical difference	OUTPUT	[same as input]	Layer containing (parts of) features from each layer not overlapping the other layer.

Python code

Algorithm ID: qgis:symmetricaldifference

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Union

Checks overlaps between features within the input layer and creates separate features for overlapping and non-overlapping parts. The area of overlap will create as many identical overlapping features as there are features that participate in that overlap.

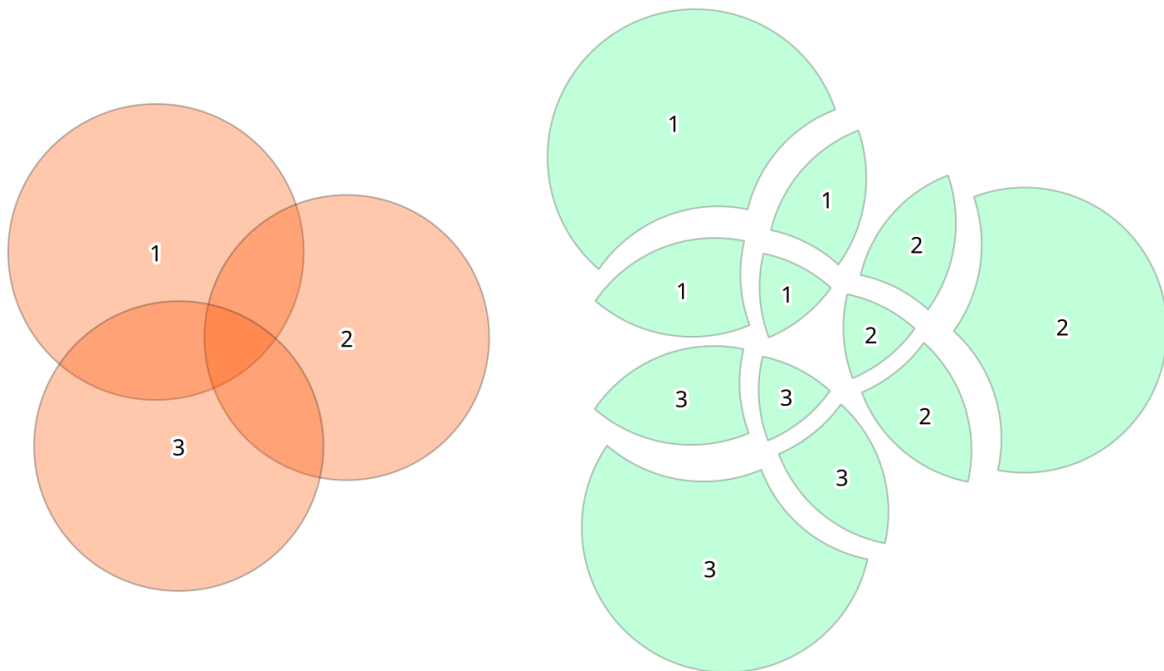


图 24.98: Union operation with a single input layer of three overlapping features (left) - resulting features are moved for clarity (right)

An overlay layer can also be used, in which case features from each layer are split at their overlap with features from the other one, creating a layer containing all the portions from both input and overlay layers. The attribute table of the union layer is filled with attribute values from the respective original layer for non-overlapping features, and attribute values from both layers for overlapping features.

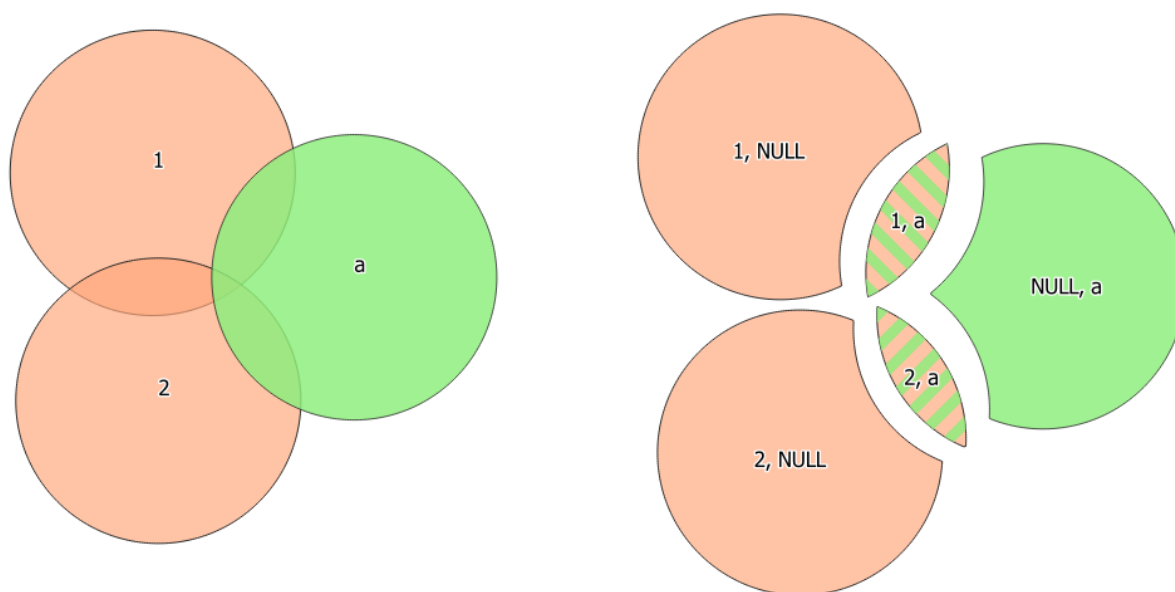


图 24.99: Union operation between a two-features input layer and a single feature overlay layer (left) - resulting features are moved for clarity (right)

注解: For $\text{union}(A, B)$ algorithm, if there are overlaps among geometries of layer A or among geometries of layer B, these are not resolved: you need to do $\text{union}(\text{union}(A, B))$ to resolve all overlaps, i.e. run single layer $\text{union}(X)$ on the produced result $X = \text{union}(A, B)$.

Default menu: *Vector ► Geoprocessing Tools*

参见:

Clip, Difference, Intersection

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer to split at any intersections.
Overlay layer Optional	OVERLAY	[vector: any]	Layer that will be combined to the first one. Ideally the geometry type should be the same as input layer.
Overlay fields prefix Optional	OVERLAY_FIELDS_PREFIX	[string]	Prefix to add to the field names of the overlay layer's fields to avoid name collisions with fields in the input layer.
Union	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the layer to contain the (split and duplicated) features from the input layer and the overlay layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Union	OUTPUT	[same as input]	Layer containing all the overlapping and non-overlapping parts from the processed layer(s).

Python code

Algorithm ID: qgis:union

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.1.18 Vector selection

Extract by attribute

Creates two vector layers from an input layer: one will contain only matching features while the second will contain all the non-matching features.

The criteria for adding features to the resulting layer is based on the values of an attribute from the input layer.

参见:

Select by attribute

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Layer to extract features from.
Selection attribute	FIELD	[tablefield: any]	Filtering field of the layer
Operator	OPERATOR	[enumeration] Default: 0	Many different operators are available: <ul style="list-style-type: none"> • 0 --- = • 1 --- ≠ • 2 --- > • 3 --- >= • 4 --- < • 5 --- <= • 6 --- begins with • 7 --- contains • 8 --- is null • 9 --- is not null • 10 --- does not contain
Value Optional	VALUE	[string]	Value to be evaluated

下页继续

表 24.130 – 续上页

Label	名称	Type	Description
Extracted (attribute)	OUTPUT	[same as input] Default: [Create Temporary Layer]	Specify the output vector layer for matching features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Extracted (non-matching)	FAIL_OUTPUT	[same as input] Default: [Skip output]	Specify the output vector layer for non-matching features. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table...

Outputs

Label	名称	Type	Description
Extracted (attribute)	OUTPUT	[same as input]	Vector layer with matching features from the input layer
Extracted (non-matching)	FAIL_OUTPUT	[same as input]	Vector layer with non-matching features from the input layer

Python code

Algorithm ID: qgis:extractbyattribute

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Extract by expression

Creates two vector layers from an input layer: one will contain only matching features while the second will contain all the non-matching features.

The criteria for adding features to the resulting layer is based on a QGIS expression. For more information about expressions see the [Expressions](#).

参见:

[Select by expression](#)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Expression	EXPRESSION	[expression]	Expression to filter the vector layer
Matching features	OUTPUT	[same as input] Default: [Create Temporary Layer]	Specify the output vector layer for matching features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.
Non-matching	FAIL_OUTPUT	[same as input] Default: [Skip output]	Specify the output vector layer for non-matching features. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table...

Outputs

Label	名称	Type	Description
Matching features	OUTPUT	[same as input]	Vector layer with matching features from the input layer
Non-matching	FAIL_OUTPUT	[same as input]	Vector layer with non-matching features from the input layer

Python code

Algorithm ID: qgis:extractbyexpression

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Extract by location

Creates a new vector layer that only contains matching features from an input layer.

The criteria for adding features to the resulting layer is based on the spatial relationship between each feature and the features in an additional layer.

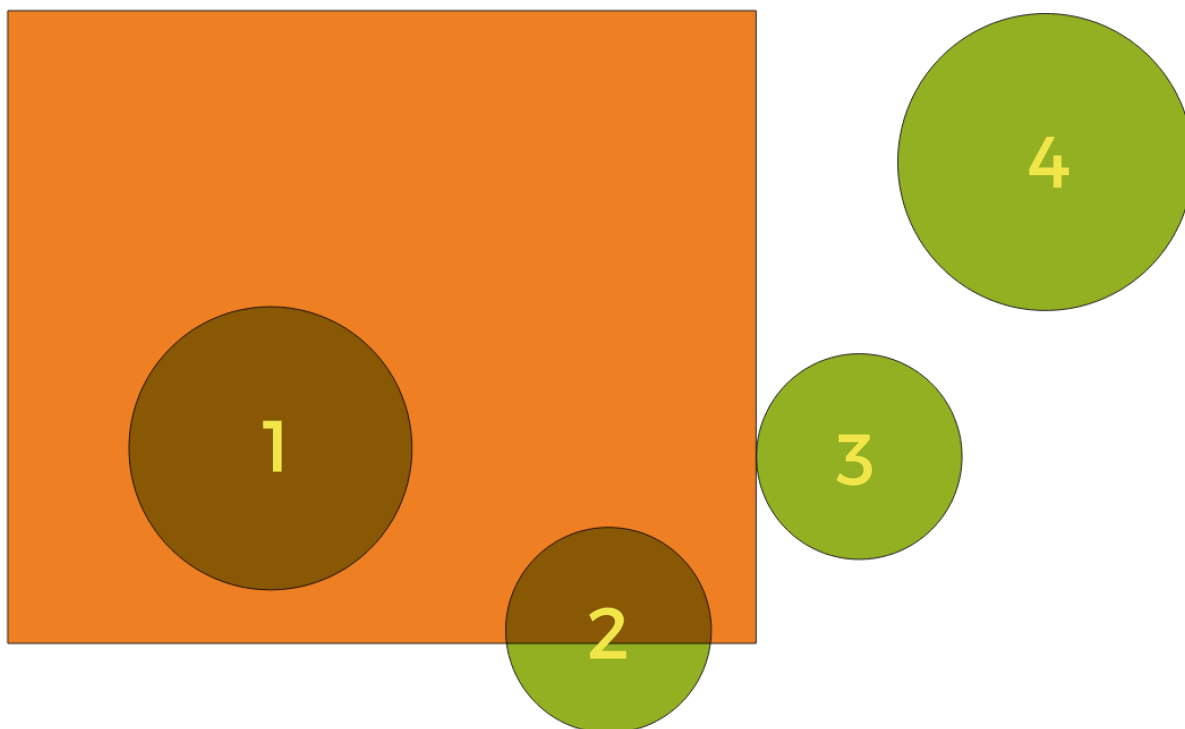


图 24.100: In this example, the dataset from which we want to select (the *source vector layer*) consists of the green circles, the orange rectangle is the dataset that it is being compared to (the *intersection vector layer*).

Available geometric predicates are:

Intersect Tests whether a geometry intersects another. Returns 1 (true) if the geometries spatially intersect (share any portion of space - overlap or touch) and 0 if they don't. In the picture above, this will select circles 1, 2 and 3.

Contain Returns 1 (true) if and only if no points of b lie in the exterior of a, and at least one point of the interior of b lies in the interior of a. In the picture, no circle is selected, but the rectangle would be if you would select it the other way around, as it contains a circle completely. This is the opposite of *are within*.

Disjoint Returns 1 (true) if the geometries do not share any portion of space (no overlap, not touching). Only circle 4 is selected.

Equal Returns 1 (true) if and only if geometries are exactly the same. No circles will be selected.

Touch Tests whether a geometry touches another. Returns 1 (true) if the geometries have at least one point in common, but their interiors do not intersect. Only circle 3 is selected.

Overlap Tests whether a geometry overlaps another. Returns 1 (true) if the geometries share space, are of the same dimension, but are not completely contained by each other. Only circle 2 is selected.

Are within Tests whether a geometry is within another. Returns 1 (true) if geometry a is completely inside geometry b. Only circle 1 is selected.

Cross Returns 1 (true) if the supplied geometries have some, but not all, interior points in common and the actual crossing is of a lower dimension than the highest supplied geometry. For example, a line crossing a polygon will cross as a line (selected). Two lines crossing will cross as a point (selected). Two polygons cross as a polygon (not selected).

参见:

[Select by location](#)

Parameters

Label	名称	Type	Description
Extract features from	INPUT	[vector: any]	Input vector layer
Where the features (geometric predicate)	PREDICATE	[enumeration] [list] Default: [0]	Spatial condition for the selection. One or more of: <ul style="list-style-type: none"> • 0 --- intersect • 1 --- contain • 2 --- disjoint • 3 --- equal • 4 --- touch • 5 --- overlap • 6 --- are within • 7 --- cross <p>If more than one condition is chosen, at least one of them (OR operation) has to be met for a feature to be extracted.</p>
By comparing to the features from	INTERSECT	[vector: any]	Intersection vector layer
Extracted (location)	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer for the features that have the chosen spatial relationship(s) with one or more features in the comparison layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table...

Outputs

Label	名称	Type	Description
Extracted (location)	OUTPUT	[same as input]	Vector layer with features from the input layer that have the chosen spatial relationship(s) with one or more features in the comparison layer.

Python code

Algorithm ID: qgis:extractbylocation

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Random extract

Takes a vector layer and generates a new one that contains only a subset of the features in the input layer.

The subset is defined randomly, based on feature IDs, using a percentage or count value to define the total number of features in the subset.

参见:

Random selection

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Source vector layer to select the features from
Method	METHOD	[enumeration] Default: 0	Random selection methods. One of: <ul style="list-style-type: none"> • 0 --- Number of selected features • 1 --- Percentage of selected features
Number/percentage of selected features	NUMBER	[number] Default: 10	Number or percentage of features to select
Extracted (random)	OUTPUT	[vector: any] Default: [Create temporary layer]	Specify the output vector layer for the randomly selected features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... Vector layer containing randomly selected features

Outputs

Label	名称	Type	Description
Extracted (random)	OUTPUT	[same as input]	Vector layer containing randomly selected features from the input layer

Python code

Algorithm ID: `qgis:randomextract`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random extract within subsets

Takes a vector layer and generates a new one that contains only a subset of the features in the input layer.

The subset is defined randomly, based on feature IDs, using a percentage or count value to define the total number of features in the subset. The percentage/count value is not applied to the whole layer, but instead to each category. Categories are defined according to a given attribute.

参见:

Random selection within subsets

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Vector layer to select the features from
ID field	FIELD	[tablefield: any]	Category of the source vector layer to select the features from
Method	METHOD	[enumeration] Default: 0	Random selection method. One of: <ul style="list-style-type: none"> • 0 --- Number of selected features • 1 --- Percentage of selected features
Number/percentage of selected features	NUMBER	[number] Default: 10	Number or percentage of features to select
Extracted (random stratified)	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer for the randomly selected features. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Extracted (random stratified)	OUTPUT	[same as input]	Vector layer containing randomly selected features from the input layer

Python code

Algorithm ID: qgis:randomextractwithinsubsets

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random selection

Takes a vector layer and selects a subset of its features. No new layer is generated by this algorithm.

The subset is defined randomly, based on feature IDs, using a percentage or count value to define the total number of features in the subset.

Default menu: *Vector ► Research Tools*

参见:

Random extract

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Vector layer for the selection
Method	METHOD	[enumeration] Default: 0	Random selection method. One of: <ul style="list-style-type: none"> • 0 --- Number of selected features • 1 --- Percentage of selected features
Number/percentage of selected features	NUMBER	[number] Default: 10	Number or percentage of features to select

Outputs

Label	名称	Type	Description
Input layer	INPUT	[same as input]	The input layer with features selected

Python code

Algorithm ID: `qgis:randomselection`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Random selection within subsets

Takes a vector layer and selects a subset of its features. No new layer is generated by this algorithm.

The subset is defined randomly, based on feature IDs, using a percentage or count value to define the total number of features in the subset.

The percentage/count value is not applied to the whole layer, but instead to each category.

Categories are defined according to a given attribute, which is also specified as an input parameter for the algorithm.

No new outputs are created.

Default menu: *Vector ► Research Tools*

参见:

Random extract within subsets

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Vector layer to select features in
ID field	FIELD	[tablefield: any]	Category of the input layer to select the features from
Method	METHOD	[enumeration] Default: 0	Random selection method. One of: <ul style="list-style-type: none"> • 0 --- Number of selected features • 1 --- Percentage of selected features
Number/percentage of selected features	NUMBER	[number] Default: 10	Number or percentage of features to select

Outputs

Label	名称	Type	Description
Input layer	INPUT	[same as input]	The input layer with features selected

Python code

Algorithm ID: qgis:randomselectionwithinsubsets

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Select by attribute

Creates a selection in a vector layer.

The criteria for selecting features is based on the values of an attribute from the input layer.

参见:

Extract by attribute

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Vector layer to select features in
Selection attribute	FIELD	[tablefield: any]	Filtering field of the layer

下页继续

表 24.134 – 续上页

Label	名称	Type	Description
Operator	OPERATOR	[enumeration] Default: 0	Many different operators are available: <ul style="list-style-type: none"> • 0 --- = • 1 --- ≠ • 2 --- > • 3 --- >= • 4 --- < • 5 --- <= • 6 --- begins with • 7 --- contains • 8 --- is null • 9 --- is not null • 10 --- does not contain
Value Optional	VALUE	[string]	Value to be evaluated
Modify current selection by	METHOD	[enumeration] Default: 0	How the selection of the algorithm should be managed. One of: <ul style="list-style-type: none"> • 0 --- creating new selection • 1 --- adding to current selection • 2 --- removing from current selection • 3 --- selecting within current selection

Outputs

Label	名称	Type	Description
Input layer	INPUT	[same as input]	The input layer with features selected

Python code

Algorithm ID: qgis:selectbyattribute

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Select by expression

Creates a selection in a vector layer.

The criteria for selecting features is based on a QGIS expression. For more information about expressions see the [Expressions](#).

参见:

[Extract by expression](#)

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Expression	EXPRESSION	[expression]	Expression to filter the input layer
Modify current selection by	METHOD	[enumeration] Default: 0	How the selection of the algorithm should be managed. One of: <ul style="list-style-type: none"> • 0 --- creating new selection • 1 --- adding to current selection • 2 --- removing from current selection • 3 --- selecting within current selection

Outputs

Label	名称	Type	Description
Input layer	INPUT	[same as input]	The input layer with features selected

Python code

Algorithm ID: qgis:selectbyexpression

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Select by location

Creates a selection in a vector layer.

The criteria for selecting features is based on the spatial relationship between each feature and the features in an additional layer.

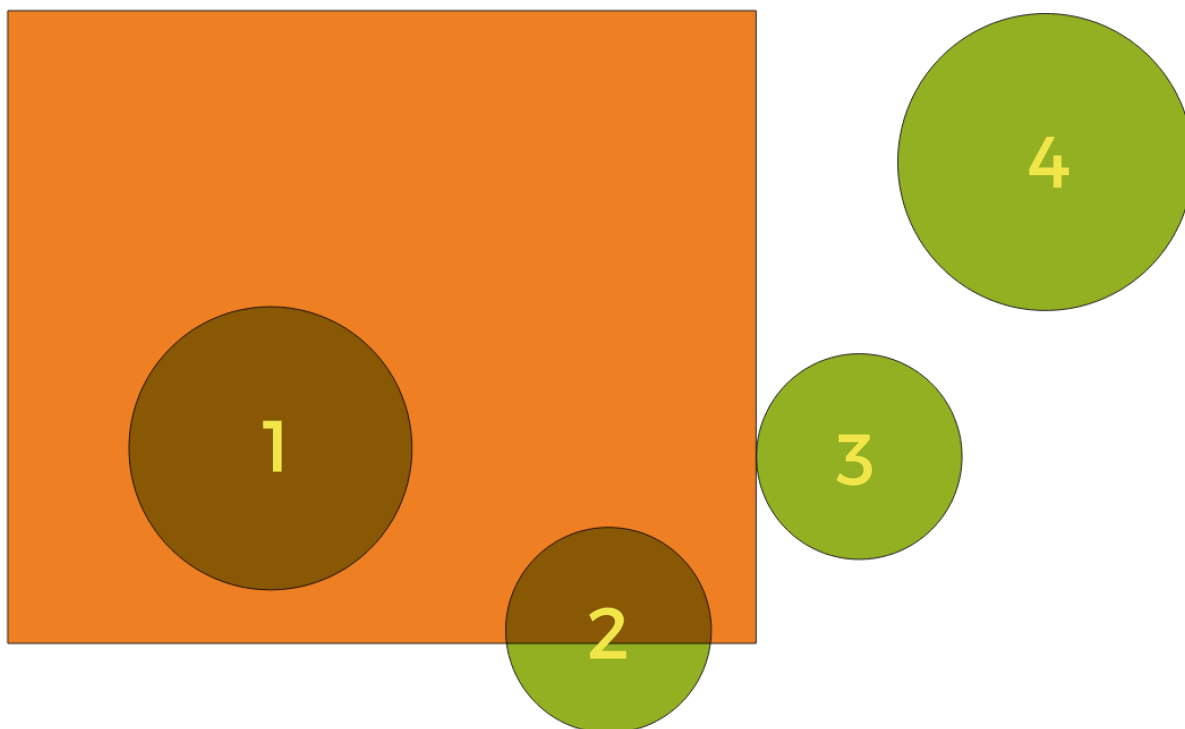


图 24.101: In this example, the dataset from which we want to select (the *source vector layer*) consists of the green circles, the orange rectangle is the dataset that it is being compared to (the *intersection vector layer*).

Available geometric predicates are:

Intersect Tests whether a geometry intersects another. Returns 1 (true) if the geometries spatially intersect (share any portion of space - overlap or touch) and 0 if they don't. In the picture above, this will select circles 1, 2 and 3.

Contain Returns 1 (true) if and only if no points of b lie in the exterior of a, and at least one point of the interior of b lies in the interior of a. In the picture, no circle is selected, but the rectangle would be if you would select it the other way around, as it contains a circle completely. This is the opposite of *are within*.

Disjoint Returns 1 (true) if the geometries do not share any portion of space (no overlap, not touching). Only circle 4 is selected.

Equal Returns 1 (true) if and only if geometries are exactly the same. No circles will be selected.

Touch Tests whether a geometry touches another. Returns 1 (true) if the geometries have at least one point in common, but their interiors do not intersect. Only circle 3 is selected.

Overlap Tests whether a geometry overlaps another. Returns 1 (true) if the geometries share space, are of the same dimension, but are not completely contained by each other. Only circle 2 is selected.

Are within Tests whether a geometry is within another. Returns 1 (true) if geometry a is completely inside geometry b. Only circle 1 is selected.

Cross Returns 1 (true) if the supplied geometries have some, but not all, interior points in common and the actual crossing is of a lower dimension than the highest supplied geometry. For example, a line crossing a polygon will cross as a line (selected). Two lines crossing will cross as a point (selected). Two polygons cross as a polygon (not selected).

Default menu: *Vector ► Research Tools*

参见:

Extract by location

Parameters

Label	名称	Type	Description
Select features from	INPUT	[vector: any]	Input vector layer
Where the features (geometric predicate)	PREDICATE	[enumeration] [list] Default: [0]	Spatial condition for the selection. One or more of: <ul style="list-style-type: none"> • 0 --- intersect • 1 --- contain • 2 --- disjoint • 3 --- equal • 4 --- touch • 5 --- overlap • 6 --- are within • 7 --- cross If more than one condition is chosen, at least one of them (OR operation) has to be met for a feature to be extracted.
By comparing to the features from	INTERSECT	[vector: any]	Intersection vector layer
Modify current selection by	METHOD	[enumeration] Default: 0	How the selection of the algorithm should be managed. One of: <ul style="list-style-type: none"> • 0 --- creating new selection • 1 --- adding to current selection • 2 --- selecting within current selection • 3 --- removing from current selection

Outputs

Label	名称	Type	Description
Input layer	INPUT	[same as input]	The input layer with features selected

Python code

Algorithm ID: qgis:selectbylocation

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.1.19 Vector table

Add autoincremental field

Adds a new integer field to a vector layer, with a sequential value for each feature.

This field can be used as a unique ID for features in the layer. The new attribute is not added to the input layer but a new layer is generated instead.

The initial starting value for the incremental series can be specified. Optionally, the incremental series can be based on grouping fields and a sort order for features can also be specified.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer.
Field name	FIELD_NAME	[string] Default: 'AUTO'	Name of the field with autoincremental values
Start values at Optional	START	[number] Default: 0	Choose the initial number of the incremental count
Group values by Optional	GROUP_FIELDS	[tablefield: any] [list]	Select grouping field(s): instead of a single count run for the whole layer, a separate count is processed for each value returned by the combination of these fields.
Sort expression Optional	SORT_EXPRESSION	[expression]	Use an expression to sort the features in the layer either globally or if set, based on group fields.
Sort ascending	SORT_ASCENDING	[boolean] Default: True	When a sort expression is set, use this option to control the order in which features are assigned values.
Sort nulls first	SORT_NULLS_FIRST	[boolean] Default: False	When a sort expression is set, use this option to set whether <i>Null</i> values are counted first or last.
Incremented	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer with the auto increment field. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Incremented	OUTPUT	[same as input]	Vector layer with auto incremental field

Python code

Algorithm ID: qgis:addautoincrementalfield

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Add field to attributes table

Adds a new field to a vector layer.

The name and characteristics of the attribute are defined as parameters.

The new attribute is not added to the input layer but a new layer is generated instead.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input layer
Field name	FIELD_NAME	[string]	Name of the new field
Field type	FIELD_TYPE	[enumeration] Default: 0	Type of the new field. You can choose between: <ul style="list-style-type: none"> • 0 --- Integer • 1 --- Float • 2 --- String
Field length	FIELD_LENGTH	[number] Default: 10	Length of the field
Field precision	FIELD_PRECISION	[number] Default: 0	Precision of the field. Useful with Float field type.
Added	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Added	OUTPUT	[same as input]	Vector layer with new field added

Python code

Algorithm ID: qgis:addfieldtoattributetable

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Add unique value index field

Takes a vector layer and an attribute and adds a new numeric field.

Values in this field correspond to values in the specified attribute, so features with the same value for the attribute will have the same value in the new numeric field.

This creates a numeric equivalent of the specified attribute, which defines the same classes.

The new attribute is not added to the input layer but a new layer is generated instead.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input layer.
Class field	FIELD	[tablefield: any]	Features that have the same value for this field will get the same index.
Output field name	FIELD_NAME	[string] Default: 'NUM_FIELD'	Name of the new field containing the indexes.
Layer with index field	OUTPUT	[vector: any] Default: [Create temporary layer]	Vector layer with the numeric field containing indexes. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.
Class summary	SUMMARY_OUTPUT	[table] Default: [Skip output]	Specify the table to contain the summary of the class field mapped to the corresponding unique value. One of: <ul style="list-style-type: none"> • Skip Output • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Layer with index field	OUTPUT	[same as input]	Vector layer with the numeric field containing indexes.
Class summary	SUMMARY_OUTPUT	[table] Default: [Skip Output]	Table with summary of the class field mapped to the corresponding unique value.

Python code

Algorithm ID: qgis:adduniquevalueindexfield

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Add X/Y fields to layer

Adds X and Y (or latitude/longitude) fields to a point layer. The X/Y fields can be calculated in a different CRS to the layer (e.g. creating latitude/longitude fields for a layer in a projected CRS).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: point]	The input layer.
Coordinate system	CRS	[crs] Default: "EPSG:4326"	Coordinate reference system to use for the generated x and y fields.
Field prefix Optional	PREFIX	[string]	Prefix to add to the new field names to avoid name collisions with fields in the input layer.
Added fields	OUTPUT	[vector: point] Default: [Create temporary layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Added fields	OUTPUT	[vector: point]	The output layer - identical to the input layer but with two new double fields, <i>x</i> and <i>y</i> .

Python code

Algorithm ID: qgis:addxyfieldstolayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Advanced Python field calculator

Adds a new attribute to a vector layer, with values resulting from applying an expression to each feature.

The expression is defined as a Python function.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Result field name	FIELD_NAME	[string] Default: 'NewField'	Name of the new field
Field type	FIELD_TYPE	[enumeration] Default: 0	Type of the new field. One of: <ul style="list-style-type: none"> • 0 --- Integer • 1 --- Float • 2 --- String
Field length	FIELD_LENGTH	[number] Default: 10	Length of the field
Field precision	FIELD_PRECISION	[number] Default: 3	Precision of the field. Useful with Float field type.
Global expression Optional	GLOBAL	[string]	The code in the global expression section will be executed only once before the calculator starts iterating through all the features of the input layer. Therefore, this is the correct place to import necessary modules or to calculate variables that will be used in subsequent calculations.
Formula	FORMULA	[string]	The Python formula to evaluate. Example: To calculate the area of an input polygon layer you can add: value = \$geom.area()

下页继续

表 24.140 – 续上页

Label	名称	Type	Description
Calculated	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the vector layer with the new calculated field. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Calculated	OUTPUT	[same as input]	Vector layer with the new calculated field

Python code

Algorithm ID: qgis:advancedpythonfieldcalculator

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Drop field(s)

Takes a vector layer and generates a new one that has the same features but without the selected columns.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer to drop field(s) from
Fields to drop	COLUMN	[tablefield: any] [list]	The field(s) to drop
Remaining fields	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer with the remaining fields. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Remaining fields	OUTPUT	[same as input]	Vector layer with the remaining fields

Python code

Algorithm ID: qgis:deletecolumn

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Explode HStore Field

Creates a copy of the input layer and adds a new field for every unique key in the HStore field.

The expected field list is an optional comma separated list. If this list is specified, only these fields are added and the HStore field is updated. By default, all unique keys are added.

The PostgreSQL *HStore* is a simple key-value store used in PostgreSQL and OGR (when reading an *OSM file* with the *other_tags* field).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
HStore field	FIELD	[tablefield: any]	The field(s) to drop
Expected list of fields separated by a comma Optional	EXPECTED_FIELDS	[string] Default: ""	Comma-separated list of fields to extract. The HStore field will be updated by removing these keys.
Exploded	OUTPUT	[same as input] Default: [Create temporary layer]	Specify the output vector layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Exploded	OUTPUT	[same as input]	Output vector layer

Python code

Algorithm ID: qgis:explodehstorefield

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extract binary field

Extracts contents from a binary field, saving them to individual files. Filenames can be generated using values taken from an attribute in the source table or based on a more complex expression.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer containing the binary data
Binary field	FIELD	[tablefield: any]	Field containing the binary data
File name	FILENAME	[expression]	Field or expression-based text to name each output file
Destination folder	FOLDER	[folder] Default: [Save to a temporary folder]	Folder in which to store the output files. One of: <ul style="list-style-type: none"> • Save to a Temporary Directory • Save to Directory... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Folder	FOLDER	[folder]	The folder that contains the output files.

Python code

Algorithm ID: qgis:extractbinary

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Feature filter

Filters features from the input layer and redirects them to one or several outputs. If you do not know about any attribute names that are common to all possible input layers, filtering is only possible on the feature geometry and general record mechanisms, such as \$id and uuid.

注解: This algorithm is only available from the *Graphical modeler*.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input layer.
Outputs and filters (one or more)	OUTPUT_<name of the filter>	[same as input]	The output layers with filters (as many as there are filters).

Outputs

Label	名称	Type	Description
Output (one or more)	native:filter_1 of filter>	[same as input]	The output layers with filtered features (as many as there are filters).

Python code

Algorithm ID: qgis:featurefilter

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Field calculator

Opens the field calculator (see *Expressions*). You can use all the supported expressions and functions.

A new layer is created with the result of the expression.

The field calculator is very useful when used in *The graphical modeler*.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The layer to calculate on
Output field name	FIELD_NAME	[string]	The name of the field for the results
Output field type	FIELD_TYPE	[enumeration] Default: 0	The type of the field. One of: <ul style="list-style-type: none"> • 0 --- Float • 1 --- Integer • 2 --- String • 3 --- Date
Output field width	FIELD_LENGTH	[number] Default: 10	The length of the result field (minimum 0)
Field precision	FIELD_PRECISION	[number] Default: 3	The precision of the result field (minimum 0, maximum 15)
Create new field	NEW_FIELD	[boolean] Default: True	Should the result field be a new field
Formula	FORMULA	[expression]	The formula to use to calculate the result
Output file	OUTPUT	[vector: any] Default: [Save to temporary file]	Specification of the output layer.

Outputs

Label	名称	Type	Description
Calculated	OUTPUT	[vector: any]	Output layer with the calculated field values

Python code

Algorithm ID: qgis:fieldcalculator

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Refactor fields

Allows editing the structure of the attribute table of a vector layer.

Fields can be modified in their type and name, using a fields mapping.

The original layer is not modified. A new layer is generated, which contains a modified attribute table, according to the provided fields mapping.

Refactor layer fields allows to:

- Change field names and types
- Add and remove fields
- Reorder fields

- Calculate new fields based on expressions
- Load field list from another layer

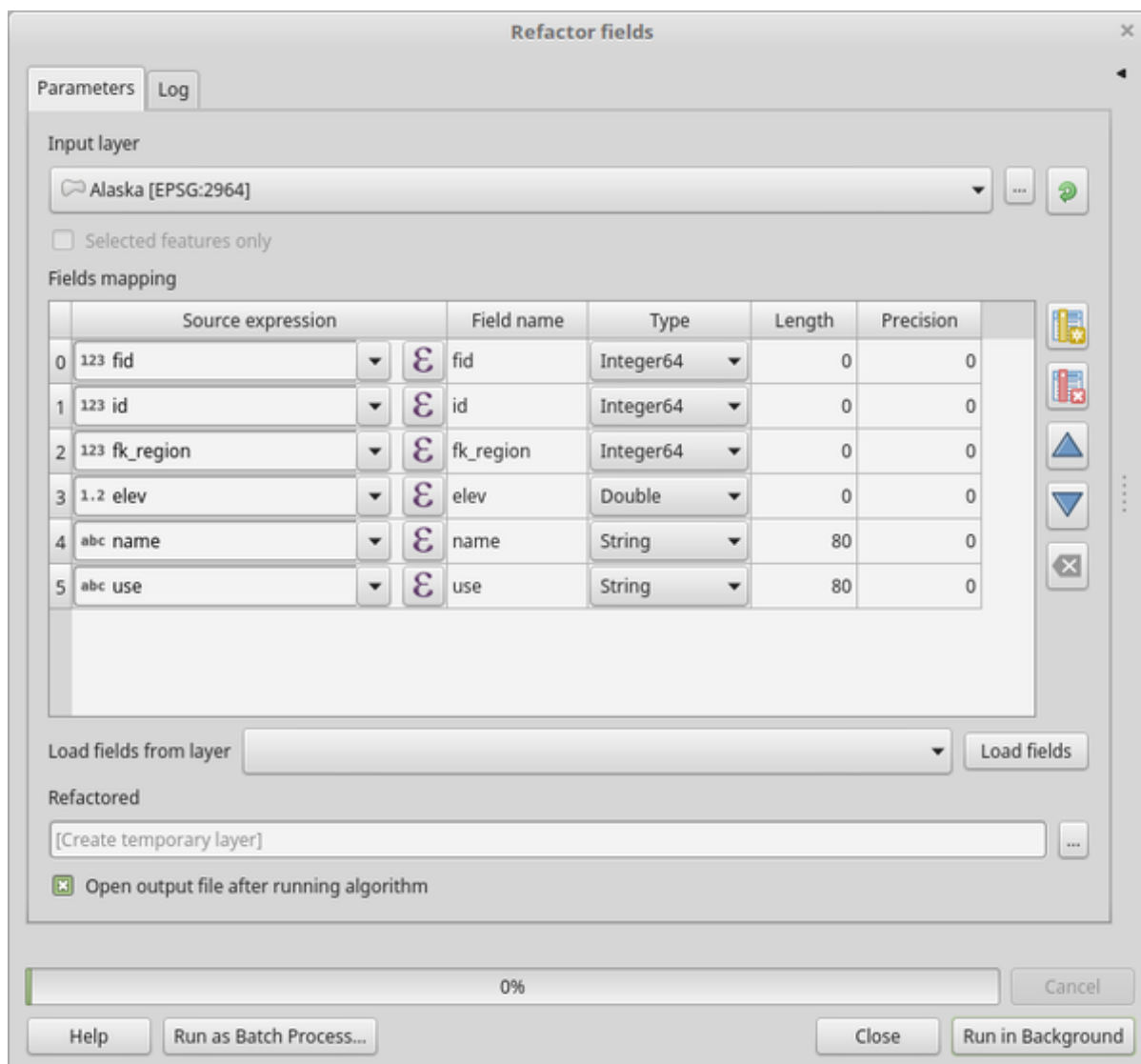







图 24.102: Refactor fields dialog

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The layer to modify

下页继续

表 24.143 – 续上页

Label	名称	Type	Description
Fields mapping	FIELDS_MAPPING	[list]	<p>List of output fields with their definitions. The embedded table lists all the fields of the source layer and allows you to edit them:</p> <ul style="list-style-type: none"> • Click  to create a new field. • Click  to remove a field. • Use  and  to change the selected field order. • Click  to reset to the default view. <p>For each of the fields you'd like to reuse, you need to fill the following options:</p> <p>Source expression (expression) [expression] Field or expression from the input layer.</p> <p>Field name (name) [string] Name of the field in the output layer. By default input field name is kept.</p> <p>Type (type) [enumeration] Data type of the output field. One of:</p> <ul style="list-style-type: none"> • Date (14) • DateTime (16) • Double (6) • Integer (2) • Integer64 (4) • String (10) • Boolean (1) <p>Length (length) [number] Length of the output field.</p> <p>Precision (precision) [number] Precision of the output field.</p> <p>Fields from another layer can be loaded into the field list in <i>Load fields from layer</i>.</p>
Refactored	OUTPUT	[vector: any] Default: [Create temporary layer]	<p>Specification of the output layer. One of:</p> <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... <p>The file encoding can also be changed here.</p>

Outputs

Label	名称	Type	Description
Refactored	OUTPUT	[vector: any]	Output layer with refactored fields

Python code

Algorithm ID: qgis:refactorfields

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Rename vector field

Renames an existing field from a vector layer.

The original layer is not modified. A new layer is generated where the attribute table contains the renamed field.

参见:

Refactor fields

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer
Field to rename	FIELD	[string]	The field to be altered
New field name	NEW_NAME	[string]	The new field name
Renamed	OUTPUT	[vector: same as input] Default: [Create temporary layer]	Specification of the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Renamed	OUTPUT	[vector: same as input]	Output layer with the renamed field

Python code

Algorithm ID: qgis:renametablefield

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Text to float

Modifies the type of a given attribute in a vector layer, converting a text attribute containing numeric strings into a numeric attribute (e.g. '1' to 1.0).

The algorithm creates a new vector layer so the source one is not modified.

If the conversion is not possible the selected column will have NULL values.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer.
Text attribute to convert to float	FIELD	[tablefield: string]	The string field for the input layer that is to be converted to a float field.
Float from text	OUTPUT	[same as input] Default: [Create Temporary Layer]	Specify the output layer. One of: <ul style="list-style-type: none"> • Create Temporary Layer • Save to File... • Save to Geopackage... • Save to PostGIS Table..... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Float from text	OUTPUT	[same as input]	Output vector layer with the string field converted into a float field

Python code

Algorithm ID: qgis:texttofloat

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.1.20 Vector Tiles

Write vector tiles (MBTiles)

Exports one or more vector layers to vector tiles, a data format optimized for fast map rendering and small data size.

MBTiles is a specification for storing tiled map data in SQLite databases for immediate usage and for transfer. MBTiles files are known as tilesets.

Parameters

Label	Name	Type	Description
Input layers	INPUT	[vector: any][list]	A list of layers to combine to generate the vector tiles
Minimum zoom level	MIN_ZOOM	[number] Default: 0	The lowest zoom level for which the tileset provides data. Set between 0 and 24.
Maximum zoom level	MAX_ZOOM	[number] Default: 3	The highest zoom level for which the tileset provides data. Set between 0 and 24.
Extent Optional	EXTENT	[extent] Default: Not set	The maximum extent of the rendered map area. Bounds must define an area covered by all zoom levels.
Metadata: Name Optional	META_NAME	[string]	Name of the tileset
Metadata: Description Optional	META_DESCRIPTION	[string]	A description of the tileset's contents
Metadata: Attribution Optional	META_ATTRIBUTION	[string]	An attribution string, which explains the sources of data and/or style for the map.
Metadata: Version Optional	META_VERSION	[string]	The version of the tileset. This refers to a revision of the tileset itself, not of the MBTiles specification.
Metadata: Type Optional	META_TYPE	[string]	Type of tileset. Possible values are overlay or baselayer.
Metadata: Center Optional	META_CENTER	[string]	The center (string of comma-separated numbers: the longitude, latitude, and zoom level) of the default view of the map. Example: -122.1906, 37.7599, 11
Destination MBTiles	OUTPUT	[vector tiles] Default: [Save to temporary file]	Specification of the output MBTiles file. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Outputs

Label	Name	Type	Description
Destination MBTiles	OUTPUT	[file]	Output vector tiles .mbtiles file.

Python code

Algorithm ID: native:writevectortiles_mbtiles

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Write vector tiles (XYZ)

Exports one or more vector layers to vector tiles, a data format optimized for fast map rendering and small data size.

Parameters

Label	Name	Type	Description
File template	XYZ_TEMPLATE	[string] Default: '{z}/{x}/{y}.pbf'	Template to generate the vector tiles url
Input layers	INPUT	[vector: any][list]	A list of layers to combine to generate the vector tiles
Minimum zoom level	MIN_ZOOM	[number] Default: 0	The lowest zoom level for which the tileset provides data. Set between 0 and 24.
Maximum zoom level	MAX_ZOOM	[number] Default: 3	The highest zoom level for which the tileset provides data. Set between 0 and 24.
Extent Optional	EXTENT	[extent] Default: Not set	The maximum extent of the rendered map area. Bounds must define an area covered by all zoom levels.
Output directory	OUTPUT_DIRECTORY	[folder] Default: [Save to temporary folder]	Specification of the output vector tiles folder. One of: <ul style="list-style-type: none"> • Save to a Temporary Directory • Save to Directory

Outputs

Label	Name	Type	Description
Output directory	OUTPUT_DIRECTORY	[folder]	A folder containing different subsets of the vector tiles files (.pbf) stored in subfolders corresponding to the zoom levels.

Python code

Algorithm ID: native:writevectortiles_xyz

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.2 GDAL algorithm provider

GDAL (Geospatial Data Abstraction Library) is a translator library for raster and vector geospatial data formats. Algorithms in the Processing Framework are derived from the [GDAL raster programs](#) and [GDAL vector programs](#).

24.2.1 Raster analysis

Aspect

Generates an aspect map from any GDAL-supported elevation raster. Aspect is the compass direction that a slope faces. The pixels will have a value from 0-360° measured in degrees from north indicating the azimuth. On the northern hemisphere, the north side of slopes is often shaded (small azimuth from 0°-90°), while the southern side receives more solar radiation (higher azimuth from 180°-270°).

This algorithm is derived from the [GDAL DEM utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input elevation raster layer
Band number	BAND	[raster band] Default: 1	The number of the band to use as elevation
Return trigonometric angle instead of azimuth	TRIG_ANGLE	[boolean] Default: False	Activating the trigonometric angle results in different categories: 0° (East), 90° (North), 180° (West), 270° (South).
Return 0 for flat instead of -9999	ZERO_FLAT	[boolean] Default: False	Activating this option will insert a 0-value for the value -9999 on flat areas.
Compute edges	COMPUTE_EDGES	[boolean] Default: False	Generates edges from the elevation raster
Use Zevenbergen&Thorne formula instead of the Horn's one	ZEVENBERGEN	[boolean] Default: False	Activates Zevenbergen&Thorne formula for smooth landscapes
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see GDAL driver options section). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Aspect	OUTPUT	[raster] Default: [Save to temporary file]	Output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Aspect	OUTPUT	[raster]	Output raster with angle values in degrees

Python code

Algorithm ID: gdal:aspect

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Color relief

Generates a color relief map from any GDAL-supported elevation raster. Color reliefs can particularly be used to depict elevations. The Algorithm outputs a 4-band raster with values computed from the elevation and a text-based color configuration file. By default, the colors between the given elevation values are blended smoothly and the result is a nice colorized elevation raster.

This algorithm is derived from the [GDAL DEM utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input elevation raster layer
Band number	BAND	[raster band] Default: 1	The number of the band to use as elevation
Compute edges	COMPUTE_EDGES	[boolean] Default: False	Generates edges from the elevation raster
Color configuration file	COLOR_TABLE	[file]	A text-based color configuration file
Matching mode	MATCH_MODE	[enumeration] Default: 2	One of: <ul style="list-style-type: none"> • 0 --- Use strict color matching • 1 --- Use closest RGBA quadruples • 2 --- Use smoothly blended colours
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options

下页继续

表 24.146 – 续上页

Label	名称	Type	Description
Color relief	OUTPUT	[raster] Default: [Save to temporary file]	Output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Color relief	OUTPUT	[raster]	A 4-band output raster

Python code

Algorithm ID: gdal:colorrelief

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Fill nodata

Fill raster regions with no data values by interpolation from edges. The values for the no-data regions are calculated by the surrounding pixel values using inverse distance weighting. After the interpolation a smoothing of the results takes place. Input can be any GDAL-supported raster layer. This algorithm is generally suitable for interpolating missing regions of fairly continuously varying rasters (such as elevation models for instance). It is also suitable for filling small holes and cracks in more irregularly varying images (like airphotos). It is generally not so great for interpolating a raster from sparse point data.

This algorithm is derived from the [GDAL fillnodata utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Band number	BAND	[raster band] Default: 1	The band to operate on. Nodata values must be represented by the value 0.
Maximum distance (in pixels) to search out for values to interpolate	DISTANCE	[number] Default: 10	The number of pixels to search in all directions to find values to interpolate from
Number of smoothing iterations to run after the interpolation	ITERATIONS	[number] Default: 0	The number of 3x3 filter passes to run (0 or more) to smoothen the results of the interpolation.

下页继续

表 24.147 - 续上页

Label	名称	Type	Description
Do not use default validity mask for the input band	NO_MASK	[boolean] Default: False	Activates the user-defined validity mask
Validity mask	MASK_LAYER	[raster]	A raster layer that defines the areas to fill.
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Filled	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Filled	OUTPUT	[raster]	Output raster

Python code

Algorithm ID: gdal:fillnodata

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Grid (Data metrics)

Computes some data metrics using the specified window and output grid geometry.

This algorithm is derived from the [GDAL grid utility](#).

Default menu: *Raster* ► *Analysis*

参见:

[GDAL grid tutorial](#)

Parameters

Label	名称	Type	Description
Point layer	INPUT	[vector: point]	Input point vector layer
Data metric to use	METRIC	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- Minimum, minimum value found in grid node search ellipse • 1 --- Maximum, maximum value found in grid node search ellipse • 2 --- Range, a difference between the minimum and maximum values found in grid node search ellipse • 3 --- Count, a number of data points found in grid node search ellipse • 4 --- Average distance, an average distance between the grid node (center of the search ellipse) and all of the data points found in grid node search ellipse • 5 --- Average distance between points, an average distance between the data points found in grid node search ellipse. The distance between each pair of points within ellipse is calculated and average of all distances is set as a grid node value
The first radius of search ellipse	RADIUS_1	[number] Default: 0.0	The first radius (X axis if rotation angle is 0) of the search ellipse
The second radius of search ellipse	RADIUS_2	[number] Default: 0.0	The second radius (Y axis if rotation angle is 0) of the search ellipse
Angle of search ellipse rotation in degrees (counter clockwise)	ANGLE	[number] Default: 0.0	Angle of ellipse rotation in degrees. Ellipse rotated counter clockwise.
Minimum number of data points to use	MIN_POINTS	[number] Default: 0.0	Minimum number of data points to average. If less amount of points found the grid node considered empty and will be filled with NODATA marker.
Nodata	NODATA	[number] Default: 0.0	No data marker to fill empty points
Z value from field Optional	Z_FIELD	[tablefield: numeric]	Field for the interpolation
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options

下页继续

表 24.148 - 续上页

Label	名称	Type	Description
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Interpolated (data metrics)	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer with interpolated values. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Interpolated (data metrics)	OUTPUT	[raster]	Output raster with interpolated values

Python code

Algorithm ID: gdal:griddatametrics

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Grid (IDW with nearest neighbor searching)

Computes the Inverse Distance to a Power gridding combined to the nearest neighbor method. Ideal when a maximum number of data points to use is required.

This algorithm is derived from the [GDAL grid utility](#).

参见:

[GDAL grid tutorial](#)

Parameters

Label	名称	Type	Description
Point layer	INPUT	[vector: point]	Input point vector layer
Weighting power	POWER	[number] Default: 2.0	Weighting power
Smoothing	SMOOTHING	[number] Default: 0.0	Smoothing parameter
The radius of the search circle	RADIUS	[number] Default: 1.0	The radius of the search circle
Maximum number of data points to use	MAX_POINTS	[number] Default: 12	Do not search for more points than this number.
Minimum number of data points to use	MIN_POINTS	[number] Default: 0	Minimum number of data points to average. If less amount of points found the grid node considered empty and will be filled with NODATA marker.
Nodata	NODATA	[number] Default: 0.0	No data marker to fill empty points
Z value from field Optional	Z_FIELD	[tablefield: numeric]	Field for the interpolation
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Interpolated (IDW with NN search)	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer with interpolated values. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Interpolated (IDW with NN search)	OUTPUT	[raster]	Output raster with interpolated values

Python code

Algorithm ID: gdal:gridinversedistancenearestneighbor

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Grid (Inverse distance to a power)

The Inverse Distance to a Power gridding method is a weighted average interpolator.

You should supply the input arrays with the scattered data values including coordinates of every data point and output grid geometry. The function will compute interpolated value for the given position in output grid.

This algorithm is derived from the [GDAL grid utility](#).

Default menu: *Raster* ► *Analysis*

参见:

[GDAL grid tutorial](#)

Parameters

Label	名称	Type	Description
Point layer	INPUT	[vector: point]	Input point vector layer
Weighting power	POWER	[number] Default: 2.0	Weighting power
Smoothing	SMOOTHING	[number] Default: 0.0	Smoothing parameter
The first radius of search ellipse	RADIUS_1	[number] Default: 0.0	The first radius (X axis if rotation angle is 0) of the search ellipse
The second radius of search ellipse	RADIUS_2	[number] Default: 0.0	The second radius (Y axis if rotation angle is 0) of the search ellipse
Angle of search ellipse rotation in degrees (counter clockwise)	ANGLE	[number] Default: 0.0	Angle of ellipse rotation in degrees. Ellipse rotated counter clockwise.
Maximum number of data points to use	MAX_POINTS	[number] Default: 0	Do not search for more points than this number.

下页继续

表 24.150 - 续上页

Label	名称	Type	Description
Minimum number of data points to use	MIN_POINTS	[number] Default: 0	Minimum number of data points to average. If less amount of points found the grid node considered empty and will be filled with NODATA marker.
Nodata	NODATA	[number] Default: 0.0	No data marker to fill empty points
Z value from field Optional	Z_FIELD	[tablefield: numeric]	Field for the interpolation
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Interpolated (IDW)	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer with interpolated values. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Interpolated (IDW)	OUTPUT	[raster]	Output raster with interpolated values

Python code

Algorithm ID: gdal:gridinversedistance

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Grid (Linear)

The Linear method perform linear interpolation by computing a Delaunay triangulation of the point cloud, finding in which triangle of the triangulation the point is, and by doing linear interpolation from its barycentric coordinates within the triangle. If the point is not in any triangle, depending on the radius, the algorithm will use the value of the nearest point or the NODATA value.

This algorithm is derived from the [GDAL grid utility](#).

Parameters

Label	名称	Type	Description
Point layer	INPUT	[vector: point]	Input point vector layer
Search distance	RADIUS	[number] Default: -1.0	In case the point to be interpolated does not fit into a triangle of the Delaunay triangulation, use that maximum distance to search a nearest neighbour, or use nodata otherwise. If set to -1, the search distance is infinite. If set to 0, no data value will be used.
Nodata	NODATA	[number] Default: 0.0	No data marker to fill empty points
Z value from field Optional	Z_FIELD	[tablefield: numeric]	Field for the interpolation
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options

下页继续

表 24.151 - 续上页

Label	名称	Type	Description
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Interpolated (Linear)	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer with interpolated values. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Interpolated (Linear)	OUTPUT	[raster]	Output raster with interpolated values

Python code

Algorithm ID: gdal:gridlinear

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Grid (Moving average)

The Moving Average is a simple data averaging algorithm. It uses a moving window of elliptic form to search values and averages all data points within the window. Search ellipse can be rotated by specified angle, the center of ellipse located at the grid node. Also the minimum number of data points to average can be set, if there are not enough points in window, the grid node considered empty and will be filled with specified NODATA value.

This algorithm is derived from the [GDAL grid utility](#).

Default menu: *Raster ► Analysis*

参见:

[GDAL grid tutorial](#)

Parameters

Label	名称	Type	Description
Point layer	INPUT	[vector: point]	Input point vector layer
The first radius of search ellipse	RADIUS_1	[number] Default: 0.0	The first radius (X axis if rotation angle is 0) of the search ellipse
The second radius of search ellipse	RADIUS_2	[number] Default: 0.0	The second radius (Y axis if rotation angle is 0) of the search ellipse
Angle of search ellipse rotation in degrees (counter clockwise)	ANGLE	[number] Default: 0.0	Angle of ellipse rotation in degrees. Ellipse rotated counter clockwise.
Minimum number of data points to use	MIN_POINTS	[number] Default: 0.0	Minimum number of data points to average. If less amount of points found the grid node considered empty and will be filled with NODATA marker.
Nodata	NODATA	[number] Default: 0.0	No data marker to fill empty points
Z value from field Optional	Z_FIELD	[tablefield: numeric]	Field for the interpolation
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Interpolated (moving average)	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Interpolated (moving average)	OUTPUT	[raster]	Output raster with interpolated values

Python code

Algorithm ID: gdal:gridaverage

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Grid (Nearest neighbor)

The Nearest Neighbor method doesn't perform any interpolation or smoothing, it just takes the value of nearest point found in grid node search ellipse and returns it as a result. If there are no points found, the specified NODATA value will be returned.

This algorithm is derived from the [GDAL grid utility](#).

Default menu: *Raster ► Analysis*

参见:

[GDAL grid tutorial](#)

Parameters

Label	名称	Type	Description
Point layer	INPUT	[vector: point]	Input point vector layer
The first radius of search ellipse	RADIUS_1	[number] Default: 0.0	The first radius (X axis if rotation angle is 0) of the search ellipse
The second radius of search ellipse	RADIUS_2	[number] Default: 0.0	The second radius (Y axis if rotation angle is 0) of the search ellipse
Angle of search ellipse rotation in degrees (counter clockwise)	ANGLE	[number] Default: 0.0	Angle of ellipse rotation in degrees. Ellipse rotated counter clockwise.
Nodata	NODATA	[number] Default: 0.0	No data marker to fill empty points
Z value from field Optional	Z_FIELD	[tablefield: numeric]	Field for the interpolation
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().

下页继续

表 24.153 - 续上页

Label	名称	Type	Description
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Interpolated (Nearest neighbour)	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer with interpolated values. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Interpolated (Nearest neighbour)	OUTPUT	[raster]	Output raster with interpolated values

Python code

Algorithm ID: gdal:gridnearestneighbor

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Hillshade

Outputs a raster with a nice shaded relief effect. It's very useful for visualizing the terrain. You can optionally specify the azimuth and altitude of the light source, a vertical exaggeration factor and a scaling factor to account for differences between vertical and horizontal units.

This algorithm is derived from the [GDAL DEM utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input Elevation raster layer
Band number	BAND	[raster band] Default: 1	Band containing the elevation information
Z factor (vertical exaggeration)	Z_FACTOR	[number] Default: 1.0	The factor exaggerates the height of the output elevation raster
Scale (ratio of vert. units to horiz.)	SCALE	[number] Default: 1.0	The ratio of vertical units to horizontal units
Azimuth of the light	AZIMUTH	[number] Default: 315.0	Defines the azimuth of the light shining on the elevation raster in degrees. If it comes from the top of the raster the value is 0, if it comes from the east it is 90 a.s.o.
Altitude of the light	ALTITUDE	[number] Default: 45.0	Defines the altitude of the light, in degrees. 90 if the light comes from above the elevation raster, 0 if it is raking light.
Compute edges	COMPUTE_EDGES	[boolean] Default: False	Generates edges from the elevation raster
Use Zevenbergen&Thorne formula (instead of the Horn's one)	ZEVENBERGEN	[boolean] Default: False	Activates Zevenbergen&Thorne formula for smooth landscapes
Combined shading	COMBINED	[boolean] Default: False	
Multidirectional shading	MULTIDIRECTIONAL	[boolean] Default: False	
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see GDAL driver options section). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Hillshade	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer with interpolated values. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Hillshade	OUTPUT	[raster]	Output raster with interpolated values

Python code

Algorithm ID: gdal:hillshade

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Near black

Converts nearly black/white borders to black.

This algorithm will scan an image and try to set all pixels that are nearly or exactly black, white or one or more custom colors around the collar to black or white. This is often used to "fix up" lossy compressed airphotos so that color pixels can be treated as transparent when mosaicking.

This algorithm is derived from the [GDAL nearblack utility](#).

Default menu: *Raster* ► *Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input Elevation raster layer
How far from black (white)	NEAR	[number] Default: 15	Select how far from black, white or custom colors the pixel values can be and still considered near black, white or custom color.
Search for nearly white pixels instead of nearly black	WHITE	[boolean] Default: False	Search for nearly white (255) pixels instead of nearly black pixels
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Nearblack	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Nearblack	OUTPUT	[raster]	Output raster

Python code

Algorithm ID: gdal:nearblack

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Proximity (raster distance)

Generates a raster proximity map indicating the distance from the center of each pixel to the center of the nearest pixel identified as a target pixel. Target pixels are those in the source raster for which the raster pixel value is in the set of target pixel values.

This algorithm is derived from the [GDAL proximity utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input Elevation raster layer
Band number	BAND	[raster band] Default: 1	Band containing the elevation information
A list of pixel values in the source image to be considered target pixels Optional	VALUES	[string] Default: ""	A list of target pixel values in the source image to be considered target pixels. If not specified, all non-zero pixels will be considered target pixels.
Distance units	UNITS	[enumeration] Default: 1	Indicate whether distances generated should be in pixel or georeferenced coordinates. One of: <ul style="list-style-type: none"> 0 --- Georeferenced coordinates 1 --- Pixel coordinates
The maximum distance to be generated Optional	MAX_DISTANCE	[number] Default: 0.0	The maximum distance to be generated. The nodata value will be used for pixels beyond this distance. If a nodata value is not provided, the output band will be queried for its nodata value. If the output band does not have a nodata value, then the value 65535 will be used. Distance is interpreted according to the value of <i>Distance units</i> .

下页继续

表 24.156 - 续上页

Label	名称	Type	Description
Value to be applied to all pixels that are within the maxdist of target pixels Optional	REPLACE	[number] Default: 0.0	Specify a value to be applied to all pixels that are closer than the maximum distance from target pixels (including the target pixels) instead of a distance value.
Nodata value to use for the destination proximity raster Optional	NODATA	[number] Default: 0.0	Specify the nodata value to use for the output raster
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Proximity map	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Proximity map	OUTPUT	[raster]	Output raster

Python code

Algorithm ID: gdal:proximity

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Roughness

Outputs a single-band raster with values computed from the elevation. Roughness is the degree of irregularity of the surface. It's calculated by the largest inter-cell difference of a central pixel and its surrounding cell. The determination of the roughness plays a role in the analysis of terrain elevation data, it's useful for calculations of the river morphology, in climatology and physical geography in general.

This algorithm is derived from the [GDAL DEM utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input elevation raster layer
Band number	BAND	[raster band] Default: 1	The number of the band to use as elevation
Compute edges	COMPUTE_EDGES	[boolean] Default: False	Generates edges from the elevation raster
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Roughness	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Roughness	OUTPUT	[raster]	Single-band output roughness raster. The value -9999 is used as nodata value.

Python code

Algorithm ID: gdal:roughness

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Sieve

Removes raster polygons smaller than a provided threshold size (in pixels) and replaces them with the pixel value of the largest neighbour polygon. It is useful if you have a large amount of small areas on your raster map.

This algorithm is derived from the [GDAL sieve utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input elevation raster layer
Threshold	THRESHOLD	[number] Default: 10	Only raster polygons smaller than this size will be removed
Use 8-connectedness	EIGHT_CONNECTED	[boolean] Default: False	Use eight connectedness instead of four connectedness
Do not use the default validity mask for the input band	NO_MASK	[boolean] Default: False	
Validity mask Optional	MASK_LAYER	[raster]	Validity mask to use instead of the default
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Sieved	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Sieved	OUTPUT	[raster]	Output raster layer.

Python code

Algorithm ID: gdal:sieve

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Slope

Generates a slope map from any GDAL-supported elevation raster. Slope is the angle of inclination to the horizontal. You have the option of specifying the type of slope value you want: degrees or percent slope.

This algorithm is derived from the [GDAL DEM utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input Elevation raster layer
Band number	BAND	[raster band] Default: 1	Band containing the elevation information
Ratio of vertical units to horizontal	SCALE	[number] Default: 1.0	The ratio of vertical units to horizontal units
Slope expressed as percent (instead of degrees)	AS_PERCENT	[boolean] Default: False	Express slope as percent instead of degrees
Compute edges	COMPUTE_EDGES	[boolean] Default: False	Generates edges from the elevation raster
Use Zevenbergen&Thorne formula (instead of the Horn's one)	ZEVENBERGEN	[boolean] Default: False	Activates Zevenbergen&Thorne formula for smooth landscapes
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Slope	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Slope	OUTPUT	[raster]	Output raster

Python code

Algorithm ID: gdal:slope

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Terrain Ruggedness Index (TRI)

Outputs a single-band raster with values computed from the elevation. TRI stands for Terrain Ruggedness Index, which is defined as the mean difference between a central pixel and its surrounding cells.

This algorithm is derived from the [GDAL DEM utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input elevation raster layer
Band number	BAND	[raster band] Default: 1	The number of the band to use as elevation
Compute edges	COMPUTE_EDGES	[boolean] Default: False	Generates edges from the elevation raster
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Terrain Ruggedness Index	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Terrain Ruggedness Index	OUTPUT	[raster]	Output ruggedness raster. The value -9999 is used as nodata value.

Python code

Algorithm ID: gdal:triterrainruggednessindex

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Topographic Position Index (TPI)

Outputs a single-band raster with values computed from the elevation. TPI stands for Topographic Position Index, which is defined as the difference between a central pixel and the mean of its surrounding cells.

This algorithm is derived from the [GDAL DEM utility](#).

Default menu: *Raster ► Analysis*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input elevation raster layer
Band number	BAND	[raster band] Default: 1	The number of the band to use for elevation values
Compute edges	COMPUTE_EDGES	[boolean] Default: False	Generates edges from the elevation raster
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Terrain Ruggedness Index	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Terrain Ruggedness Index	OUTPUT	[raster]	Output raster.

Python code

Algorithm ID: gdal:tpitopographicpositionindex

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.2.2 Raster conversion

gdal2xyz

Converts raster data to XYZ ASCII file format.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Raster layer to convert
Band number	BAND	[raster band] Default: The first band of the input layer	If the raster is multiband, choose the band you want to convert
Output comma-separated values	CSV	[boolean] Default: False	Sets whether the output file should be of type comma-separated values (csv).
XYZ ASCII file	OUTPUT	[file] Default: [Save to temporary file]	Specification of the output file. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
XYZ ASCII file	INPUT	[table]	Table file containing the values exported from the raster band.

Python code

Algorithm ID: gdal:gdal2xyz

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

PCT to RGB

Converts an 8 bit paletted image to a 24 bit RGB. It will convert a pseudocolor band from the input file to an RGB file of the desired format.

This algorithm is derived from the GDAL `pct2rgb` utility.

Default menu: *Raster ► Conversion*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input 8 bit raster image
Band number	BAND	[raster band] Default: The first band of the input layer	If the raster is multiband, choose the band you want to convert
Generate a RGBA file	RGBA	[boolean] Default: False	Sets whether the output file should be of type RGBA.
PCT to RGB	OUTPUT	[file] Default: [Save to temporary file]	Specification of the output file. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
PCT to RGB	OUTPUT	[raster]	24 bit RGB raster image

Python code

Algorithm ID: gdal:pcttorgb

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Polygonize (raster to vector)

Creates vector polygons for all connected regions of pixels in the raster sharing a common pixel value. Each polygon is created with an attribute indicating the pixel value of that polygon.

This algorithm is derived from the [GDAL polygonize utility](#).

Default menu: *Raster ► Conversion*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Band number	BAND	[raster band] Default: The first band of the input layer	If the raster is multiband, choose the band you want to use
Name of the field to create	FIELD	[string] Default: 'DN'	Specify the field name for the attributes of the connected regions.
Use 8-connectedness	EIGHT_CONNECTED	[boolean] Default: False	If not set, raster cells must have a common border to be considered connected (<i>4-connected</i>). If set, touching raster cells are also considered connected (<i>8-connected</i>).
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Vectorized	OUTPUT	[vector: polygon] Default: [Save to temporary file]	Specification of the output (polygon) vector layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Vectorized	OUTPUT	[vector: polygon]	Output vector layer

Python code

Algorithm ID: gdal:polygonize

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Rearrange bands

Creates a new raster using selected band(s) from a given raster layer. The algorithm also makes it possible to reorder the bands for the newly-created raster.

This algorithm is derived from the [GDAL translate utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Selected band(s)	BANDS	[raster band] [list] Default: None	Ordered list of the bands to use to create the new raster
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Output data type	DATA_TYPE	[enumeration] Default: 0	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Use Input Layer Data Type • 1 --- Byte • 2 --- Int16 • 3 --- UInt16 • 4 --- UInt32 • 5 --- Int32 • 6 --- Float32 • 7 --- Float64 • 8 --- CInt16 • 9 --- CInt32 • 10 --- CFloat32 • 11 --- CFloat64
Converted	OUTPUT	[raster] Default: Save to temporary file	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Converted	OUTPUT	[raster]	Output raster layer with rearranged bands.

Python code

Algorithm ID: gdal:rearrange_bands

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

RGB to PCT

Converts a 24 bit RGB image into a 8 bit paletted. Computes an optimal pseudo-color table for the given RGB-image using a median cut algorithm on a downsampled RGB histogram. Then it converts the image into a pseudo-colored image using the color table. This conversion utilizes Floyd-Steinberg dithering (error diffusion) to maximize output image visual quality.

If you want to classify a raster map and want to reduce the number of classes it can be helpful to downsample your image with this algorithm before.

This algorithm is derived from the GDAL `rgb2pct` utility.

Default menu: *Raster* ► *Conversion*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input (RGB) raster layer
Number of colors	N_COLORS	[number] Default: 2	The number of colors the resulting image will contain. A value from 2-256 is possible.
RGB to PCT	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
RGB to PCT	OUTPUT	[raster]	Output raster layer.

Python code

Algorithm ID: gdal:rgbtopct

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Translate (convert format)

Converts raster data between different formats.

This algorithm is derived from the [GDAL translate utility](#).

Default menu: *Raster ► Conversion*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Override the projection of the output file Optional	TARGET_CRS	[crs]	Specify a projection for the output file
Assign a specified nodata value to output bands Optional	NODATA	[number] Default: Not set	Defines the value to use for nodata in the output raster
Copy all sub-datasets of this file to individual output files	COPY_SUBDATASETS	[boolean] Default: False	Create individual files for subdatasets
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see GDAL driver options section). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Output data type	DATA_TYPE	[enumeration] Default: 0	Defines the data type of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Use Input Layer Data Type • 1 --- Byte • 2 --- Int16 • 3 --- UInt16 • 4 --- UInt32 • 5 --- Int32 • 6 --- Float32 • 7 --- Float64 • 8 --- CInt16 • 9 --- CInt32 • 10 --- CFloat32 • 11 --- CFloat64
Converted	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output (translated) raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Converted	OUTPUT	[raster]	Output (translated) raster layer.

Python code

Algorithm ID: gdal:translate

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.2.3 Raster extraction

Clip raster by extent

Clips any GDAL-supported raster file to a given extent.

This algorithm is derived from the [GDAL warp utility](#).

Default menu: *Raster* ► *Extraction*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	The input raster
Clipping extent	EXTENT	[extent]	Extent that should be used for the output raster. Only pixels within the specified bounding box will be included in the output.
Assign a specified nodata value to output bands Optional	NODATA	[number] Default: None	Defines a value that should be inserted for the nodata values in the output raster
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().

下页继续

表 24.168 – 续上页

Label	名称	Type	Description
Output data type	DATA_TYPE	[enumeration] Default: 0	Defines the format of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Use Input Layer Data Type • 1 --- Byte • 2 --- Int16 • 3 --- UInt16 • 4 --- UInt32 • 5 --- Int32 • 6 --- Float32 • 7 --- Float64 • 8 --- CInt16 • 9 --- CInt32 • 10 --- CFloat32 • 11 --- CFloat64
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Clipped (extent)	OUTPUT	[raster] Default: '[Save to temporary file]'	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here

Outputs

Label	名称	Type	Description
Clipped (extent)	OUTPUT	[raster]	Output raster layer clipped by the given extent

Python code

Algorithm ID: gdal:cliprasterbyextent

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Clip raster by mask layer

Clips any GDAL-supported raster by a vector mask layer.

This algorithm is derived from the [GDAL warp utility](#).

Default menu: *Raster* ► *Extraction*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	The input raster
Mask layer	MASK	[vector: polygon]	Vector mask for clipping the raster
Source CRS	SOURCE_CRS	[crs]	Set the coordinate reference to use for the input raster
Target CRS	TARGET_CRS	[crs]	Set the coordinate reference to use for the mask layer
Assign a specified nodata value to output bands Optional	NODATA	[number] Default: None	Defines a value that should be inserted for the nodata values in the output raster
Create an output alpha band	ALPHA_BAND	[boolean] Default: False	Creates an alpha band for the result. The alpha band then includes the transparency values of the pixels.
Match the extent of the clipped raster to the extent of the mask layer	CROP_TO_OUTLINE	[boolean] Default: True	Applies the vector layer extent to the output raster if checked.
Keep resolution of input raster	KEEP_RESOLUTION	[boolean] Default: False	The resolution of the output raster will not be changed
Set output file resolution	SET_RESOLUTION	[boolean] Default: False	Shall the output resolution (cell size) be specified
X Resolution to output bands Optional	X_RESOLUTION	[number] Default: None	The width of the cells in the output raster
Y Resolution to output band Optional	Y_RESOLUTION	[number] Default: None	The height of the cells in the output raster
Use multithreaded warping implementation	MULTITHREADING	[boolean] Default: False	Two threads will be used to process chunks of image and perform input/output operation simultaneously. Note that computation is not multithreaded itself.
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see GDAL driver options section). For Batch Process: separate multiple options with a pipe character ().

下页继续

表 24.169 – 续上页

Label	名称	Type	Description
Output data type	DATA_TYPE	[enumeration] Default: 0	Defines the format of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Use Input Layer Data Type • 1 --- Byte • 2 --- Int16 • 3 --- UInt16 • 4 --- UInt32 • 5 --- Int32 • 6 --- Float32 • 7 --- Float64 • 8 --- CInt16 • 9 --- CInt32 • 10 --- CFloat32 • 11 --- CFloat64
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options
Clipped (mask)	OUTPUT	[raster] Default: '[Save to temporary file]'	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here

Outputs

Label	名称	Type	Description
Clipped (mask)	OUTPUT	[raster]	Output raster layer clipped by the vector layer

Python code

Algorithm ID: gdal:cliprasterbymasklayer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Contour

Extracts contour lines from any GDAL-supported elevation raster.

This algorithm is derived from the [GDAL contour utility](#).

Default menu: *Raster ► Extraction*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster
Band number	BAND	[raster band] Default: 1	Raster band to create the contours from
Interval between contour lines	INTERVAL	[number] Default: 10.0	Defines the interval between the contour lines in the given units of the elevation raster (minimum value 0)
Attribute name (if not set, no elevation attribute is attached) Optional	FIELD_NAME	[string] Default: 'ELEV'	Provides a name for the attribute in which to put the elevation.
Offset from zero relative to which to interpret intervals Optional	OFFSET	[number] Default: 0.0	
Produce 3D vector	CREATE_3D	[boolean] Default: False	Forces production of 3D vectors instead of 2D. Includes elevation at every vertex.
Treat all raster values as valid	IGNORE_NODATA	[boolean] Default: False	Ignores any nodata values in the dataset.
Input pixel value to treat as "no-data" Optional	NODATA	[number] Default: None	Defines a value that should be inserted for the nodata values in the output raster
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options. Refer to the corresponding GDAL utility documentation.
Contours	OUTPUT	[vector: line] Default: '[Save to temporary file]'	Specification of the output vector layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Contours	OUTPUT	[vector: line]	Output vector layer with contour lines

Python code

Algorithm ID: gdal:contour

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Contour Polygons

Extracts contour polygons from any GDAL-supported elevation raster.

This algorithm is derived from the [GDAL contour utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster
Band number	BAND	[raster band] Default: 1	Raster band to create the contours from
Interval between contour lines	INTERVAL	[number] Default: 10.0	Defines the interval between the contour lines in the given units of the elevation raster (minimum value 0)
Offset from zero relative to which to interpret intervals Optional	OFFSET	[number] Default: 0.0	
Attribute name for minimum elevation of contour polygon Optional	FIELD_NAME_MIN	[string] Default: 'ELEV_MIN'	Provides a name for the attribute in which to put the minimum elevation of contour polygon. If not provided no minimum elevation attribute is attached.
Attribute name for maximum elevation of contour polygon Optional	FIELD_NAME_MAX	[string] Default: 'ELEV_MAX'	Provides a name for the attribute in which to put the maximum elevation of contour polygon. If not provided no maximum elevation attribute is attached.
Produce 3D vector	CREATE_3D	[boolean] Default: False	Forces production of 3D vectors instead of 2D. Includes elevation at every vertex.
Treat all raster values as valid	IGNORE_NODATA	[boolean] Default: False	Ignores any nodata values in the dataset.
Input pixel value to treat as "no-data" Optional	NODATA	[number] Default: None	Defines a value that should be inserted for the nodata values in the output raster

下页继续

表 24.171 - 续上页

Label	名称	Type	Description
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options. Refer to the corresponding GDAL utility documentation.
Contours	OUTPUT	[vector: polygon] Default: '[Save to temporary file]'	Specification of the output vector layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Contours	OUTPUT	[vector: polygon]	Output vector layer with contour polygons

Python code

Algorithm ID: gdal:contour_polygon

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.2.4 Raster miscellaneous

Build overviews (pyramids)

To speed up rendering time of raster layers overviews (pyramids) can be created. Overviews are lower resolution copies of the data which QGIS uses depending of the level of zoom.

This algorithm is derived from the [GDAL addo utility](#).

Default menu: *Raster ► Miscellaneous*

Parameters

Basic parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Remove all existing overviews	CLEAN	[boolean] Default: False	Removes existing overviews from the raster. By default these are not removed.

Advanced parameters

Label	名称	Type	Description
Overview levels	LEVELS	[string] Default: '2 4 8 16'	Defines the number of overview levels calculated by the original resolution of the input raster layer. By default 4 levels will be taken into consideration.
Resampling method Optional	RESAMPLING	[enumeration] Default: 0	Calculates the overviews with a defined resampling method. Possible resampling methods are: <ul style="list-style-type: none"> • 0 -- Nearest Neighbour (nearest) • 1 -- Average (average) • 2 -- Gaussian (gauss) • 3 -- Cubic Convolution (cubic) • 4 -- B-Spline Convolution (cubicspline) • 5 -- Lanczos Windowed Sinc (lanczos) • 6 -- Average MP (average_mp) • 7 -- Average in Mag/Phase Space (average_magphase) • 8 -- Mode (mode)
Overviews format Optional	FORMAT	[enumeration] Default: 0	The overviews can be stored internally, or externally as GTiff or ERDAS Imagine file. By default the overviews are stored in the output raster. Possible formats methods are: <ul style="list-style-type: none"> • 0 -- Internal (if possible) • 1 -- External (GTiff .ovr) • 2 -- External (ERDAS Imagine .aux)
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Pyramidized	OUTPUT	[raster]	Output raster layer with overviews

Python code

Algorithm ID: gdal:overviews

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Build virtual raster

Builds a VRT (Virtual Dataset) that is a mosaic of the list of input GDAL-supported rasters. With a mosaic you can merge several raster files.

This algorithm is derived from the [GDAL buildvrt utility](#).

Default menu: *Raster ► Miscellaneous*

Parameters

Basic parameters

Label	名称	Type	Description
Input layers	INPUT	[raster] [list]	GDAL-supported raster layers.
Resolution	RESOLUTION	[enumeration] Default: 0	The output resolution of the mosaic. By default the average resolution of the raster files will be chosen. Options: <ul style="list-style-type: none"> • 0 --- Average (average) • 1 --- Highest (highest) • 2 --- Lowest (lowest)
Place each input file into a separate band	SEPARATE	[boolean] Default: False	With 'True' you can define that each raster file goes into a separated stacked band in the VRT band.
Allow projection difference	PROJ_DIFFERENCE	[boolean] Default: False	Allows that the output bands have different projections derived from the projection of the input raster layers.
Virtual	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Add alpha mask band to VRT when source raster has none	ADD_ALPHA	[boolean] Default: False	Adds an alpha mask band to the VRT when the source raster has none.
Override projection for the output file Optional	ASSIGN_CRIS	[crs] Default: None	Overrides the projection for the output file. No reprojection is done.
Resampling algorithm	RESAMPLING	[enumeration] Default: 0	The resampling algorithm to be used Options: <ul style="list-style-type: none"> • 0 --- Nearest Neighbour (nearest) • 1 --- Bilinear (bilinear) • 2 --- Cubic Convolution (cubic) • 3 --- B-Spline Convolution (cubicspline) • 4 --- Lanczos Windowed Sinc (lanczos) • 5 --- Average (average) • 6 --- Mode (mode)
Nodata value(s) for input bands (space separated) Optional	SRC_NODATA	[string] Default: None	Space separated Nodata value(s) for input band(s)
Additional command-line parameters	EXTRA	[string] Default: None	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Virtual	OUTPUT	[raster]	Output raster layer

Python code

Algorithm ID: gdal:buildvirtualraster

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

gdal2tiles

Generates a directory with small tiles and metadata, following the [OSGeo Tile Map Service Specification](#). See also the [OpenGIS Web Map Tile Service Implementation Standard](#). Simple web pages with viewers based on Google Maps, OpenLayers and Leaflet are generated as well. To explore your maps on-line in the web browser, you only need to upload the generated directory onto a web server.

This algorithm also creates the necessary metadata for Google Earth (KML SuperOverlay), in case the supplied map uses EPSG:4326 projection.

ESRI world files and embedded georeferencing is used during tile generation, but you can publish a picture without proper georeferencing too.

This algorithm is derived from the [GDAL gdal2tiles utility](#).

Parameters

Basic parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	GDAL-supported raster layer.
Tile cutting profile	PROFILE	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> 0 --- Mercator (mercator) 1 --- Geodetic (geodetic) 2 --- Raster (raster)
Zoom levels to render Optional	ZOOM	[string] Default: ”	
Web viewer to generate	VIEWER	[enumerate] Default: 0	One of: <ul style="list-style-type: none"> 0 --- All (all) 1 --- GoogleMaps (google) 2 --- OpenLayers (openlayers) 3 --- Leaflet (leaflet) 4 --- None (none)
Title of the map Optional	TITLE	[string] Default: ”	
Copyright of the map	COPYRIGHT	[string] Default: ”	
Output directory	OUTPUT	[folder] Default: [Save to temporary folder]	Specify the output folder for the tiles. One of: <ul style="list-style-type: none"> Save to a Temporary Directory Save to Directory

Advanced parameters

Label	名称	Type	Description
Resampling method	RESAMPLING	[enumeration] Default: 0	The resampling algorithm to be used Options: <ul style="list-style-type: none"> • 0 --- Average (average) • 1 --- Nearest neighbour (near) • 2 --- Bilinear (bilinear) • 3 --- Cubic (cubic) • 4 --- Cubic spline (cubicspline) • 5 --- Lanczos Windowed sinc (lanczos) • 6 --- Antialias (antialias)
The spatial reference system used for the source input data Optional	SOURCE_CRS	[crs] Default: None	
Transparency value to assign to the input data Optional	NODATA	[number] Default: 0.0	
URL address where the generated tiles are going to be published Optional	URL	[string] Default: ""	
Google Maps API key (http://code.google.com/apis/maps/signup.html) Optional	GOOGLE_KEY	[string] Default: ""	Your Google maps API key.
Bing Maps API key (https://www.bingmapsportal.com/) Optional	BING_KEY	[string] Default: ""	Your Bing maps API key.
Generate only missing files	RESUME	[boolean] Default: False	
Generate KML for Google Earth	KML	[boolean] Default: False	
Avoid automatic generation of KML files for EPSG:4326	NO_KML	[boolean] Default: False	

Outputs

Label	名称	Type	Description
Output directory	OUTPUT	[folder]	The output folder (for the tiles)

Python code

Algorithm ID: gdal:gdal2tiles

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Merge

Merges raster files in a simple way. Here you can use a pseudocolor table from an input raster and define the output raster type. All the images must be in the same coordinate system.

This algorithm is derived from the [GDAL merge utility](#).

Default menu: *Raster ► Miscellaneous*

Parameters

Basic parameters

Label	名称	Type	Description
Input layers	INPUT	[raster] [list]	Input raster layers
Grab pseudocolor table from first layer	PCT	[boolean] Default: False	The pseudocolor table from the first layer will be used for the coloring
Place each input file into a separate band	SEPARATE	[boolean] Default: False	Place each input file into a separate band
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the format of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64

下页继续

表 24.175 - 续上页

Label	名称	Type	Description
Merged	OUTPUT	[raster] Default: [Save to temporary file]	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Input pixel value to treat as "no-data" Optional	NODATA_INPUT	[number] Default: None	Ignores pixels from files being merged in with this pixel value
Assign specified "nodata" value to output Optional	NODATA_OUTPUT	[number] Default: None	Assigns the specified nodata value to output bands.
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters	EXTRA	[string] Default: None	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Merged	OUTPUT	[raster]	Output raster layer

Python code

Algorithm ID: gdal:merge

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Pansharpening

Performs a pan-sharpening operation. It can create a "classic" output dataset (such as GeoTIFF), or a VRT dataset describing the pan-sharpening operation.

See [GDAL Pansharpen](#).

Parameters

Basic parameters

Label	名称	Type	Description
Spectral dataset	SPECTRAL	[raster]	Input (spectral) raster layer
Panchromatic dataset	PANCHROMATIC	[raster]	Input (panchromatic) raster layer
Output	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output (sharpened) raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Resampling algorithm	RESAMPLING	[enumeration] Default: 2	The resampling algorithm to be used Options: <ul style="list-style-type: none"> • 0 --- Nearest Neighbour (nearest) • 1 --- Bilinear (bilinear) • 2 --- Cubic (cubic) • 3 --- Cubic Spline (cubicspline) • 4 --- Lanczos Windowed Sinc (lanczos) • 5 --- Average (average)
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see GDAL driver options section). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Output	OUTPUT	[raster]	Output (sharpened) raster layer

Python code

Algorithm ID: gdal:pansharp

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster calculator

Command line raster calculator with numpy syntax. Use any basic arithmetic supported by numpy arrays, such as +, -, *, and / along with logical operators, such as >. Note that all input rasters must have the same dimensions, but no projection checking is performed.

See the [GDAL Raster Calculator utility docs](#).

参见:

Raster calculator

Parameters

Basic parameters

Label	名称	Type	Description
Input layer A	INPUT_A	[raster]	First input raster layer (mandatory)
Number of raster band for A	BAND_A	[raster band]	Band for input layer A (mandatory)
Input layer B Optional	INPUT_B	[raster] Default: None	Second input raster layer
Number of raster band for B Optional	BAND_B	[raster band]	Band for input layer B
Input layer C Optional	INPUT_C	[raster] Default: None	Third input raster layer
Number of raster band for C Optional	BAND_C	[raster band]	Band for input layer C
Input layer D Optional	INPUT_D	[raster] Default: None	Fourth input raster layer
Number of raster band for D Optional	BAND_D	[raster band]	Band for input layer D
Input layer E Optional	INPUT_E	[raster] Default: None	Fifth input raster layer

下页继续

表 24.177 - 续上页

Label	名称	Type	Description
Number of raster band for E Optional	BAND_E	[raster band]	Band for input layer E
Input layer F Optional	INPUT_F	[raster]	Sixth input raster layer
Number of raster band for F Optional	BAND_F	[raster band] Default: None	Band for input layer F
Calculation in gdalnumeric syntax using +/-* or any numpy array functions (i.e. logical_and())	FORMULA	[string] Default: ""	The calculation formula. Examples: <ul style="list-style-type: none"> • $A * (A > 0)$ --- outputs the value of the raster A if the value of A is greater than 0. If not, outputs 0. • $A * (A > 0 \text{ and } A > B)$ --- outputs the value of A if that value is bigger than 0 and bigger than the value of B. If not, outputs 0. • $A * \text{logical_or}(A \leq 177, A \geq 185)$ --- outputs the value of A if $A \leq 177$ or $A \geq 185$. If not, outputs 0. • $\text{sqrt}(A^2 + B^2)$ --- Outputs the square root of the sum of the value of A squared and the value of B squared.
Set output nodata value Optional	NO_DATA	[number] Default: None	Value to use for nodata
Output raster type	RTYPE	[enumeration] Default: 5	Defines the format of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64
Calculated	OUTPUT	[raster] Default: [Save to temporary file]	Specify the output (calculated) raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters Optional	EXTRA	[string] Default: ""	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Calculated	OUTPUT	[raster]	Output (calculated) raster layer

Python code

Algorithm ID: gdal:rastercalculator

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Raster information

The gdalinfo program lists various information about a GDAL supported raster dataset.

This algorithm is derived from the [GDAL info utility](#).

Default menu: *Raster ► Miscellaneous*

Parameters

Basic parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer
Force computation of the actual min/max values for each band	MIN_MAX	[boolean] Default: False	Forces computation of the actual min/max values for each band in the dataset
Read and display image statistics (force computation if necessary)	STATS	[boolean] Default: False	Reads and displays image statistics. Forces computation if no statistics are stored in an image.

下页继续

表 24.179 – 续上页

Label	名称	Type	Description
Suppress GCP info	NO_GCP	[boolean] Default: False	Suppresses ground control points list printing. It may be useful for datasets with huge amount of GCPs, such as L1B AVHRR or HDF4 MODIS which contain thousands of them.
Suppress meta-data info	NO_METADATA	[boolean] Default: False	Suppresses metadata printing. Some datasets may contain a lot of metadata strings.
Layer information	OUTPUT	[html] Default: [Save to temporary file]	Specify the HTML file for output. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Layer information	OUTPUT	[html]	The HTML file containing information about the input raster layer

Python code

Algorithm ID: gdal:gdalinfo

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Retile

Retiles a set of input tiles. All the input tiles must be georeferenced in the same coordinate system and have a matching number of bands. Optionally pyramid levels are generated.

This algorithm is derived from the [GDAL Retile utility](#).

Parameters

Basic parameters

Label	名称	Type	Description
Input files	INPUT	[raster] [list]	The input raster files
Tile width	TILE_SIZE_X	[number] Default: 256	Width of the tiles in pixels (minimum 0)
Tile height	TILE_SIZE_Y	[number] Default: 256	Height of the tiles in pixels (minimum 0)
Overlap in pixels between consecutive tiles	OVERLAP	[number] Default: 0	
Number of pyramid levels to build	LEVELS	[number] Default: 1	Minimum: 0
Output directory	OUTPUT	[folder] Default: [Save to temporary folder]	Specify the output folder for the tiles. One of: <ul style="list-style-type: none"> • Save to a Temporary Directory • Save to Directory
CSV file containing the tile(s) georeferencing information	OUTPUT_CSV	[file] Default: [Skip output]	Specify the output file for the tiles. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Source coordinate reference system Optional	SOURCE_CRIS	[crs] Default: None	
Resampling method	RESAMPLING	[enumeration] Default: 0	The resampling algorithm to be used Options: <ul style="list-style-type: none"> • 0 --- Nearest Neighbour (<i>nearest</i>) • 1 --- Bilinear (<i>bilinear</i>) • 2 --- Cubic (<i>cubic</i>) • 3 --- Cubic Spline (<i>cubicspline</i>) • 4 --- Lanczos Windowed Sinc (<i>lanczos</i>)
Column delimiter used in the CSV file Optional	DELIMITER	[string] Default: `;`	Delimiter to use in the CSV file containing the tile(s) georeferencing information
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().

下页继续

表 24.182 – 续上页

Label	名称	Type	Description
Additional command-line parameters Optional	EXTRA	[string] Default: ""	Add extra GDAL command line options
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the format of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Build only the pyramids	ONLY_PYRAMIDS	[boolean] Default: False	
Use separate directory for each tile row	DIR_FOR_ROW	[boolean] Default: False	

Outputs

Label	名称	Type	Description
Output directory	OUTPUT	[folder]	The output folder for the tiles.
CSV file containing the tile(s) georeferencing information	OUTPUT_CSV	[file]	The CSV file with georeferencing information for the tiles.

Python code

Algorithm ID: gdal:retiler

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Tile index

Builds a vector layer with a record for each input raster file, an attribute containing the filename, and a polygon geometry outlining the raster. This output is suitable for use with MapServer as a raster tileindex.

This algorithm is derived from the [GDAL Tile Index utility](#).

Default menu: *Raster ► Miscellaneous*

Parameters

Basic parameters

Label	名称	Type	Description
Input files	LAYERS	[raster] [list]	The input raster files. Can be multiple files.
Field name to hold the file path to the indexed rasters	PATH_FIELD_NAME Optional	[string] Default: 'location'	The output field name to hold the file path/location to the indexed rasters.
Store absolute path to the indexed rasters	ABSOLUTE_PATH	[boolean] Default: False	Set whether the absolute path to the raster files is stored in the tile index file. By default the raster filenames will be put in the file exactly as they are specified in the command.
Skip files with different projection reference	PROJ_DIFFERENCE	[boolean] Default: False	Only files with same projection as files already inserted in the tile index will be inserted. Default does not check projection and accepts all inputs.
Tile index	OUTPUT	[vector: polygon] Default: [Save to temporary file]	Specify the polygon vector layer to write the index to. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Transform geometries to the given CRS Optional	TARGET_CRS	[crs]	Geometries of input files will be transformed to the specified target coordinate reference system. Default creates simple rectangular polygons in the same coordinate reference system as the input rasters.
The name of the field to store the SRS of each tile Optional	CRS_FIELD_NAME	[string]	The name of the field to store the SRS of each tile
The format in which the CRS of each tile must be written	CRS_FORMAT	[enumeration] Default: 0	Format for the CRS. One of: <ul style="list-style-type: none"> • 0 -- Auto (AUTO) • 1 -- Well-known text (WKT) • 2 -- EPSG (EPSG) • 3 -- Proj.4 (PROJ)

Outputs

Label	名称	Type	Description
Tile index	OUTPUT	[vector: polygon]	The polygon vector layer with the tile index.

Python code

Algorithm ID: gdal:tileindex

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Viewshed

Calculates a viewshed raster from an input raster DEM using method defined in Wang2000 for a user defined point.

Parameters

Basic parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input elevation raster layer
Band number	BAND	[raster band] Default: 1	The number of the band to use as elevation
Observer location	OBSERVER	[point]	The location of the observer
Observer height	OBSERVER_HEIGHT	[number] Default: 1.0	The altitude of the observer, in the DEM units
Target height	TARGET_HEIGHT	[number] Default: 1.0	The altitude of the target element, in the DEM units
Maximum distance from observer to compute visibility	MAX_DISTANCE	[number] Default: 100.0	Maximum distance from observer to compute visibility, in the DEM units
Output	OUTPUT	[raster] Default: [Save to temporary file]	Output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File...

Advanced parameters

Label	名称	Type	Description
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Additional command-line parameters	EXTRA	[string] Default: None	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Output	OUTPUT	[raster]	The raster layer displaying the viewshed.

Python code

Algorithm ID: gdal:viewshed

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.2.5 Raster projections

Assign projection

Applies a coordinate system to a raster dataset.

This algorithm is derived from the *GDAL edit utility*.

Default menu: *Raster ► Projections*

Parameters

Label	名称	Type	Description
Input layer	INPUT_LAYER	[raster]	Input raster layer
Desired CRS	CRS	[crs]	The projection (CRS) of the output layer

Outputs

Label	名称	Type	Description
Layer with projection	OUTPUT	[raster]	The output raster layer (with the new projection information)

Python code

Algorithm ID: gdal:assignprojection

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Extract projection

Extracts the projection of a raster file and writes it into a *world* file with extension *.wld*.

This algorithm is derived from the [GDAL srsinfo utility](#).

Default menu: *Raster ► Projections*

Parameters

Label	名称	Type	Description
Input file	INPUT_LAYER	[raster]	Input raster The raster layer has to be file based, as the algorithm uses the path to the raster file as the location of the generated <i>.wld</i> file. Using a non-file raster layer will lead to an error.
Create also .prj file	PRJ_FILE_CREATE	[boolean] Default: False	If this is activated a <i>.prj</i> file containing the projection information is also created.

Outputs

Label	名称	Type	Description
World file	WORLD_FILE	[file]	Text file with extension <i>.wld</i> containing transformation parameters for the raster file.
ESRI Shapefile prj file	PRJ_FILE	[file]	Text file with <i>.prj</i> extension that describes the CRS. Will be <i>None</i> if <i>Create also .prj file</i> is <i>False</i> .

Python code

Algorithm ID: gdal:extractprojection

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Warp (reproject)

Reprojects a raster layer into another Coordinate Reference System (CRS). The output file resolution and the resampling method can be chosen.

This algorithm is derived from the [GDAL warp utility](#).

Default menu: *Raster ► Projections*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[raster]	Input raster layer to reproject
Source CRS Optional	SOURCE_CRS	[crs]	Defines the CRS of the input raster layer
Target CRS Optional	TARGET_CRS	[crs] Default: EPSG:4326	The CRS of the output layer
Resampling method to use	RESAMPLING	[enumeration] Default: 0	Pixel value resampling method to use. Options: <ul style="list-style-type: none"> • 0 --- Nearest neighbour • 1 --- Bilinear • 2 --- Cubic • 3 --- Cubic spline • 4 --- Lanczos windowed sinc • 5 --- Average • 6 --- Mode • 7 --- Maximum • 8 --- Minimum • 9 --- Median • 10 --- First quartile • 11 --- Third quartile
Nodata value for output bands Optional	NODATA	[number] Default: None	Sets nodata value for output bands. If not provided, then nodata values will be copied from the source dataset.
Output file resolution in target geo-referenced units Optional	TARGET_RESOLUTION	[number] Default: None	Defines the output file resolution of reprojection result

下页继续

表 24.187 - 续上页

Label	名称	Type	Description
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see <i>GDAL driver options section</i>). For Batch Process: separate multiple options with a pipe character ().
Output data type	DATA_TYPE	[enumeration] Default: 0	Defines the format of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Use input layer data type • 1 --- Byte • 2 --- Int16 • 3 --- UInt16 • 4 --- UInt32 • 5 --- Int32 • 6 --- Float32 • 7 --- Float64 • 8 --- CInt16 • 9 --- CInt32 • 10 --- CFloat32 • 11 --- CFloat64
Georeferenced extents of output file to be created Optional	TARGET_EXTENT	[extent]	Sets the georeferenced extent of the output file to be created (in the <i>Target CRS</i> by default. In the <i>CRS of the target raster extent</i> , if specified).
CRS of the target raster extent Optional	TARGET_EXTENT_CRS	[CRS]	Specifies the CRS in which to interpret the coordinates given for the extent of the output file. This must not be confused with the target CRS of the output dataset. It is instead a convenience e.g. when knowing the output coordinates in a geodetic long/lat CRS, but wanting a result in a projected coordinate system.
Use multithreaded warping implementation	MULTITHREADING	[boolean] Default: False	Two threads will be used to process chunks of the image and perform input/output operations simultaneously. Note that the computation itself is not multithreaded.
Additional command-line parameters Optional	EXTRA	[string] Default: None	Add extra GDAL command line options.
Reprojected	OUTPUT	[raster] Default: '[Save to temporary file]'	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Reprojected	OUTPUT	[raster] Default: [Save to temporary file]	Reprojected output raster layer

Python code

Algorithm ID: gdal:warpreproject

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.2.6 Vector conversion

Convert format

Converts any OGR-supported vector layer into another OGR-supported format.

This algorithm is derived from the `ogr2ogr` utility.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
Converted	OUTPUT	[same as input]	Specification of the output vector layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here. For Save to File, the output format has to be specified. All GDAL vector formats are supported. For Save to a Temporary File the QGIS default vector format will be used.

Outputs

Label	名称	Type	Description
Converted	OUTPUT	[same as input]	The output vector layer

Python code

Algorithm ID: gdal:convertformat

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Rasterize (overwrite with attribute)

Overwrites a raster layer with values from a vector layer. New values are assigned based on the attribute value of the overlapping vector feature.

This algorithm is derived from the [GDAL rasterize utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Input raster layer	INPUT_RASTER	[raster]	Input raster layer
Field to use for a burn-in value Optional	FIELD	[tablefield: numeric]	Defines the attribute field to use to set the pixels values
Add burn in values to existing raster values	ADD	[boolean] Default: False	If False, pixels are assigned the selected field's value. If True, the selected field's value is added to the value of the input raster layer.
Additional command-line parameters Optional	EXTRA	[string] Default: ""	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Rasterized	OUTPUT	[raster]	The overwritten input raster layer

Python code

Algorithm ID: gdal:rasterize_over

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Rasterize (overwrite with fixed value)

Overwrites parts of a raster layer with a fixed value. The pixels to overwrite are chosen based on the supplied (overlapping) vector layer.

This algorithm is derived from the [GDAL rasterize utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Input raster layer	INPUT_RASTER	[raster]	Input raster layer
A fixed value to burn	BURN	[number] Default: 0.0	The value to burn
Add burn in values to existing raster values	ADD	[boolean] Default: False	If False, pixels are assigned the fixed value. If True, the fixed value is added to the value of the input raster layer.
Additional command-line parameters Optional	EXTRA	[string] Default: ""	Add extra GDAL command line options

Outputs

Label	名称	Type	Description
Rasterized	OUTPUT	[raster]	The overwritten input raster layer

Python code

Algorithm ID: gdal:rasterize_over_fixed_value

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Rasterize (vector to raster)

Converts vector geometries (points, lines and polygons) into a raster image.

This algorithm is derived from the [GDAL rasterize utility](#).

Default menu: *Raster ► Conversion*

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Field to use for a burn-in value Optional	FIELD	[tablefield: numeric]	Defines the attribute field from which the attributes for the pixels should be chosen
A fixed value to burn Optional	BURN	[number] Default: 0.0	A fixed value to burn into a band for all features.
Output raster size units	UNITS	[enumeration] Default: 0	Units to use when defining the output raster size/resolution. One of: <ul style="list-style-type: none"> • 0 --- Pixels • 1 --- Georeferenced units
Width/Horizontal resolution	WIDTH	[number] Default: 0.0	Sets the width (if size units is "Pixels") or horizontal resolution (if size units is "Georeferenced units") of the output raster. Minimum value: 0.0.
Height/Vertical resolution	HEIGHT	[number] Default: 0.0	Sets the height (if size units is "Pixels") or vertical resolution (if size units is "Georeferenced units") of the output raster.
Output extent	EXTENT	[extent]	Extent of the output raster layer. If the extent is not specified, the minimum extent that covers the selected reference layer(s) will be used.
Assign a specified nodata value to output bands Optional	NODATA	[number] Default: 0.0	Assigns a specified nodata value to output bands
Additional creation options Optional	OPTIONS	[string] Default: ""	For adding one or more creation options that control the raster to be created (colors, block size, file compression...). For convenience, you can rely on predefined profiles (see GDAL driver options section). For Batch Process: separate multiple options with a pipe character ().

下页继续

表 24.190 – 续上页

Label	名称	Type	Description
Output data type	DATA_TYPE	[enumeration] Default: 5	Defines the format of the output raster file. Options: <ul style="list-style-type: none"> • 0 --- Byte • 1 --- Int16 • 2 --- UInt16 • 3 --- UInt32 • 4 --- Int32 • 5 --- Float32 • 6 --- Float64 • 7 --- CInt16 • 8 --- CInt32 • 9 --- CFloat32 • 10 --- CFloat64
Pre-initialize the output image with value Optional	INIT	[number]	Pre-initializes the output image bands with this value. Not marked as the nodata value in the output file. The same value is used in all the bands.
Invert rasterization	INVERT	[boolean] Default: False	Burns the fixed burn value, or the burn value associated with the first feature into all parts of the image not inside the provided polygon.
Rasterized	OUTPUT	[raster] Default: '[Save to temporary file]'	Specification of the output raster layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here For Save to File, the output format has to be specified. All GDAL raster formats are supported. For Save to a Temporary File the QGIS default raster format will be used.

Outputs

Label	名称	Type	Description
Rasterized	OUTPUT	[raster]	Output raster layer

Python code

Algorithm ID: gdal:rasterize

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.2.7 Vector geoprocessing

Buffer vectors

Create buffers around the features of a vector layer.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer
Geometry column name	GEOMETRY	[string] Default: 'geometry'	The name of the input layer geometry column to use
Buffer distance	DISTANCE	[number] Default: 10.0	Minimum: 0.0
Dissolve by attribute Optional	FIELD	[tablefield: any] Default: None	Field to use for dissolving
Dissolve results	DISSOLVE	[boolean] Default: False	If set, the result is dissolved. If no field is set for dissolving, all the buffers are dissolved into one feature.
Produce one feature for each geometry in any kind of geometry collection in the source file	EXPLODE_COLLECT	[boolean] Default: False	
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
Buffer	OUTPUT	[vector: polygon] Default: [Save to temporary file]	Specify the output buffer layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Buffer	OUTPUT	[vector: polygon]	The output buffer layer

Python code

Algorithm ID: gdal:bufferectors

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Clip vector by extent

Clips any OGR-supported vector file to a given extent.

This algorithm is derived from the [GDAL ogr2ogr utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer
Clip extent	EXTENT	[extent]	Defines the bounding box that should be used for the output vector file. It has to be defined in target CRS coordinates.
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
Clipped (extent)	OUTPUT	[same as input] Default: [Save to temporary file]	Specify the output (clipped) layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Clipped (extent)	OUTPUT	[same as input]	The output (clipped) layer. The default format is "ESRI Shapefile".

Python code

Algorithm ID: gdal:clipvectorbyextent

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Clip vector by mask layer

Clips any OGR-supported vector layer by a mask polygon layer.

This algorithm is derived from the [GDAL ogr2ogr utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input vector layer
Mask layer	MASK	[vector: polygon]	Layer to be used as clipping extent for the input vector layer.
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
Clipped (mask)	OUTPUT	[same as input] Default: [Save to temporary file]	The output (masked) layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Clipped (mask)	OUTPUT	[same as input]	The output (masked) layer. The default format is "ESRI Shapefile".

Python code

Algorithm ID: gdal:clipvectorbypolygon

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

融合 (dissolve)

Dissolve (combine) geometries that have the same value for a given attribute / field. The output geometries are multipart.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	The input layer to dissolve
Dissolve field Optional	FIELD	[tablefield: any]	The field of the input layer to use for dissolving
Geometry column name	GEOMETRY	[string] Default: 'geometry'	The name of the input layer geometry column to use for dissolving.
Produce one feature for each geometry in any kind of geometry collection in the source file	EXPLODE_COLLECTION	[boolean] Default: False	Produce one feature for each geometry in any kind of geometry collection in the source file

下页继续

表 24.192 - 续上页

Label	名称	Type	Description
Keep input attributes	KEEP_ATTRIBUTES	[boolean] Default: False	Keep all attributes from the input layer
Count dissolved features	COUNT_FEATURES	[boolean] Default: False	Count the dissolved features and include it in the output layer.
Compute area and perimeter of dissolved features	COMPUTE_AREA	[boolean] Default: False	Compute the area and perimeter of dissolved features and include them in the output layer
Compute min/max/sum/mean for attribute	COMPUTE_STATISTICS	[boolean] Default: False	Calculate statistics (min, max, sum and mean) for the numeric attribute specified and include them in the output layer
Numeric attribute to calculate statistics on Optional	STATISTICS_ATTRIBUTE	[field: numeric]	The numeric attribute to calculate statistics on
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
Dissolved	OUTPUT	[same as input] Default: [Save to temporary file]	Specify the output layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Dissolved	OUTPUT	[same as input]	The output multipart geometry layer (with dissolved geometries)

Python code

Algorithm ID: gdal:dissolve

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Offset curve

Offsets lines by a specified distance. Positive distances will offset lines to the left, and negative distances will offset them to the right.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	The input line layer
Geometry column name	GEOMETRY	[string] Default: 'geometry'	The name of the input layer geometry column to use
Offset distance (left-sided: positive, right-sided: negative)	DISTANCE	[number] Default: 10.0	
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
Offset curve	OUTPUT	[vector: line] Default: [Save to temporary file]	Specify the output line layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Offset curve	OUTPUT	[vector: line]	The output offset curve layer

Python code

Algorithm ID: gdal:offsetcurve

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

One side buffer

Creates a buffer on one side (right or left) of the lines in a line vector layer.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	The input line layer
Geometry column name	GEOMETRY	[string] Default: 'geometry'	The name of the input layer geometry column to use
Buffer distance	DISTANCE	[number] Default: 10.0	
Buffer side	BUFFER_SIDE	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- Right • 1 --- Left

下页继续

表 24.194 – 续上页

Label	名称	Type	Description
Dissolve by attribute Optional	FIELD	[tablefield: any] Default: None	Field to use for dissolving
Dissolve all results	DISSOLVE	[boolean] Default: False	If set, the result is dissolved. If no field is set for dissolving, all the buffers are dissolved into one feature.
Produce one feature for each geometry in any kind of geometry collection in the source file	EXPLODE_COLLECT	[boolean] Default: False	
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
One-sided buffer	OUTPUT	[vector: polygon] Default: [Save to temporary file]	Specify the output buffer layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
One-sided buffer	OUTPUT	[vector: polygon]	The output buffer layer

Python code

Algorithm ID: gdal:onesidebuffer

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Points along lines

Generates a point on each line of a line vector layer at a distance from start. The distance is provided as a fraction of the line length.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: line]	The input line layer
Geometry column name	GEOMETRY	[string] Default: 'geometry'	The name of the input layer geometry column to use
Distance from line start represented as a fraction of line length	DISTANCE	[number] Default: 0.5 (middle of the line)	
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
Points along line	OUTPUT	[vector: point] Default: [Save to temporary file]	Specify the output point layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Points along line	OUTPUT	[vector: point]	The output point layer

Python code

Algorithm ID: gdal:pointsalonglines

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.2.8 Vector miscellaneous

Build virtual vector

Creates a virtual vector layer that contains a set of vector layers. The output virtual vector layer will not be opened in the current project.

This algorithm is especially useful in case another algorithm needs multiple layers but accept only one `vrt` in which the layers are specified.

Parameters

Label	名称	Type	Description
Input datasources	INPUT	[vector: any] [list]	Select the vector layers you want to use to build the virtual vector
Create "unioned" VRT	UNIONED	[boolean] Default: False	Check if you want to unite all the vectors in a single vrt file
Virtual vector	OUTPUT	[same as input] Default: [Save to temporary file]	Specify the output layer containing only the duplicates. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Virtual vector	OUTPUT	[vector: any]	The output virtual vector made from the chosen sources

Python code

Algorithm ID: gdal:buildvirtualvector

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Execute SQL

Runs a simple or complex query with SQL syntax on the source layer. The result of the query will be added as a new layer.

This algorithm is derived from the [GDAL ogr2ogr utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	OGR-supported input vector layer
SQL expression	SQL	[string]	Defines the SQL query, for example SELECT * FROM my_table WHERE name is not null.
SQL dialect	DIALECT	[enumeration] Default: 0	SQL dialect to use. One of: <ul style="list-style-type: none"> • 0 --- None • 1 --- OGR SQL • 2 --- SQLite

下页继续

表 24.196 – 续上页

Label	名称	Type	Description
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.
SQL result	OUTPUT	[vector: any]	Specification of the output layer. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here. For Save to File, the output format has to be specified. All GDAL vector formats are supported. For Save to a Temporary File the default output vector layer format will be used.

Outputs

Label	名称	Type	Description
SQL result	OUTPUT	[vector: any]	Vector layer created by the query

Python code

Algorithm ID: gdal:executesql

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Export to PostgreSQL (available connections)

Imports vector layers inside a PostgreSQL database on the basis of an available connection. The connection has to *be defined properly* beforehand. Be aware that the checkboxes 'Save Username' and 'Save Password' are activated. Then you can use the algorithm.

This algorithm is derived from the GDAL ogr2ogr utility.

Parameters

Label	名称	Type	Description
Database (connection name)	DATABASE	[string]	The PostgreSQL database to connect to
Input layer	INPUT	[vector: any]	OGR-supported vector layer to export to the database
Shape encoding Optional	SHAPE_ENCODING	[string] Default: ""	Sets the encoding to apply to the data

下页继续

表 24.197 - 续上页

Label	名称	Type	Description
Output geometry type	GTYPE	[enumeration] Default: 0	Defines the output geometry type. One of: <ul style="list-style-type: none"> • 0 --- • 1 --- NONE • 2 --- GEOMETRY • 3 --- POINT • 4 --- LINESTRING • 5 --- POLYGON • 6 --- GEOMETRYCOLLECTION • 7 --- MULTIPOINT • 8 --- MULTIPOLYGON • 9 --- MULTILINESTRING
Assign an output CRS Optional	A_SRS	[crs] Default: None	Defines the output CRS of the database table
Reproject to this CRS on output Optional	T_SRS	[crs] Default: None	Reprojects/transforms to this CRS on output
Override source CRS Optional	S_SRS	[crs] Default: None	Overrides the input layer CRS
Schema (schema name) Optional	SCHEMA	[string] Default: 'public'	Defines the schema for the database table
Table to export to (leave blank to use layer name) Optional	TABLE	[string] Default: ''	Defines a name for the table that will be imported into the database. By default the table name is the name of the input vector file.
Primary Key (new field) Optional	PK	[string] Default: 'id'	Defines which attribute field will be the primary key of the database table
Primary Key (existing field, used if the above option is left empty) Optional	PRIMARY_KEY	[tablefield: any] Default: None	Defines which attribute field in the exported layer will be the primary key of the database table
Geometry column name Optional	GEOCOLUMN	[string] Default: 'geom'	Defines in which attribute field of the database there will be the geometry information
Vector dimensions Optional	DIM	[enumeration] Default: 0 (2D)	Defines if the vector file to be imported has 2D or 3D data. One of: <ul style="list-style-type: none"> • 0 --- 2 • 1 --- 3
Distance tolerance for simplification Optional	SIMPLIFY	[string] Default: ''	Defines a distance tolerance for the simplification of the vector geometries to be imported. By default there is no simplification.
Maximum distance between 2 nodes (densification) Optional	SEGMENTIZE	[string] Default: ''	The maximum distance between two nodes. Used to create intermediate points. By default there is no densification.

下页继续

表 24.197 – 续上页

Label	名称	Type	Description
Select features by extent (defined in input layer CRS) Optional	SPAT	[extent] Default: None	You can select features from a given extent that will be in the output table.
Clip the input layer using the above (rectangle) extent Optional	CLIP	[boolean] Default: False	The input layer will be clipped by the extent you defined before
Select features using a SQL "WHERE" statement (Ex: column="value") Optional	WHERE	[string] Default: "	Defines with a SQL "WHERE" statement which features should be selected from the input layer
Group N features per transaction (Default: 2000) Optional	GT	[string] Default: "	You can group the input features in transactions where N defines the size. By default N limits the transaction size to 20000 features.
Overwrite existing table Optional	OVERWRITE	[boolean] Default: True	If there is a table with the same name in the database, and if this option is set to True, the table will be overwritten.
Append to existing table Optional	APPEND	[boolean] Default: False	If checked / True the vector data will be appended to an existing table. New fields found in the input layer are ignored. By default a new table will be created.
Append and add new fields to existing table Optional	ADDFIELDS	[boolean] Default: False	If activated the vector data will be appended to an existing table, there won't be a new table created. New fields found in input layer are added to the table. By default a new table will be created.
Do not launder columns/table names Optional	LAUNDER	[boolean] Default: False	With this option checked you can prevent the default behaviour (converting column names to lowercase, removing spaces and other invalid characters).
Do not create Spatial Index Optional	INDEX	[boolean] Default: False	Prevents a spatial index for the output table from being created. By default, a spatial index is added.
Continue after a failure, skipping the failed feature Optional	SKIPFAILURES	[boolean] Default: False	
Promote to Multi-part Optional	PROMOTETOMULTI	[boolean] Default: True	Casts features geometry type to multipart in the output table
Keep width and precision of input attributes Optional	PRECISION	[boolean] Default: True	Avoids modifying column attributes to comply with input data
Additional creation options Optional	OPTIONS	[string] Default: " (no additional options)	Additional GDAL creation options.

Outputs

This algorithm has no output.

Python code

Algorithm ID: gdal:importvectorintopostgisdatabaseavailableconnections

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Export to PostgreSQL (new connection)

Imports vector layers inside a PostgreSQL database. A new connection to the PostGIS database must be created.

This algorithm is derived from the GDAL `ogr2ogr` utility.

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	OGR-supported vector layer to export to the database
Shape encoding Optional	SHAPE_ENCODING	[string] Default: ""	Sets the encoding to apply to the data
Output geometry type	GTYPE	[enumeration] Default: 0	Defines the output geometry type. One of: <ul style="list-style-type: none"> • 0 --- • 1 --- NONE • 2 --- GEOMETRY • 3 --- POINT • 4 --- LINESTRING • 5 --- POLYGON • 6 --- GEOMETRYCOLLECTION • 7 --- MULTIPOINT • 8 --- MULTIPOLYGON • 9 --- MULTILINESTRING
Assign an output CRS Optional	A_SRS	[crs] Default: None	Defines the output CRS of the database table
Reproject to this CRS on output Optional	T_SRS	[crs] Default: None	Reprojects/transforms to this CRS on output
Override source CRS Optional	S_SRS	[crs] Default: None	Overrides the input layer CRS
Host Optional	HOST	[string] Default: 'localhost'	Name of the database host
Port Optional	PORT	[string] Default: '5432'	Port number the PostgreSQL database server listens on

下页继续

表 24.198 – 续上页

Label	名称	Type	Description
Username Optional	USER	[string] Default: ""	User name used to log in to the database
Database name Optional	DBNAME	[string] Default: ""	Name of the database
Password Optional	PASSWORD	[string] Default: ""	Password used with Username to connect to the database
Schema (schema name) Optional	SCHEMA	[string] Default: 'public'	Defines the schema for the database table
Table name, leave blank to use input name Optional	TABLE	[string] Default: ""	Defines a name for the table that will be imported into the database. By default the table name is the name of the input vector file.
Primary Key (new field) Optional	PK	[string] Default: 'id'	Defines which attribute field will be the primary key of the database table
Primary Key (existing field, used if the above option is left empty) Optional	PRIMARY_KEY	[tablefield: any] Default: None	Defines which attribute field in the exported layer will be the primary key of the database table
Geometry column name Optional	GEOCOLUMN	[string] Default: 'geom'	Defines in which attribute field to store the geometry information
Vector dimensions Optional	DIM	[enumeration] Default: 0 (2D)	Defines if the vector file to be imported has 2D or 3D data. One of: <ul style="list-style-type: none"> • 0 --- 2D • 1 --- 3D
Distance tolerance for simplification Optional	SIMPLIFY	[string] Default: ""	Defines a distance tolerance for the simplification of the vector geometries to be imported. By default no simplification there is no simplification.
Maximum distance between 2 nodes (densification) Optional	SEGMENTIZE	[string] Default: ""	The maximum distance between two nodes. Used to create intermediate points. By default there is no densification.
Select features by extent (defined in input layer CRS) Optional	SPAT	[extent] Default: None	You can select features from a given extent that will be in the output table.
Clip the input layer using the above (rectangle) extent Optional	CLIP	[boolean] Default: False	The input layer will be clipped by the extent you defined before
Fields to include (leave empty to use all fields) Optional	FIELDS	[string] [list] Default: []	Defines fields to keep from the imported vector file. If none is selected, all the fields are imported.

下页继续

表 24.198 – 续上页

Label	名称	Type	Description
Select features using a SQL "WHERE" statement (Ex: column="value") Optional	WHERE	[string] Default: ""	Defines with a SQL "WHERE" statement which features should be selected for the output table
Group N features per transaction (Default: 2000) Optional	GT	[string] Default: ""	You can group the input features in transactions where N defines the size. By default N limits the transaction size to 20000 features.
Overwrite existing table Optional	OVERWRITE	[boolean] Default: True	If there is a table with the same name in the database, and if this option is set to True, the table will be overwritten.
Append to existing table Optional	APPEND	[boolean] Default: False	If checked / True the vector data will be appended to an existing table. New fields found in the input layer are ignored. By default a new table will be created.
Append and add new fields to existing table Optional	ADDFIELDS	[boolean] Default: False	If activated the vector data will be appended to an existing table, there won't be created a new table. New fields found in input layer are added to the table. By default a new table will be created.
Do not launder columns/table names Optional	LAUNDER	[boolean] Default: False	With this option checked you can prevent the default behaviour (converting column names to lowercase, removing spaces and other invalid characters).
Do not create Spatial Index Optional	INDEX	[boolean] Default: False	Prevents a spatial index for the output table from being created. By default, a spatial index is added.
Continue after a failure, skipping the failed feature Optional	SKIPFAILURES	[boolean] Default: False	
Promote to Multi-part Optional	PROMOTETOMULTI	[boolean] Default: True	Casts features geometry type to multipart in the output table
Keep width and precision of input attributes Optional	PRECISION	[boolean] Default: True	Avoids modifying column attributes to comply with input data
Additional creation options Optional	OPTIONS	[string] Default: "" (no additional options)	Additional GDAL creation options.

Outputs

This algorithm has no output.

Python code

Algorithm ID: gdal:importvectorintopostgisdatabasewconnection

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Vector Information

Creates an information file that lists information about an OGR-supported data source. The output will be shown in a 'Result' window and can be written into a HTML-file. The information includes the geometry type, feature count, the spatial extent, the projection information and many more.

This algorithm is derived from the [GDAL ogrinfo utility](#).

Parameters

Label	名称	Type	Description
Input layer	INPUT	[vector: any]	Input vector layer
Summary output only Optional	SUMMARY_ONLY	[boolean] Default: True	
Suppress meta-data info Optional	NO_METADATA	[boolean] Default: False	
Layer information	OUTPUT	[html] Default: [Save to temporary file]	Specify the output HTML file that includes the file information. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here. If no HTML-file is defined the output will be written to a temporary file

Outputs

Label	名称	Type	Description
Layer information	OUTPUT	[html]	The output HTML-file that includes the file information.

Python code

Algorithm ID: gdal:ogrinfo

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.3 LAsTools algorithm provider

LAsTools is a collection of highly efficient, multicore command line tools for LiDAR data processing.

24.3.1 blast2dem

Description

Turns points (up to billions) via seamless Delaunay triangulation implemented using streaming into large elevation, intensity, or RGB rasters.

For more info see the [blast2dem](#) page and its online [README](#) file.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
open GUI	GUI	[boolean] Default: False	Starts the GUI of LAsTools with pre-populated input files
input file	INPUT_LASLAZ	[file]	The file containing the points to be rastered in LAS/LAZ format.

下页继续

表 24.200 - 续上页

Label	名称	Type	Description
filter (by return, classification, flag)	FILTER_RETURN_CLASSIFICATION	[enumeration] Default: 0	Specifies which points to use to construct the temporary TIN that is then rasterized. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- keep_last • 2 --- keep_first • 3 --- keep_middle • 4 --- keep_single • 5 --- drop_single • 6 --- keep_double • 7 --- keep_class 2 • 8 --- keep_class 2 8 • 9 --- keep_class 8 • 10 --- keep_class 6 • 11 --- keep_class 9 • 12 --- keep_class 3 4 5 • 13 --- keep_class 2 6 • 14 --- drop_class 7 • 15 --- drop_withheld • 16 --- drop_synthetic • 17 --- drop_overlap • 18 --- keep_withheld • 19 --- keep_synthetic • 20 --- keep_keypoint • 21 --- keep_overlap
step size / pixel size	STEP	[number] Default: 1.0	Specifies the size of the cells of the grid the TIN is rasterized onto
Attribute	ATTRIBUTE	[enumeration] Default: 0	Specifies the attribute that is to be rastered. One of: <ul style="list-style-type: none"> • 0 --- elevation • 1 --- slope • 2 --- intensity • 3 --- rgb
Product	PRODUCT	[enumeration] Default: 0	Specifies how the attribute is to be turned into raster values. One of: <ul style="list-style-type: none"> • 0 --- actual values • 1 --- hillshade • 2 --- gray • 3 --- false
Use tile bounding box (after tiling with buffer)	USE_TILE_BB	[boolean] Default: False	Specifies to limit the rastered area to the tile bounding box (only meaningful for input LAS/LAZ tiles that were created with lastile).
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.

下页继续

表 24.200 – 续上页

Label	名称	Type	Description
Output raster file	OUTPUT_RASTER	[raster] Default: [Skip output]	Specifies where the output raster is stored. Use image rasters like TIF, PNG, and JPG for false color, gray ramps, and hillshades. Use value rasters like TIF, BIL, IMG, ASC, DTM, FLT, XYZ, and CSV for actual values. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output raster file	OUTPUT_RASTER	[raster]	The output raster

Python code

Algorithm ID: lastools:blast2dem

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.2 blast2iso

Description

Turns points (up to billions) via seamless Delaunay triangulation implemented using streaming into iso-contour lines. For more info see the [blast2iso](#) page and its online [README](#) file.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
open LAStools GUI	GUI	[boolean] Default: False	Starts the GUI of LAStools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file containing the points to be used for creating iso-contour lines.
smooth underlying TIN	SMOOTH	[number] Default: 0	Specifies if and with how many passes the temporary TIN should be smoothed
extract isoline with a spacing of	ISO_EVERY	[number] Default: 10.0	Specifies spacing at which iso-contour lines are getting extracted (contour interval)

下页继续

表 24.201 - 续上页

Label	名称	Type	Description
clean isolines shorter than (0 = do not clean)	CLEAN	[number] Default: 0.0	Omits iso-contour lines that are shorter than the specified length
simplify segments shorter than (0 = do not simplify)	SIMPLIFY_LENGTH	[number] Default: 0.0	Rudimentary simplification of iso-contour line segments that are shorter than the specified length.
simplify segment pairs with area less than (0 = do not simplify)	SIMPLIFY_AREA	[number] Default: 0.0	Rudimentary simplification of bumps formed by consecutive line segments whose area is smaller than the specified size.
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output vector file	OUTPUT_VECTOR	[vector: line] Default: [Skip output]	Specifies where the output vector is stored. Use SHP or WKT output files. If your input LiDAR file is in geographic coordinates (long/lat) or has geo-referencing information (but only then) you can also create a KML output file. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output vector file	OUTPUT_VECTOR	[vector: line]	The output line vector layer with contours

Python code

Algorithm ID: lastools:blast2iso

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.3 las2dem

Description

Turns points (up to 20 million) via a temporary Delaunay triangulation that is rasterized with a user-defined step size into an elevation, intensity, or RGB raster.

For more info see the [las2dem](#) page and its online [README](#) file.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAStools GUI	GUI	[boolean] Default: False	Starts the GUI of LAStools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file containing the points to be rastered in LAS/LAZ format.
filter (by return, classification, flags)	FILTER_RETURN_CLASSIFICATION	[enumeration] Default: 0	Specifies which points to use to construct the temporary TIN that is then rasterized. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- keep_last • 2 --- keep_first • 3 --- keep_middle • 4 --- keep_single • 5 --- drop_single • 6 --- keep_double • 7 --- keep_class 2 • 8 --- keep_class 2 8 • 9 --- keep_class 8 • 10 --- keep_class 6 • 11 --- keep_class 9 • 12 --- keep_class 3 4 5 • 13 --- keep_class 3 • 14 --- keep_class 4 • 15 --- keep_class 5 • 16 --- keep_class 2 6 • 17 --- drop_class 7 • 18 --- drop_withheld • 19 --- drop_synthetic • 20 --- drop_overlap • 21 --- keep_withheld • 22 --- keep_synthetic • 23 --- keep_keypoint • 24 --- keep_overlap
step size / pixel size	STEP	[number] Default: 1.0	Specifies the size of the cells of the grid the TIN is rasterized onto
Attribute	ATTRIBUTE	[enumeration] Default: 0	Specifies the attribute to rasterise. One of: <ul style="list-style-type: none"> • 0 --- elevation • 1 --- slope • 2 --- intensity • 3 --- rgb • 4 --- edge_longest • 5 --- edge_shortest

下页继续

表 24.202 – 续上页

Label	名称	Type	Description
Product	PRODUCT	[enumeration] Default: 0	Specifies how the attribute is to be turned into raster values. One of: <ul style="list-style-type: none"> • 0 --- actual values • 1 --- hillshade • 2 --- gray • 3 --- false
Use tile bounding box (after tiling with buffer)	USE_TILE_BB	[boolean] Default: False	Specifies to limit the rastered area to the tile bounding box (only meaningful for input LAS/LAZ tiles that were created with lastile).
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output raster file	OUTPUT_RASTER	[raster] Default: [Skip output]	Specifies where the output raster is stored. Use image rasters like TIF, PNG, and JPG for false color, gray ramps, and hillshades. Use value rasters like TIF, BIL, IMG, ASC, DTM, FLT, XYZ, and CSV for actual values. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output raster file	OUTPUT_RASTER	[raster]	The output raster

Python code

Algorithm ID: lastools:las2dem

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.4 las2iso

Description

Turns point clouds (up to 20 million per file) into iso-contour lines by creating a temporary Delaunay triangulation on which the contours are then traced.

For more info see the [las2iso](#) page and its online [README](#) file.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAsTools GUI	GUI	[boolean] Default: False	Starts the GUI of LAsTools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file containing the points to be used for creating iso-contour lines.
smooth underlying TIN	SMOOTH	[number] Default: 0	Specifies if and with how many passes the temporary TIN should be smoothed
extract isoline with a spacing of	ISO_EVERY	[number] Default: 10.0	Specifies spacing at which iso-contour lines are getting extracted (contour interval)
clean isolines shorter than (0 = do not clean)	CLEAN	[number] Default: 0.0	Omits iso-contour lines that are shorter than the specified length
simplify segments shorter than (0 = do not simplify)	SIMPLIFY_LENGTH	[number] Default: 0.0	Rudimentary simplification of iso-contour line segments that are shorter than the specified length.
simplify segment pairs with area less than (0 = do not simplify)	SIMPLIFY_AREA	[number] Default: 0.0	Rudimentary simplification of bumps formed by consecutive line segments whose area is smaller than the specified size.
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output vector file	OUTPUT_VECTOR	[vector: line] Default: [Skip output]	Specifies where the output vector is stored. Use SHP or WKT output files. If your input LiDAR file is in geographic coordinates (long/lat) or has geo-referencing information (but only then) you can also create a KML output file. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output vector file	OUTPUT_VECTOR	[vector: line]	The output line vector layer with contours

Python code

Algorithm ID: lastools:las2iso

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.3.5 las2las_filter

Description

Uses las2las to filter LiDAR points based on different attributes and to write the surviving subset of points to a new LAZ or LAS file.

For more info see the [las2las](#) page and its online [README](#) file.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAStools GUI	GUI	[boolean] Default: False	Starts the GUI of LAStools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file containing the points to be used for creating iso-contour lines.

下页继续

表 24.204 - 续上页

Label	名称	Type	Description
filter (by return, classification, flags)	FILTER_RETURN_CLASSIFICATION	Enumeration Default: 0	Filters points based on various options such as return, classification, or flags. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- keep_last • 2 --- keep_first • 3 --- keep_middle • 4 --- keep_single • 5 --- drop_single • 6 --- keep_double • 7 --- keep_class 2 • 8 --- keep_class 2 8 • 9 --- keep_class 8 • 10 --- keep_class 6 • 11 --- keep_class 9 • 12 --- keep_class 3 4 5 • 13 --- keep_class 3 • 14 --- keep_class 4 • 15 --- keep_class 5 • 16 --- keep_class 2 6 • 17 --- drop_class 7 • 18 --- drop_withheld • 19 --- drop_synthetic • 20 --- drop_overlap • 21 --- keep_withheld • 22 --- keep_synthetic • 23 --- keep_keypoint • 24 --- keep_overlap

下页继续

表 24.204 - 续上页

Label	名称	Type	Description
second filter (by return, classification, flags)	FILTER_RETURN_CLASSIFICATION	[enumeration] Default: 0	Filters points based on various options such as return, classification, or flags. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- keep_last • 2 --- keep_first • 3 --- keep_middle • 4 --- keep_single • 5 --- drop_single • 6 --- keep_double • 7 --- keep_class 2 • 8 --- keep_class 2 8 • 9 --- keep_class 8 • 10 --- keep_class 6 • 11 --- keep_class 9 • 12 --- keep_class 3 4 5 • 13 --- keep_class 3 • 14 --- keep_class 4 • 15 --- keep_class 5 • 16 --- keep_class 2 6 • 17 --- drop_class 7 • 18 --- drop_withheld • 19 --- drop_synthetic • 20 --- drop_overlap • 21 --- keep_withheld • 22 --- keep_synthetic • 23 --- keep_keypoint • 24 --- keep_overlap
filter (by coordinate, intensity, GPS time, ...)	FILTER_COORDS_INTENSITY	[enumeration] Default: 0	Filters points based on various other options (that require a value as argument). One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- drop_x_above • 2 --- drop_x_below • 3 --- drop_y_above • 4 --- drop_y_below • 5 --- drop_z_above • 6 --- drop_z_below • 7 --- drop_intensity_above • 8 --- drop_intensity_below • 9 --- drop_gps_time_above • 10 --- drop_gps_time_below • 11 --- drop_scan_angle_above • 12 --- drop_scan_angle_below • 13 --- keep_point_source • 14 --- drop_point_source • 15 --- drop_point_source_above • 16 --- drop_point_source_below • 17 --- keep_user_data • 18 --- drop_user_data • 19 --- drop_user_data_above • 20 --- drop_user_data_below • 21 --- keep_every_nth • 22 --- keep_random_fraction • 23 --- thin_with_grid

下页继续

表 24.204 - 续上页

Label	名称	Type	Description
value for filter (by coordinate, intensity, GPS time, ...)	FILTER_COORDS_INTENSITY1_ARG	[number] Default: None	The value to use as the argument for the filter selected above
second filter (by coordinate, intensity, GPS time, ...)	FILTER_COORDS_INTENSITY2_ARG	[enumeration] Default: 0	Filters points based on various other options (that require a value as argument). One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- drop_x_above • 2 --- drop_x_below • 3 --- drop_y_above • 4 --- drop_y_below • 5 --- drop_z_above • 6 --- drop_z_below • 7 --- drop_intensity_above • 8 --- drop_intensity_below • 9 --- drop_gps_time_above • 10 --- drop_gps_time_below • 11 --- drop_scan_angle_above • 12 --- drop_scan_angle_below • 13 --- keep_point_source • 14 --- drop_point_source • 15 --- drop_point_source_above • 16 --- drop_point_source_below • 17 --- keep_user_data • 18 --- drop_user_data • 19 --- drop_user_data_above • 20 --- drop_user_data_below • 21 --- keep_every_nth • 22 --- keep_random_fraction • 23 --- thin_with_grid
value for second filter (by coordinate, intensity, GPS time, ...)	FILTER_COORDS_INTENSITY2_ARG	[number] Default: None	The value to use as the argument for the filter selected above
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output LAS/LAZ file	OUTPUT_LASLAZ	[file] Default: [Skip output]	Specifies where the output point cloud is stored. Use LAZ for compressed output, LAS for uncompressed output, and TXT for ASCII. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output LAS/LAZ file	OUTPUT_LASLAZ	[file]	The output LAS/LAZ format file

Python code

Algorithm ID: lastools:las2las_filter

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.6 las2las_project

Transform LAS/LAZ files in a folder to another CRS.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAsTools GUI	GUI	[boolean] Default: False	Starts the GUI of LAsTools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	Input LAS/LAZ file
source projection	SOURCE_PROJECTION	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- epsg • 2 --- utm • 3 --- sp83 • 4 --- sp27 • 5 --- longlat • 6 --- latlong • 7 --- ecef

下页继续

表 24.205 - 续上页

Label	名称	Type	Description
source utm zone	SOURCE_UTM	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- 0 • 1 --- 1 (north) • 2 --- 2 (north) • 3 --- 3 (north) • 4 --- 4 (north) • 5 --- 5 (north) • 6 --- 6 (north) • 7 --- 7 (north) • 8 --- 8 (north) • 9 --- 9 (north) • 10 --- 10 (north) • 11 --- 11 (north) • 12 --- 12 (north) • 13 --- 13 (north) • 14 --- 14 (north) • 15 --- 15 (north) • 16 --- 16 (north) • 17 --- 17 (north) • 18 --- 18 (north) • 19 --- 19 (north) • 20 --- 20 (north) • 21 --- 21 (north) • 22 --- 22 (north) • 23 --- 23 (north) • 24 --- 24 (north) • 25 --- 25 (north) • 26 --- 26 (north) • 27 --- 27 (north) • 28 --- 28 (north) • 29 --- 29 (north) • 30 --- 30 (north) • 31 --- 31 (north) • 32 --- 32 (north) • 33 --- 33 (north) • 34 --- 34 (north) • 35 --- 35 (north) • 36 --- 36 (north) • 37 --- 37 (north) • 38 --- 38 (north) • 39 --- 39 (north) • 40 --- 40 (north) • 41 --- 41 (north) • 42 --- 42 (north) • 43 --- 43 (north) • 44 --- 44 (north) • 45 --- 45 (north) • 46 --- 46 (north) • 47 --- 47 (north) • 48 --- 48 (north) • 49 --- 49 (north) • 50 --- 50 (north) • 51 --- 51 (north) • 52 --- 52 (north) • 53 --- 53 (north) • 54 --- 54 (north) • 55 --- 55 (north) • 56 --- 56 (north)
1208			Chapter 24 (数据处理提供程序和算法) <ul style="list-style-type: none"> • 58 --- 58 (north) • 59 --- 59 (north) • 60 --- 60 (north) • 61 --- 61 (north)

表 24.205 - 续上页

Label	名称	Type	Description
source state plane code	SOURCE_SP	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- AK_10 • 2 --- AK_2 • 3 --- AK_3 • 4 --- AK_4 • 5 --- AK_5 • 6 --- AK_6 • 7 --- AK_7 • 8 --- AK_8 • 9 --- AK_9 • 10 --- AL_E • 11 --- AL_W • 12 --- AR_N • 13 --- AR_S • 14 --- AZ_C • 15 --- AZ_E • 16 --- AZ_W • 17 --- CA_I • 18 --- CA_II • 19 --- CA_III • 20 --- CA_IV • 21 --- CA_V • 22 --- CA_VI • 23 --- CA_VII • 24 --- CO_C • 25 --- CO_N • 26 --- CO_S • 27 --- CT • 28 --- DE • 29 --- FL_E • 30 --- FL_N • 31 --- FL_W • 32 --- GA_E • 33 --- GA_W • 34 --- HI_1 • 35 --- HI_2 • 36 --- HI_3 • 37 --- HI_4 • 38 --- HI_5 • 39 --- IA_N • 40 --- IA_S • 41 --- ID_C • 42 --- ID_E • 43 --- ID_W • 44 --- IL_E • 45 --- IL_W • 46 --- IN_E • 47 --- IN_W • 48 --- KS_N • 49 --- KS_S • 50 --- KY_N • 51 --- KY_S • 52 --- LA_N • 53 --- LA_S • 54 --- MA_I • 55 --- MA_M • 56 --- MD
24.3. LAStools algorithm provider			<ul style="list-style-type: none"> • 57 --- ME_E • 58 --- ME_W • 59 --- MI_C • 60 --- MI_N • 61 --- MI_S

表 24.205 - 续上页

Label	名称	Type	Description
target projection	TARGET_PROJECTION	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- epsg • 2 --- utm • 3 --- sp83 • 4 --- sp27 • 5 --- longlat • 6 --- latlong • 7 --- ecef

下页继续

表 24.205 - 续上页

Label	名称	Type	Description
target utm zone	TARGET_UTM	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- 0 • 1 --- 1 (north) • 2 --- 2 (north) • 3 --- 3 (north) • 4 --- 4 (north) • 5 --- 5 (north) • 6 --- 6 (north) • 7 --- 7 (north) • 8 --- 8 (north) • 9 --- 9 (north) • 10 --- 10 (north) • 11 --- 11 (north) • 12 --- 12 (north) • 13 --- 13 (north) • 14 --- 14 (north) • 15 --- 15 (north) • 16 --- 16 (north) • 17 --- 17 (north) • 18 --- 18 (north) • 19 --- 19 (north) • 20 --- 20 (north) • 21 --- 21 (north) • 22 --- 22 (north) • 23 --- 23 (north) • 24 --- 24 (north) • 25 --- 25 (north) • 26 --- 26 (north) • 27 --- 27 (north) • 28 --- 28 (north) • 29 --- 29 (north) • 30 --- 30 (north) • 31 --- 31 (north) • 32 --- 32 (north) • 33 --- 33 (north) • 34 --- 34 (north) • 35 --- 35 (north) • 36 --- 36 (north) • 37 --- 37 (north) • 38 --- 38 (north) • 39 --- 39 (north) • 40 --- 40 (north) • 41 --- 41 (north) • 42 --- 42 (north) • 43 --- 43 (north) • 44 --- 44 (north) • 45 --- 45 (north) • 46 --- 46 (north) • 47 --- 47 (north) • 48 --- 48 (north) • 49 --- 49 (north) • 50 --- 50 (north) • 51 --- 51 (north) • 52 --- 52 (north) • 53 --- 53 (north) • 54 --- 54 (north) • 55 --- 55 (north) • 56 --- 56 (north)
24.3. LAStools algorithm provider			<ul style="list-style-type: none"> • 57 --- 57 (north) • 58 --- 58 (north) • 59 --- 59 (north) • 60 --- 60 (north) • 61 --- 61 (north)

表 24.205 - 续上页

Label	名称	Type	Description
target state plane code	TARGET_SP	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- AK_10 • 2 --- AK_2 • 3 --- AK_3 • 4 --- AK_4 • 5 --- AK_5 • 6 --- AK_6 • 7 --- AK_7 • 8 --- AK_8 • 9 --- AK_9 • 10 --- AL_E • 11 --- AL_W • 12 --- AR_N • 13 --- AR_S • 14 --- AZ_C • 15 --- AZ_E • 16 --- AZ_W • 17 --- CA_I • 18 --- CA_II • 19 --- CA_III • 20 --- CA_IV • 21 --- CA_V • 22 --- CA_VI • 23 --- CA_VII • 24 --- CO_C • 25 --- CO_N • 26 --- CO_S • 27 --- CT • 28 --- DE • 29 --- FL_E • 30 --- FL_N • 31 --- FL_W • 32 --- GA_E • 33 --- GA_W • 34 --- HI_1 • 35 --- HI_2 • 36 --- HI_3 • 37 --- HI_4 • 38 --- HI_5 • 39 --- IA_N • 40 --- IA_S • 41 --- ID_C • 42 --- ID_E • 43 --- ID_W • 44 --- IL_E • 45 --- IL_W • 46 --- IN_E • 47 --- IN_W • 48 --- KS_N • 49 --- KS_S • 50 --- KY_N • 51 --- KY_S • 52 --- LA_N • 53 --- LA_S • 54 --- MA_I • 55 --- MA_M • 56 --- MD
1212			Chapter 24 数据处理提供程序和算法 • 57 --- ME_E • 58 --- ME_W • 59 --- MI_C • 60 --- MI_N • 61 --- MI_S

表 24.205 - 续上页

Label	名称	Type	Description
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output LAS/LAZ file	OUTPUT_LASLAZ	[folder] Default: [Save to temporary folder]	Specifies where the folder for the output point clouds. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary Directory • Save to Directory... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output LAS/LAZ file	OUTPUT_LASLAZ	[file]	The output LAS/LAZ format file

Python code

Algorithm ID: lastools:las2las_project

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.7 las2las_transform

Description

Uses las2las to filter LiDAR points based on different attributes and to write the surviving subset of points to a new LAZ or LAS file.

For more info see the [las2las](#) page and its online [README](#) file.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAsTools GUI	GUI	[boolean] Default: False	Starts the GUI of LAsTools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The first file containing points to be merged

下页继续

表 24.206 - 续上页

Label	名称	Type	Description
transform (coordinates)	TRANSFORM_COORD	[enumeration] Default: 0	Either translate, scale, or clamp the X, Y, or Z coordinate by the value specified below. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- translate_x • 2 --- translate_y • 3 --- translate_z • 4 --- scale_x • 5 --- scale_y • 6 --- scale_z • 7 --- clamp_z_above • 8 --- clamp_z_below
value for transform (coordinates)	TRANSFORM_COORD	[string] 1_ARG Default: ""	The value that specifies the amount of translating, scaling, or clamping done by the transform selected above.
second transform (coordinates)	TRANSFORM_COORD	[enumeration] Default: 0	Either translate, scale, or clamp the X, Y, or Z coordinate by the value specified below. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- translate_x • 2 --- translate_y • 3 --- translate_z • 4 --- scale_x • 5 --- scale_y • 6 --- scale_z • 7 --- clamp_z_above • 8 --- clamp_z_below
value for second transform (coordinates)	TRANSFORM_COORD	[string] 2_ARG Default: ""	The value that specifies the amount of translating, scaling, or clamping done by the transform selected above.
transform (intensities, scan angles, GPS times, ...)	TRANSFORM_OTHER	[enumeration] Default: 0	Either translate, scale, or clamp the X, Y, or Z coordinate by the value specified below. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- scale_intensity • 2 --- translate_intensity • 3 --- clamp_intensity_above • 4 --- clamp_intensity_below • 5 --- scale_scan_angle • 6 --- translate_scan_angle • 7 --- translate_gps_time • 8 --- set_classification • 9 --- set_user_data • 10 --- set_point_source • 11 --- scale_rgb_up • 12 --- scale_rgb_down • 13 --- repair_zero_returns
value for transform (intensities, scan angles, GPS times, ...)	TRANSFORM_OTHER	[string] Default: ""	The value that specifies the amount of scaling, translating, clamping or setting that is done by the transform selected above.

下页继续

表 24.206 - 续上页

Label	名称	Type	Description
second transform (intensities, scan angles, GPS times, ...)	TRANSFORM_OTHER	[enumeration] Default: 0	Either translate, scale, or clamp the X, Y, or Z coordinate by the value specified below. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- scale_intensity • 2 --- translate_intensity • 3 --- clamp_intensity_above • 4 --- clamp_intensity_below • 5 --- scale_scan_angle • 6 --- translate_scan_angle • 7 --- translate_gps_time • 8 --- set_classification • 9 --- set_user_data • 10 --- set_point_source • 11 --- scale_rgb_up • 12 --- scale_rgb_down • 13 --- repair_zero_returns
value for second transform (intensities, scan angles, GPS times, ...)	TRANSFORM_OTHER	[string] Default: ""	The value that specifies the amount of scaling, translating, clamping or setting that is done by the transform selected above.
operations (first 7 need an argument)	OPERATION	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- set_point_type • 2 --- set_point_size • 3 --- set_version_minor • 4 --- set_version_major • 5 --- start_at_point • 6 --- stop_at_point • 7 --- remove_vlr • 8 --- auto_reoffset • 9 --- week_to_adjusted • 10 --- adjusted_to_week • 11 --- auto_reoffset • 12 --- scale_rgb_up • 13 --- scale_rgb_down • 14 --- remove_all_vlrs • 15 --- remove_extra • 16 --- clip_to_bounding_box
argument for operation	OPERATIONARG	[string] Default: ""	The value to use as the argument for the operation selected above
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output LAS/LAZ file	OUTPUT_LASLAZ	[file] Default: [Skip output]	Specifies where the output point cloud is stored. Use LAZ for compressed output, LAS for uncompressed output, and TXT for ASCII. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output LAS/LAZ file	OUTPUT_LASLAZ	[file]	The output (merged) LAS/LAZ format file

Python code

Algorithm ID: lastools:las2las_transform

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.8 las2txt

Description

Translates a LAS/LAZ file to a text file.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAStools GUI	GUI	[boolean] Default: False	
input LAS/LAZ file	INPUT_LASLAZ	[file] Default: None	
parse_string	PARSE	[string] Default: 'xyz'	
additional command line parameters Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAStools user.
Output ASCII file	OUTPUT_GENERIC	[file] Default: [Create temporary layer]	Specify the output file. One of: <ul style="list-style-type: none"> • Create Temporary Layer (TEMPORARY_OUTPUT) • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output ASCII file	OUTPUT_GENERIC	[file]	The output file

Python code

Algorithm ID: lastools:las2txt

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.9 lasindex

Description

<put algorithm description here>

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAStools GUI	GUI	[boolean] Default: False	
input LAS/LAZ file	INPUT_LASLAZ	[file] Default: None	
append *.lax file to *.laz file	APPEND_LAX	[boolean] Default: False	
is mobile or terrestrial LiDAR (not airborne)	MOBILE_OR_TERR	[boolean] Default: False	
additional command line parameters Optional	ADDITIONAL_OPTI	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAStools user.

Outputs

The algorithm has no output.

Python code

Algorithm ID: lastools:lasindex

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.3.10 lasgrid

Grids a selected attribute (e.g. elevation, intensity, classification, scan angle, ...) of a large point clouds with a user-defined step size onto raster using a particular method (e.g. min, max, average).

For more info see the [lasgrid](#) page and its online [README](#) file.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAStools GUI	GUI	[boolean] Default: False	Starts the GUI of LAStools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file containing the points to be rastered in LAS/LAZ format.

下页继续

表 24.209 – 续上页

Label	名称	Type	Description
filter (by return, classification, flags)	FILTER_RETURN_CLASSIFICATION	[enumeration] Default: 0	Specifies the subset of points to use for the gridding. One of: <ul style="list-style-type: none"> • 0 --- • 1 --- keep_last • 2 --- keep_first • 3 --- keep_middle • 4 --- keep_single • 5 --- drop_single • 6 --- keep_double • 7 --- keep_class 2 • 8 --- keep_class 2 8 • 9 --- keep_class 8 • 10 --- keep_class 6 • 11 --- keep_class 9 • 12 --- keep_class 3 4 5 • 13 --- keep_class 3 • 14 --- keep_class 4 • 15 --- keep_class 5 • 16 --- keep_class 2 6 • 17 --- drop_class 7 • 18 --- drop_withheld • 19 --- drop_synthetic • 20 --- drop_overlap • 21 --- keep_withheld • 22 --- keep_synthetic • 23 --- keep_keypoint • 24 --- keep_overlap
step size / pixel size	STEP	[number] Default: 1.0	Specifies the size of the cells of the grid the TIN is rasterized onto
Attribute	ATTRIBUTE	[enumeration] Default: 0	Specifies the attribute to rasterise. One of: <ul style="list-style-type: none"> • 0 --- elevation • 1 --- intensity • 2 --- rgb • 3 --- classification
Method	METHOD	[enumeration] Default: 0	Specifies how the attributes falling into one cell are turned into a raster value. One of: <ul style="list-style-type: none"> • 0 --- lowest • 1 --- highest • 2 --- average • 3 --- stddev
use tile bounding box (after tiling with buffer)	USE_TILE_BB	[boolean] Default: False	Specifies to limit the rastered area to the tile bounding box (only meaningful for input LAS/LAZ tiles that were created with lastile).
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.

下页继续

表 24.209 – 续上页

Label	名称	Type	Description
Output raster file	OUTPUT_RASTER	[raster] Default: [Skip output]	Specifies where the output raster is stored. Use image rasters like TIF, PNG, and JPG for false color, gray ramps, and hillshades. Use value rasters like TIF, BIL, IMG, ASC, DTM, FLT, XYZ, and CSV for actual values. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output raster file	OUTPUT_RASTER	[raster]	The output raster

Python code

Algorithm ID: lastools:lasgrid

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.11 lasinfo

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAStools GUI	GUI	[boolean] Default: False	Starts the GUI of LAStools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file to get information about.
compute density	COMPUTE_DENSITY	[boolean] Default: False	
repair bounding box	REPAIR_BB	[boolean] Default: False	
repair counters	REPAIR_COUNTERS	[boolean] Default: False	

下页继续

表 24.210 - 续上页

Label	名称	Type	Description
histogram	HISTO1	[enumeration] Default: 0	First histogram. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- x • 2 --- y • 3 --- z • 4 --- intensity • 5 --- classification • 6 --- scan_angle • 7 --- user_data • 8 --- point_source • 9 --- gps_time • 10 --- X • 11 --- Y • 12 --- Z • 13 --- attribute0 • 14 --- attribute1 • 15 --- attribute2
bin size	HISTO1_BIN	[number] Default: 1.0	
histogram	HISTO2	[enumeration] Default: 0	Second histogram. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- x • 2 --- y • 3 --- z • 4 --- intensity • 5 --- classification • 6 --- scan_angle • 7 --- user_data • 8 --- point_source • 9 --- gps_time • 10 --- X • 11 --- Y • 12 --- Z • 13 --- attribute0 • 14 --- attribute1 • 15 --- attribute2
bin size	HISTO2_BIN	[number] Default: 1.0	

下页继续

表 24.210 - 续上页

Label	名称	Type	Description
histogram	HISTO3	[enumeration] Default: 0	Third histogram. One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- x • 2 --- y • 3 --- z • 4 --- intensity • 5 --- classification • 6 --- scan_angle • 7 --- user_data • 8 --- point_source • 9 --- gps_time • 10 --- X • 11 --- Y • 12 --- Z • 13 --- attribute0 • 14 --- attribute1 • 15 --- attribute2
bin size	HISTO3_BIN	[number] Default: 1.0	
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output ASCII file	OUTPUT_GENERIC	[file] Default: [Skip output]	Specifies where the output is stored. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output ASCII file	OUTPUT_GENERIC	[file]	The file with the output

Python code

Algorithm ID: lastools:lasinfo

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.12 lasmerge

Merge up to seven LAS/LAZ files into one.

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAStools GUI	GUI	[boolean] Default: False	Starts the GUI of LAStools with pre-populated input files
files are flightlines	FILES_ARE_FLIGHTLINES	[boolean] Default: False	
apply file source ID	APPLY_FILE_SOURCE_ID	[boolean] Default: False	
input LAS/LAZ file	INPUT_LASLAZ	[file]	The first file containing points to be merged
2nd file Optional	FILE2	[file]	The second file to merge
3rd file Optional	FILE3	[file]	The third file to merge
4th file Optional	FILE4	[file]	The fourth file to merge
5th file Optional	FILE5	[file]	The fifth file to merge
6th file Optional	FILE6	[file]	The sixth file to merge
7th file Optional	FILE7	[file]	The seventh file to merge
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAStools user.
Output LAS/LAZ file	OUTPUT_LASLAZ	[file] Default: [Skip output]	Specifies where the output point cloud is stored. Use LAZ for compressed output, LAS for uncompressed output, and TXT for ASCII. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output LAS/LAZ file	OUTPUT_LASLAZ	[file]	The output (merged) LAS/LAZ format file

Python code

Algorithm ID: lastools:lasmerge

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.13 lasprecision

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
open LAStools GUI	GUI	[boolean] Default: False	Starts the GUI of LAStools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file the input point cloud
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAStools user.
Output ASCII file	OUTPUT_GENERIC	[file] Default: [Skip output]	Specifies where the output ASCII file is stored. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output ASCII file	OUTPUT_GENERIC	[file]	The output ASCII file

Python code

Algorithm ID: lastools:lasprecision

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.3.14 lasquery

Description

<put algorithm description here>

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
open LASStools GUI	GUI	[boolean] Default: False	Starts the GUI of LASStools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file the input point cloud
area of interest	AOI	[extent]	The extent
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LASStools user.

Outputs

Python code

Algorithm ID: lastools:lasquery

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.3.15 激光雷达数据校验 (lasvalidate)

Parameters

Label	名称	Type	Description
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file the input point cloud
save report to '*_LVS.xml'	ONE_REPORT_PER_POINT	[boolean]	
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAStools user.
Output XML file	OUTPUT_GENERIC	[file] Default: [Skip output]	Specifies where the output XML file is stored. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output XML file	OUTPUT_GENERIC	[file]	The output XML file

Python code

Algorithm ID: lastools:lasvalidate

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.16 laszip

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAStools GUI	GUI	[boolean] Default: False	Starts the GUI of LAStools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file to be zipped
only report size	REPORT_SIZE	[boolean] Default: False	

下页继续

表 24.215 - 续上页

Label	名称	Type	Description
create spatial indexing file (*.lax)	CREATE_LAX	[boolean] Default: False	
append *.lax into *.laz file	APPEND_LAX	[boolean] Default: False	
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output LAS/LAZ file	OUTPUT_LASLAZ	[file] Default: [Skip output]	Specifies where the output point cloud is stored. Use LAZ for compressed output, LAS for uncompressed output, and TXT for ASCII. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Output LAS/LAZ file	OUTPUT_LASLAZ	[file]	The output file

Python code

Algorithm ID: lastools:laszip

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

24.3.17 txt2las

Parameters

Label	名称	Type	Description
verbose	VERBOSE	[boolean] Default: False	Generates more textual control output to the console
run new 64 bit executable	CPU64	[boolean] Default: False	
open LAsTools GUI	GUI	[boolean] Default: False	Starts the GUI of LAsTools with pre-populated input files
input LAS/LAZ file	INPUT_LASLAZ	[file]	The file to be zipped
parse lines as	PARSE	[string] Default: 'xyz'	

下页继续

表 24.216 - 续上页

Label	名称	Type	Description
skip the first n lines	SKIP	[number] Default: 0	
resolution of x and y coordinate	SCALE_FACTOR_XY	[number] Default: 0.01	
resolution of z coordinate	SCALE_FACTOR_Z	[number] Default: 0.01	
resolution of z coordinate	SCALE_FACTOR_Z	[number] Default: 0.01	
source projection	PROJECTION	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- --- • 1 --- epsg • 2 --- utm • 3 --- sp83 • 4 --- sp27 • 5 --- longlat • 6 --- latlong • 7 --- ecef
source epsg code	EPSG_CODE	[number]	

下页继续

表 24.216 - 续上页

Label	名称	Type	Description
utm zone	UTM	[enumeration] Default: 0	One of: <ul style="list-style-type: none"> • 0 --- 0 • 1 --- 1 (north) • 2 --- 2 (north) • 3 --- 3 (north) • 4 --- 4 (north) • 5 --- 5 (north) • 6 --- 6 (north) • 7 --- 7 (north) • 8 --- 8 (north) • 9 --- 9 (north) • 10 --- 10 (north) • 11 --- 11 (north) • 12 --- 12 (north) • 13 --- 13 (north) • 14 --- 14 (north) • 15 --- 15 (north) • 16 --- 16 (north) • 17 --- 17 (north) • 18 --- 18 (north) • 19 --- 19 (north) • 20 --- 20 (north) • 21 --- 21 (north) • 22 --- 22 (north) • 23 --- 23 (north) • 24 --- 24 (north) • 25 --- 25 (north) • 26 --- 26 (north) • 27 --- 27 (north) • 28 --- 28 (north) • 29 --- 29 (north) • 30 --- 30 (north) • 31 --- 31 (north) • 32 --- 32 (north) • 33 --- 33 (north) • 34 --- 34 (north) • 35 --- 35 (north) • 36 --- 36 (north) • 37 --- 37 (north) • 38 --- 38 (north) • 39 --- 39 (north) • 40 --- 40 (north) • 41 --- 41 (north) • 42 --- 42 (north) • 43 --- 43 (north) • 44 --- 44 (north) • 45 --- 45 (north) • 46 --- 46 (north) • 47 --- 47 (north) • 48 --- 48 (north) • 49 --- 49 (north) • 50 --- 50 (north) • 51 --- 51 (north) • 52 --- 52 (north) • 53 --- 53 (north) • 54 --- 54 (north) • 55 --- 55 (north) • 56 --- 56 (north)
24.3. LAStools algorithm provider			<ul style="list-style-type: none"> • 57 --- 57 (north) • 58 --- 58 (north) • 59 --- 59 (north) • 60 --- 60 (north) • 61 --- 61 (north)

表 24.216 - 续上页

Label	名称	Type	Description
additional command line parameter(s) Optional	ADDITIONAL_OPTIONS	[string] Default: ""	Specifies other command-line switches not available via this menu but known to the (advanced) LAsTools user.
Output LAS/LAZ file	OUTPUT_LASLAZ	[file] Default: [Skip output]	Specifies where the output point cloud is stored. Use LAZ for compressed output, LAS for uncompressed output, and TXT for ASCII. One of: <ul style="list-style-type: none"> • Skip Output • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
output LAS/LAZ file	OUTPUT_LASLAZ	[file]	The output file

Python code

Algorithm ID: lastools:txt2las

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.4 TauDEM algorithm provider

TauDEM (Terrain Analysis Using Digital Elevation Models) is a set of Digital Elevation Model (DEM) tools for the extraction and analysis of hydrologic information from topography as represented by a DEM. This is software developed at Utah State University (USU) for hydrologic digital elevation model analysis and watershed delineation.

TauDEM is distributed as a set of standalone command line executable programs for a Windows and source code for compiling and use on other systems.

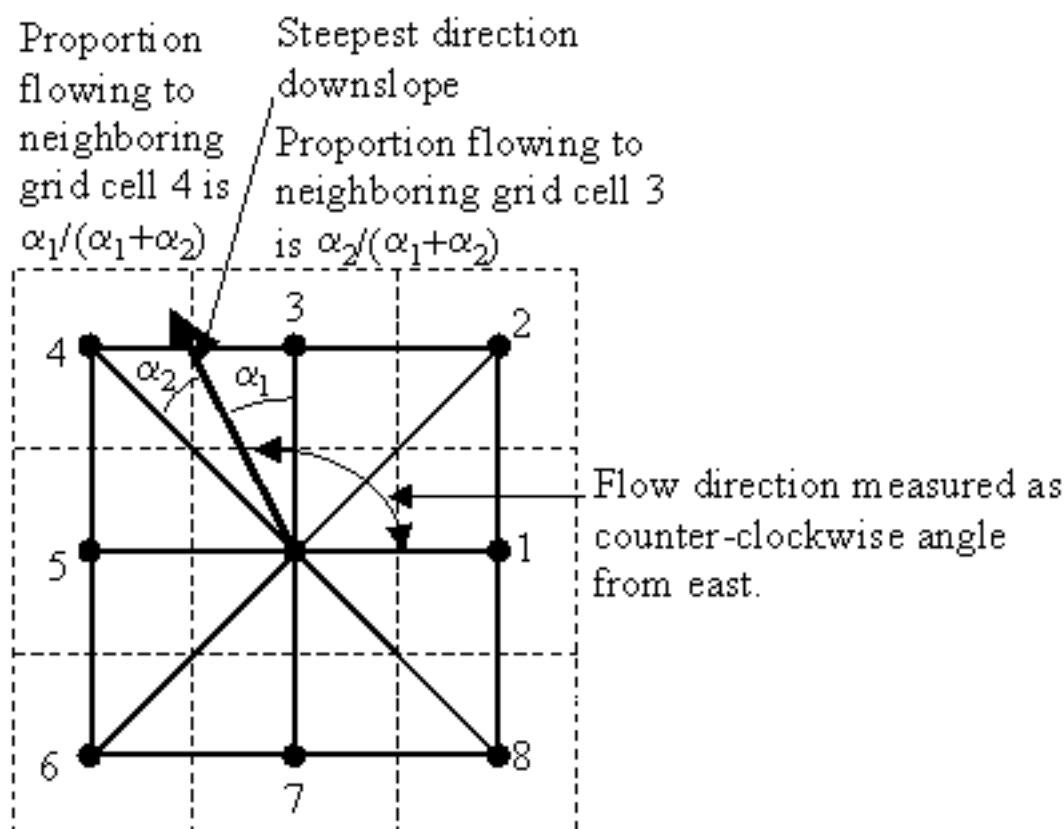
注解: Please remember that Processing contains only the interface description, so you need to install TauDEM 5.0.6 by yourself and configure Processing properly.

Documentation for TauDEM algorithms derived from official [TauDEM documentation](#)

24.4.1 Basic Grid Analysis

D-Infinity Contributing Area

Calculates a grid of specific catchment area which is the contributing area per unit contour length using the multiple flow direction D-infinity approach. D-infinity flow direction is defined as steepest downward slope on planar triangular facets on a block centered grid. The contribution at each grid cell is taken as the grid cell length (or when the optional weight grid input is used, from the weight grid). The contributing area of each grid cell is then taken as its own contribution plus the contribution from upslope neighbors that have some fraction draining to it according to the D-infinity flow model. The flow from each cell either all drains to one neighbor, if the angle falls along a cardinal ($0, \pi/2, \pi, 3\pi/2$) or ordinal ($\pi/4, 3\pi/4, 5\pi/4, 7\pi/4$) direction, or is on an angle falling between the direct angle to two adjacent neighbors. In the latter case the flow is proportioned between these two neighbor cells according to how close the flow direction angle is to the direct angle to those cells. The contour length used here is the grid cell size. The resulting units of the specific catchment area are length units the same as those of the grid cell size.



When the optional weight grid is not used, the result is reported in terms of specific catchment area, the upslope area per unit contour length, taken here as the number of cells times grid cell length (cell area divided by cell length). This assumes that grid cell length is the effective contour length, in the definition of specific catchment area and does not distinguish any difference in contour length dependent upon the flow direction. When the optional weight grid is used, the result is reported directly as a summation of weights, without any scaling.

If the optional outlet point shapefile is used, only the outlet cells and the cells upslope (by the D-infinity flow model) of them are in the domain to be evaluated.

By default, the tool checks for edge contamination. This is defined as the possibility that a contributing area value may be underestimated due to grid cells outside of the domain not being counted. This occurs when drainage is inwards from the boundaries or areas with "no data" values for elevation. The algorithm recognizes this and reports "no data" for the contributing area. It is common to see streaks of "no data" values extending inwards from boundaries along flow paths that enter the domain at a boundary. This is the desired effect and indicates that contributing area for these grid cells is unknown due to it being dependent on terrain outside of the domain of data available. Edge

contamination checking may be turned off in cases where you know it is not an issue or want to ignore these problems, if for example, the DEM has been clipped along a watershed outline.

Parameters

Label	名称	Type	Description
D-infinity flow directions	DINF_FLOWDIR	[raster]	A grid of flow directions based on the D-infinity flow method using the steepest slope of a triangular facet. Flow direction is determined as the direction of the steepest downward slope on the 8 triangular facets of a 3x3 block centered grid. Flow direction is encoded as an angle in radians, counter-clockwise from east as a continuous (floating point) quantity between 0 and 2π . The resulting flow in a grid is then usually interpreted as being proportioned between the two neighboring cells that define the triangular facet with the steepest downward slope.
Outlets Optional	OUTLETS	[vector: point]	A point shapefile defining the outlets of interest. If this input file is used, only the cells upslope of these outlet cells are considered to be within the domain being evaluated.
Weight grid Optional	WEIGHT_GRID	[raster]	A grid giving contribution to flow for each cell. These contributions (also sometimes referred to as weights or loadings) are used in the contributing area accumulation. If this input file is not used, the result is reported in terms of specific catchment area (the upslope area per unit contour length) taken as the number of cells times grid cell length (cell area divided by cell length).

下页继续

表 24.217 - 续上页

Label	名称	Type	Description
Check for edge contamination	EDGE_CONTAMINATION	[boolean] Default: True	A flag that indicates whether the tool should check for edge contamination. Edge contamination is defined as the possibility that a contributing area value may be underestimated due to the fact that grid cells outside of the domain have not been evaluated. This occurs when drainage is inwards from the boundaries or areas with NODATA values for elevation. The algorithm recognizes this and reports NODATA for the impacted cells. It is common to see streaks of NODATA values extending inwards from boundaries along flow paths that enter the domain at a boundary. This is the desired effect and indicates that contributing area for these grid cells is unknown due to it being dependent on terrain outside of the domain of available data. Edge contamination checking may be turned off in cases where you know this is not an issue, or want to ignore these problems, if for example, the DEM has been clipped along a watershed outline.
D-infinity specific catchment area	DINF_CONTRIB_AREA	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
D-infinity specific catchment area	DINF_CONTRIB_AREA	[raster]	A grid of specific catchment area which is the contributing area per unit contour length using the multiple flow direction D-infinity approach. The contributing area of each grid cell is then taken as its own contribution plus the contribution from upslope neighbors that have some fraction draining to it according to the D-infinity flow model.

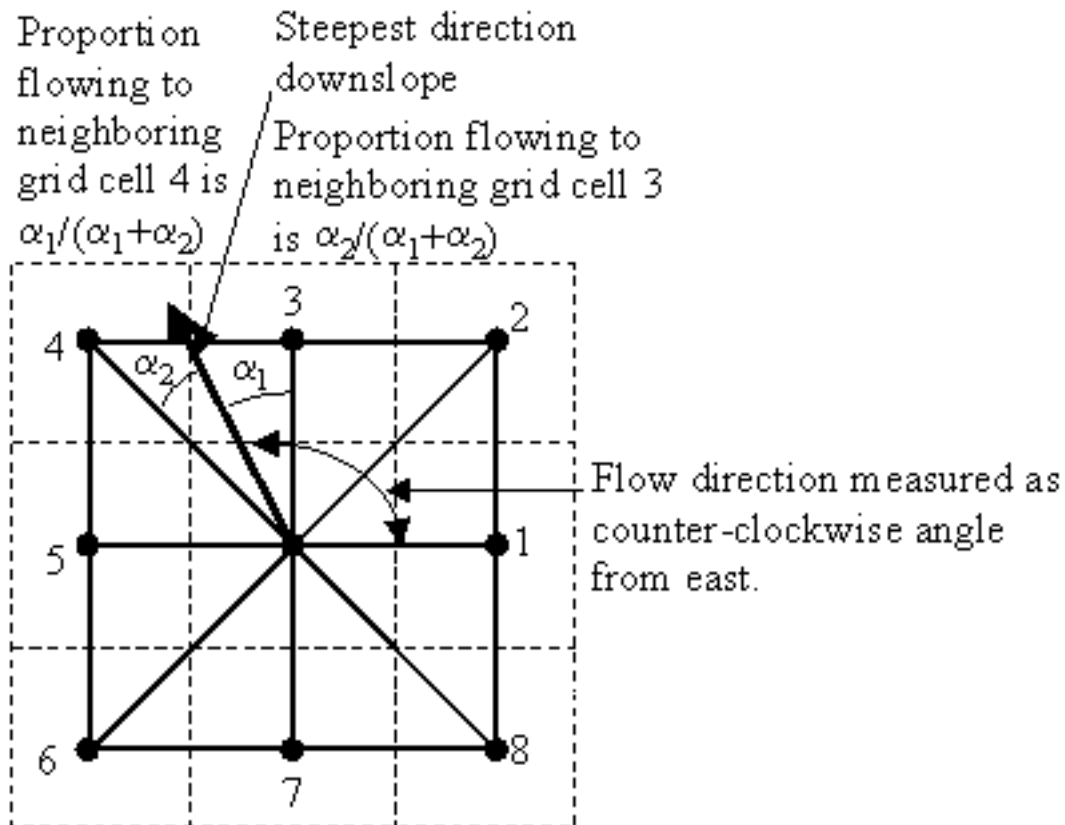
Algorithm ID: taudem:areadinf

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

D-Infinity 流向

Assigns a flow direction based on the D-infinity flow method using the steepest slope of a triangular facet (Tarboton, 1997, "A New Method for the Determination of Flow Directions and Contributing Areas in Grid Digital Elevation Models", Water Resources Research, 33(2): 309-319). Flow direction is defined as steepest downward slope on planar triangular facets on a block centered grid. Flow direction is encoded as an angle in radians counter-clockwise from east as a continuous (floating point) quantity between 0 and 2π . The flow direction angle is determined as the direction of the steepest downward slope on the eight triangular facets formed in a 3 x 3 grid cell window centered on the grid cell of interest. The resulting flow in a grid is then usually interpreted as being proportioned between the two neighboring cells that define the triangular facet with the steepest downward slope.



A block-centered representation is used with each elevation value taken to represent the elevation of the center of the corresponding grid cell. Eight planar triangular facets are formed between each grid cell and its eight neighbors. Each of these has a downslope vector which when drawn outwards from the center may be at an angle that lies within or outside the 45 degree ($\pi/4$ radian) angle range of the facet at the center point. If the slope vector angle is within the facet angle, it represents the steepest flow direction on that facet. If the slope vector angle is outside a facet, the steepest flow direction associated with that facet is taken along the steepest edge. The slope and flow direction associated with the grid cell is taken as the magnitude and direction of the steepest downslope vector from all eight facets. Slope is measured as drop/distance, i.e. tan of the slope angle.

In the case where no slope vectors are positive (downslope), the flow direction is set using the method of Garbrecht and Martz (1997) for the determination of flow across flat areas. This makes flat areas drain away from high ground and towards low ground. The flow path grid to enforce drainage along existing streams is an optional input, and if used, takes precedence over elevations for the setting of flow directions.

The D-infinity flow direction algorithm may be applied to a DEM that has not had its pits filled, but it will then result in "no data" values for the D-infinity flow direction and slope associated with the lowest point of the pit.

Parameters

Label	名称	Type	Description
Pit filled elevation	PIT_FILLED	[raster]	A grid of elevation values. This is usually the output of the "Pit Remove" tool, in which case it is elevations with pits removed. Pits are low elevation areas in digital elevation models (DEMs) that are completely surrounded by higher terrain. They are generally taken to be artifacts of the digitation process that interfere with the processing of flow across DEMs. So they are removed by raising their elevation to the point where they just drain off the domain. This step is not essential if you have reason to believe that the pits in your DEM are real. If a few pits actually exist and so should not be removed, while at the same time others are believed to be artifacts that need to be removed, the actual pits should have NODATA elevation values inserted at their lowest point. NODATA values serve to define edges of the domain in the flow field, and elevations are only raised to where flow is off an edge, so an internal NODATA value will stop a pit from being removed, if necessary.
D-infinity flow directions	DINF_FLOWDIR	[raster] Default: [Save to temporary file]	Specification of the output flow direction raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.
D-infinity slope	DINF_SLOPE	[raster] Default: [Save to temporary file]	Specification of the output slope raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
D-infinity flow directions	DINF_FLOWDIR	[raster]	A grid of flow directions based on the D-infinity flow method using the steepest slope of a triangular facet. Flow direction is determined as the direction of the steepest downward slope on the 8 triangular facets of a 3x3 block centered grid. Flow direction is encoded as an angle in radians, counter-clockwise from east as a continuous (floating point) quantity between 0 and 2π . The resulting flow in a grid is then usually interpreted as being proportioned between the two neighboring cells that define the triangular facet with the steepest downward slope.
D-infinity slope	DINF_SLOPE	[raster]	A grid of slope evaluated using the D-infinity method described in Tarboton, D. G., (1997), "A New Method for the Determination of Flow Directions and Contributing Areas in Grid Digital Elevation Models", <i>Water Resources Research</i> , 33(2): 309-319. This is the steepest outwards slope on one of eight triangular facets centered at each grid cell, measured as drop/distance, i.e. tan of the slope angle.

Algorithm ID: taudem:dinfflowdir

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

D8 Contributing Area

Calculates a grid of contributing areas using the single direction D8 flow model. The contribution of each grid cell is taken as one (or when the optional weight grid is used, the value from the weight grid). The contributing area for each grid cell is taken as its own contribution plus the contribution from upslope neighbors that drain in to it according to the D8 flow model.

If the optional outlet point shapefile is used, only the outlet cells and the cells upslope (by the D8 flow model) of them are in the domain to be evaluated.

By default, the tool checks for edge contamination. This is defined as the possibility that a contributing area value may be underestimated due to grid cells outside of the domain not being counted. This occurs when drainage is inwards from the boundaries or areas with "no data" values for elevation. The algorithm recognizes this and reports "no data" for the contributing area. It is common to see streaks of "no data" values extending inwards from boundaries along flow paths that enter the domain at a boundary. This is the desired effect and indicates that contributing area for these grid cells is unknown due to it being dependent on terrain outside of the domain of data available. Edge contamination checking may be turned off in cases where you know this is not an issue or want to ignore these problems, if for example, the DEM has been clipped along a watershed outline.

Parameters

Label	名称	Type	Description
D8 flow directions	D8_FLOWDIR	[raster]	A grid of D8 flow directions which are defined, for each cell, as the direction of the one of its eight adjacent or diagonal neighbors with the steepest downward slope. This grid can be obtained as the output of the "D8 Flow Directions" tool.
Outlets Optional	OUTLETS	[vector: point]	A point shapefile defining the outlets of interest. If this input file is used, only the cells upslope of these outlet cells are considered to be within the domain being evaluated.
Weight grid Optional	WEIGHT_GRID	[raster]	A grid giving contribution to flow for each cell. These contributions (also sometimes referred to as weights or loadings) are used in the contributing area accumulation. If this input file is not used, the contribution to flow will assumed to be one for each grid cell.
Check for edge contamination	EDGE_CONTAMINATION	[boolean] Default: True	A flag that indicates whether the tool should check for edge contamination. Edge contamination is defined as the possibility that a contributing area value may be underestimated due to the fact that grid cells outside of the domain have not been evaluated. This occurs when drainage is inwards from the boundaries or areas with NODATA values for elevation. The algorithm recognizes this and reports NODATA for the impacted cells. It is common to see streaks of NODATA values extending inwards from boundaries along flow paths that enter the domain at a boundary. This is the desired effect and indicates that contributing area for these grid cells is unknown due to it being dependent on terrain outside of the domain of available data. Edge contamination checking may be turned off in cases where you know this is not an issue, or want to ignore these problems, if for example, the DEM has been clipped along a watershed outline.
D8 specific catchment area	D8_CONTRIB_AREA	[raster] Default: [Save to temporary file]	Specification of the output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
D8 specific catchment area	D8_CONTRIB_AREA	[raster]	A grid of contributing area values calculated as the cells own contribution plus the contribution from upslope neighbors that drain in to it according to the D8 flow model.

Algorithm ID: taudem:aread8

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

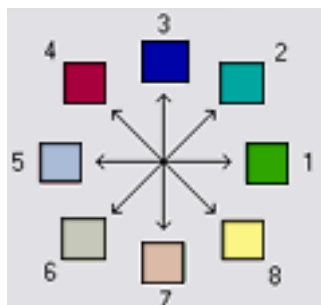
The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

D8 Flow Directions

Creates 2 grids. The first contains the flow direction from each grid cell to one of its adjacent or diagonal neighbors, calculated using the direction of steepest descent. The second contain the slope, as evaluated in the direction of steepest descent, and is reported as drop/distance, i.e. \tan of the angle. Flow direction is reported as NODATA for any grid cell adjacent to the edge of the DEM domain, or adjacent to a NODATA value in the DEM. In flat areas, flow directions are assigned away from higher ground and towards lower ground using the method of Garbrecht and Martz (1997). The D8 flow direction algorithm may be applied to a DEM that has not had its pits filled, but it will then result in NODATA values for flow direction and slope at the lowest point of each pit.

D8 Flow Direction Coding:

- 1 --- East
- 2 --- Northeast
- 3 --- North
- 4 --- Northwest
- 5 --- West
- 6 --- Southwest
- 7 --- South
- 8 --- Southeast



The flow direction routing across flat areas is performed according to the method described by Garbrecht, J. and L. W. Martz, (1997), "The Assignment of Drainage Direction Over Flat Surfaces in Raster Digital Elevation Models", Journal of Hydrology, 193: 204-213.

Parameters

Label	名称	Type	Description
Pit filled elevation	PIT_FILLED	[raster]	A grid of elevation values. This is usually the output of the "Pit Remove" tool, in which case it is elevations with pits removed. Pits are low elevation areas in digital elevation models (DEMs) that are completely surrounded by higher terrain. They are generally taken to be artifacts of the digitization process that interfere with the processing of flow across DEMs. So they are removed by raising their elevation to the point where they just drain off the domain. This step is not essential if you have reason to believe that the pits in your DEM are real. If a few pits actually exist and so should not be removed, while at the same time others are believed to be artifacts that need to be removed, the actual pits should have NODATA elevation values inserted at their lowest point. NODATA values serve to define edges of the domain in the flow field, and elevations are only raised to where flow is off an edge, so an internal NODATA value will stop a pit from being removed, if necessary.
D8 flow directions	D8_FLOWDIR	[raster] Default: [Save to temporary file]	Specification of the output flow direction raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.
D8 slope	D8_SLOPE	[raster] Default: [Save to temporary file]	Specification of the output slope raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
D8 flow directions	D8_FLOWDIR	[raster]	A grid of D8 flow directions which are defined, for each cell, as the direction of the one of its eight adjacent or diagonal neighbors with the steepest downward slope.
D8 slope	D8_SLOPE	[raster]	A grid giving slope in the D8 flow direction. This is measured as drop/distance.

Algorithm ID: taudem:d8flowdir

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to

run processing algorithms from the Python console.

Grid Network

Creates 3 grids that contain for each grid cell: 1) the longest path, 2) the total path, and 3) the Strahler order number. These values are derived from the network defined by the D8 flow model.

The longest upslope length is the length of the flow path from the furthest cell that drains to each cell. The total upslope path length is the length of the entire grid network upslope of each grid cell. Lengths are measured between cell centers taking into account cell size and whether the direction is adjacent or diagonal.

Strahler order is defined as follows: A network of flow paths is defined by the D8 Flow Direction grid. Source flow paths have a Strahler order number of one. When two flow paths of different order join the order of the downstream flow path is the order of the highest incoming flow path. When two flow paths of equal order join the downstream flow path order is increased by 1. When more than two flow paths join the downstream flow path order is calculated as the maximum of the highest incoming flow path order or the second highest incoming flow path order + 1. This generalizes the common definition to cases where more than two flow paths join at a point.

Where the optional mask grid and threshold value are input, the function is evaluated only considering grid cells that lie in the domain with mask grid value greater than or equal to the threshold value. Source (first order) grid cells are taken as those that do not have any other grid cells from inside the domain draining in to them, and only when two of these flow paths join is order propagated according to the ordering rules. Lengths are also only evaluated counting paths within the domain greater than or equal to the threshold.

If the optional outlet point shapefile is used, only the outlet cells and the cells upslope (by the D8 flow model) of them are in the domain to be evaluated.

Parameters

Label	名称	Type	Description
D8 flow directions	D8_FLOWDIR	[raster]	A grid of D8 flow directions which are defined, for each cell, as the direction of the one of its eight adjacent or diagonal neighbors with the steepest downward slope. This grid can be obtained as the output of the "D8 Flow Directions" tool.
Mask Grid Optional	MASK_GRID	[raster]	A grid that is used to determine the domain to be analyzed. If the mask grid value \geq mask threshold (see below), then the cell will be included in the domain. While this tool does not have an edge contamination flag, if edge contamination analysis is needed, then a mask grid from a function like "D8 Contributing Area" that does support edge contamination can be used to achieve the same result.
Mask threshold Optional	THRESHOLD	[number] Default: 100.0	This input parameter is used in the calculation mask grid value \geq mask threshold to determine if the grid cell is in the domain to be analyzed.
Outlets Optional	OUTLETS	[vector: point]	A point shapefile defining the outlets of interest. If this input file is used, only the cells upslope of these outlet cells are considered to be within the domain being evaluated.

下页继续

表 24.222 - 续上页

Label	名称	Type	Description
Longest upslope length	LONGEST_PATH	[raster] Default: [Save to temporary file]	Specification of the output raster with total upslope lengths. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.
Total upslope length	TOTAL_PATH	[raster] Default: [Save to temporary file]	Specification of the output raster with upslope lengths. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.
Strahler network order	STRAHLER_ORDER	[raster] Default: [Save to temporary file]	Specification of the output raster with Strahler network order. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Longest upslope length	LONGEST_PATH	[raster]	A grid that gives the length of the longest upslope D8 flow path terminating at each grid cell. Lengths are measured between cell centers taking into account cell size and whether the direction is adjacent or diagonal.
Total upslope length	TOTAL_PATH	[raster]	The total upslope path length is the length of the entire D8 flow grid network upslope of each grid cell. Lengths are measured between cell centers taking into account cell size and whether the direction is adjacent or diagonal.
Strahler network order	STRAHLER_ORDER	[raster]	A grid giving the Strahler order number for each cell. A network of flow paths is defined by the D8 Flow Direction grid. Source flow paths have a Strahler order number of one. When two flow paths of different order join the order of the downstream flow path is the order of the highest incoming flow path. When two flow paths of equal order join the downstream flow path order is increased by 1. When more than two flow paths join the downstream flow path order is calculated as the maximum of the highest incoming flow path order or the second highest incoming flow path order + 1. This generalizes the common definition to cases where more than two flow paths join at a point.

Algorithm ID: taudem:gridnet

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Pit Remove

Identifies all pits in the DEM and raises their elevation to the level of the lowest pour point around their edge. Pits are low elevation areas in digital elevation models (DEMs) that are completely surrounded by higher terrain. They are generally taken to be artifacts that interfere with the routing of flow across DEMs, so are removed by raising their elevation to the point where they drain off the edge of the domain. The pour point is the lowest point on the boundary of the "watershed" draining to the pit. This step is not essential if you have reason to believe that the pits in your DEM are real. If a few pits actually exist and so should not be removed, while at the same time others are believed to be artifacts that need to be removed, the actual pits should have NODATA elevation values inserted at their lowest point. NODATA values serve to define edges in the domain, and elevations are only raised to where flow is off an edge, so an internal NODATA value will stop a pit from being removed, if necessary.

Parameters

Label	名称	Type	Description
Elevation	ELEVATION	[raster]	A digital elevation model (DEM) grid to serve as the base input for the terrain analysis and stream delineation.
Depression mask Optional	DEPRESSION_MASK	[raster]	
Consider only 4 way neighbors	FOUR_NEIGHBOURS	[boolean] Default: False	
Pit removed elevation	PIT_FILLED	[raster] Default: [Save to temporary file]	Specification of the (pit filled) output raster. One of: <ul style="list-style-type: none"> • Save to a Temporary File • Save to File... The file encoding can also be changed here.

Outputs

Label	名称	Type	Description
Pit removed elevation	PIT_FILLED	[raster]	A grid of elevation values with pits removed so that flow is routed off of the domain.

Algorithm ID: taudem:pitremove

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.4.2 Specialized Grid Analysis

D8 Distance To Streams

Computes the horizontal distance to stream for each grid cell, moving downslope according to the D8 flow model, until a stream grid cell is encountered.

Parameters

Label	Name	Type	Description
D8 Flow Direction Grid		[raster]	This input is a grid of flow directions that are encoded using the D8 method where all flow from a cells goes to a single neighboring cell in the direction of steepest descent. This grid can be obtained as the output of the "D8 Flow Directions" tool.
Stream Raster Grid		[raster]	A grid indicating streams. Such a grid can be created by several of the tools in the "Stream Network Analysis" toolset. However, the tools in the "Stream Network Analysis" toolset only create grids with a value of 0 for no stream, or 1 for stream cells. This tool can also accept grids with values greater than 1, which can be used in conjunction with the <code>Threshold</code> parameter to determine the location of streams. This allows Contributing Area grids to be used to define streams as well as the normal Stream Raster grids. This grid expects integer (long integer) values and any non-integer values will be truncated to an integer before being evaluated.
Threshold		[number] Default: 50	This value acts as threshold on the <code>Stream Raster Grid</code> to determine the location of streams. Cells with a <code>Stream Raster Grid</code> value greater than or equal to the <code>Threshold</code> value are interpreted as streams.

Outputs

Label	Name	Type	Description
Output Distance to Streams		[raster]	A grid giving the horizontal distance along the flow path as defined by the D8 Flow Directions Grid to the streams in the Stream Raster Grid.

Python code

Algorithm ID: `taudem:d8hdisttostrm`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

D-Infinity Avalanche Runout

Identifies an avalanche’s affected area and the flow path length to each cell in that affected area. All cells downslope from each source area cell, up to the point where the slope from the source to the affected area is less than a threshold angle called the Alpha Angle can be in the affected area. This tool uses the D-infinity multiple flow direction method for determining flow direction. This will likely cause very small amounts of flow to be dispersed to some downslope cells that might overstate the affected area, so a threshold proportion can be set to avoid this excess dispersion. The flow path length is the distance from the cell in question to the source cell that has the highest angle.

All points downslope from the source area are potentially in the affected area, but not beyond a point where the slope from the source to the affected area is less than a threshold angle called the Alpha Angle.

Elevations						
10	10	10	10	10	10	Yellow cell is the source
10	9	9	9	9	10	Green: downslope of source
10	9	8	7	6.99	10	
10	9	9	8	6.98	10	
10	9	8	7	6.97	10	
10	10	10	10	6.96	10	
Straight-line distance from highest point of source						
0	1	2	3	4	5	Yellow cell is the source
1	1.414214	2.236068	3.162278	4.123106	5.09902	Green: downslope of source
2	2.236068	2.828427	3.605551	4.472136	5.385165	
3	3.162278	3.605551	4.242641	5	5.830952	
4	4.123106	4.472136	5	5.656854	6.403124	
5	5.09902	5.385165	5.830952	6.403124	7.071068	
Drop in elevation from highest point in source						
0	0	0	0	0	0	Yellow cell is the source
0	1	1	1	1	0	Green: downslope of source
0	1	2	3	3.01	0	
0	1	1	2	3.02	0	
0	1	2	3	3.03	0	
0	0	0	0	3.04	0	
<p>2 The cell size (a fiddle factor for me to make sensible values)</p> <p>18 The threshold angle for being in the runout zone</p>						
The slope angle from the highest point in the source to each cell						
0	0	0	0	0	0	Yellow cell is the source
0	19	13	9	7	0	Green: downslope of source
0	13	19	23	19	0	Grey cells are BOTH
0	9	8	13	17	0	downslope of the source AND
0	7	13	17	15	0	have a sufficiently steep
0	0	0	0	13	0	angle to be in the runout zone

Slope is to be measured using the straight line distance from source point to evaluation point.

It makes more physical sense to me for the angle to be measured along the flow path. Nevertheless it is equally easy to code straight line angles as angles along the flow path, so an option that allows switching will be provided. The most practical way to evaluate avalanche runout is to keep track of the source point with the greatest angle to each point. Then the recursive upslope flow algebra approach will look at a grid cell and all its upslope neighbors that flow to it. Information from the upslope neighbors will be used to calculate the angle to the grid cell in question and retain it in the runout zone if the angle exceeds the alpha angle. This procedure makes the assumption that the maximum angle at a grid cell will be from the set of cells that have maximum angles to the inflowing neighbors. This will always be true of angle is calculated along a flow path, but I can conceive of cases where flow paths bend back on themselves where this would not be the case for straight line angles.

The D-infinity multiple flow direction field assigns flow from each grid cell to multiple downslope neighbors using

proportions (P_{ik}) that vary between 0 and 1 and sum to 1 for all flows out of a grid cell. It may be desirable to specify a threshold T that this proportion has to exceed before a grid cell is counted as flowing to a downslope grid cell, e.g. $P_{ik} > T$ ($=0.2$ say) to avoid dispersion to grid cells that get very little flow. T will be specified as a user input. If all upslope grid cells are to be used T may be input as 0.

Avalanche source sites are to be input as a short integer grid (name suffix **ass*, e.g. *demass*) comprised of positive values where avalanches may be triggered and 0 values elsewhere.

The following grids are output:

- *rz* --- A runout zone indicator with value 0 to indicate that this grid cell is not in the runout zone and value > 0 to indicate that this grid cell is in the runout zone. Since there may be information in the angle to the associated source site, this variable will be assigned the angle to the source site (in degrees)
- *dm* --- Along flow distance from the source site that has the highest angle to the point in question

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-infinity method. Flow direction is measured in radians, counter clockwise from east. This can be created by the tool "D-Infinity Flow Directions" .
Pit Filled Elevation Grid		[raster]	This input is a grid of elevation values. As a general rule, it is recommended that you use a grid of elevation values that have had the pits removed for this input. Pits are generally taken to be artifacts that interfere with the analysis of flow across them. This grid can be obtained as the output of the "Pit Remove" tool, in which case it contains elevation values where the pits have been filled to the point where they just drain.
Avalanche Source Site Grid		[raster]	This is a grid of source areas for snow avalanches that are commonly identified manually using a mix of experience and visual interpretation of maps. Avalanche source sites are to be input as a short integer grid (name suffix <i>*ass</i> , e.g. <i>demass</i>) comprised of positive values where avalanches may be triggered and 0 values elsewhere.
Proportion Threshold		[number] Default: 0.2	This value is a threshold proportion that is used to limit the dispersion of flow caused by using the D-infinity multiple flow direction method for determining flow direction. The D-infinity multiple flow direction method often causes very small amounts of flow to be dispersed to some downslope cells that might overstate the affected area, so a threshold proportion can be set to avoid this excess dispersion.
Alpha Angle Threshold		[number] Default: 18	This value is the threshold angle, called the Alpha Angle, that is used to determine which of the cells downslope from the source cells are in the affected area. Only the cells downslope from each source area cell, up to the point where the slope from the source to the affected area is less than a threshold angle are in the affected area.
Measure distance along flow path		[boolean] Default: True	This option selects the method used to measure the distance used to calculate the slope angle. If option is <i>True</i> then measure it along the flow path, where the <i>False</i> option causes the slope to be measure along the straight line distance from the source cell to the evaluation cell.

Outputs

Label	Name	Type	Description
Runout Zone Grid		[raster]	This grid Identifies the avalanche’s runout zone (affected area) using a runout zone indicator with value 0 to indicate that this grid cell is not in the runout zone and value > 0 to indicate that this grid cell is in the runout zone. Since there may be information in the angle to the associated source site, this variable will be assigned the angle to the source site (in degrees).
Path Distance Grid		[raster]	This is a grid of the flow distance from the source site that has the highest angle to each cell.

Python code

Algorithm ID: taudem:dinfaavalanche

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

D-Infinity Concentration Limited Accumulation

This function applies to the situation where an unlimited supply of a substance is loaded into flow at a concentration or solubility threshold C_{sol} over a region indicated by an indicator grid (dg). It a grid of the concentration of a substance at each location in the domain, where the supply of substance from a supply area is loaded into the flow at a concentration or solubility threshold. The flow is first calculated as a D-infinity weighted contributing area of an input Effective Runoff Weight Grid (notionally excess precipitation). The concentration of substance over the supply area (indicator grid) is at the concentration threshold. As the substance moves downslope with the D-infinity flow field, it is subject to first order decay in moving from cell to cell as well as dilution due to changes in flow. The decay multiplier grid gives the fractional (first order) reduction in quantity in moving from grid cell x to the next downslope cell. If the outlets shapefile is used, the tool only evaluates the part of the domain that contributes flow to the locations given by the shapefile. This is useful for a tracking a contaminant or compound from an area with unlimited supply of that compound that is loaded into a flow at a concentration or solubility threshold over a zone and flow from the zone may be subject to decay or attenuation.

The indicator grid (dg) is used to delineate the area of the substance supply using the (0, 1) indicator function $i(x)$. $A[]$ denotes the weighted accumulation operator evaluated using the D-Infinity Contributing Area function. The Effective Runoff Weight Grid gives the supply to the flow (e.g. the excess rainfall if this is overland flow) denoted as $w(x)$. The specific discharge is then given by:

$$Q(x) = A[w(x)]$$

This weighted accumulation $Q(x)$ is output as the Overland Flow Specific Discharge Grid. Over the substance supply area concentration is at the threshold (the threshold is a saturation or solubility limit). If $i(x) = 1$, then

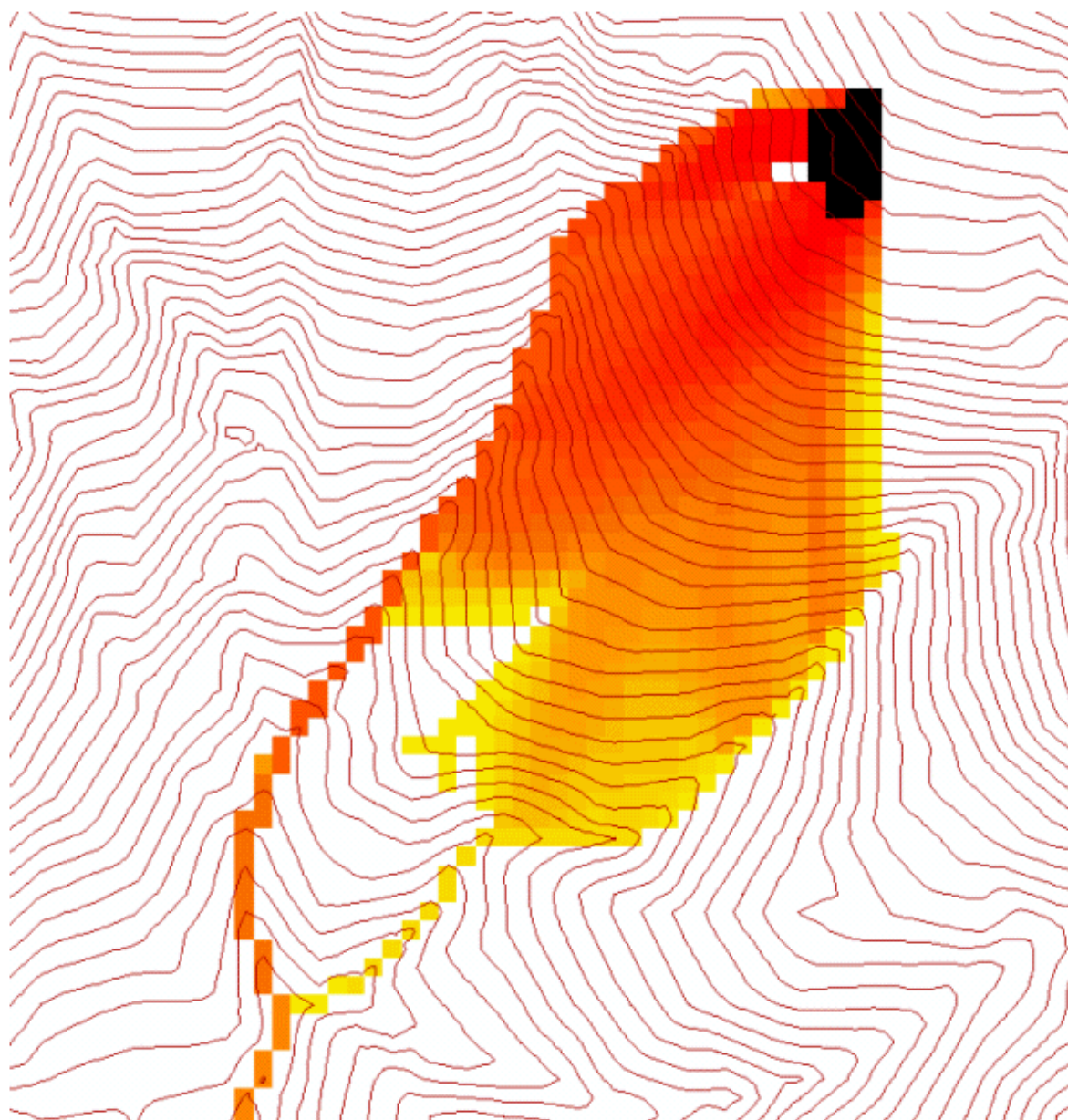
$$C(x) = C_{sol}, \text{ and } L(x) = C_{sol} Q(x),$$

where $L(x)$ denotes the load being carried by the flow. At remaining locations, the load is determined by load accumulation and the concentration by dilution:

$$L(x) = L(i, j) = \sum_{k \text{ contributing neighbors}} p_k d(i_k, j_k) L(i_k, j_k)$$

$$C(x) = L(x)/Q(x)$$

Here $d(x) = d(i, j)$ is a decay multiplier giving the fractional (first order) reduction in mass in moving from grid cell x to the next downslope cell. If travel (or residence) times $t(x)$ associated with flow between cells are available $d(x)$ may be evaluated as $\exp(-k t(x))$ where k is a first order decay parameter. The Concentration grid output is $C(x)$. If the outlets shapefile is used, the tool only evaluates the part of the domain that contributes flow to the locations given by the shapefile.



Useful for a tracking a contaminant released or partitioned to flow at a fixed threshold concentration.

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-infinity method. Flow direction is measured in radians, counter clockwise from east. This grid can be created by the function "D-Infinity Flow Directions" .
Disturbance Indicator Grid		[raster]	A grid that indicates the source zone of the area of substance supply and must be 1 inside the zone and 0 or NODATA over the rest of the domain.
Decay Multiplier Grid		[raster]	A grid giving the factor by which flow leaving each grid cell is multiplied before accumulation on downslope grid cells. This may be used to simulate the movement of an attenuating or decaying substance. If travel (or residence) times $t(x)$ associated with flow between cells are available $d(x)$ may be evaluated as $\exp(-k t(x))$ where k is a first order decay parameter.
Effective Runoff Weight Grid		[raster]	A grid giving the input quantity (notionally effective runoff or excess precipitation) to be used in the D-infinity weighted contributing area evaluation of Overland Flow Specific Discharge.
Outlets shapefile Optional		[vector: point]	This optional input is a point shapefile defining outlets of interest. If this file is used, the tool will only evaluate the area upslope of these outlets.
Concentration Threshold		[number] Default: 1.0	The concentration or solubility threshold. Over the substance supply area, concentration is at this threshold.
Check for edge contamination		[boolean] Default: True	This option determines whether the tool should check for edge contamination. Edge contamination is defined as the possibility that a value may be underestimated due to grid cells outside of the domain not being considered when determining contributing area.

Outputs

Label	Name	Type	Description
Concentration Grid		[raster]	A grid giving the resulting concentration of the compound of interest in the flow.

Python code

Algorithm ID: taudem:dinfconclimaccum

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

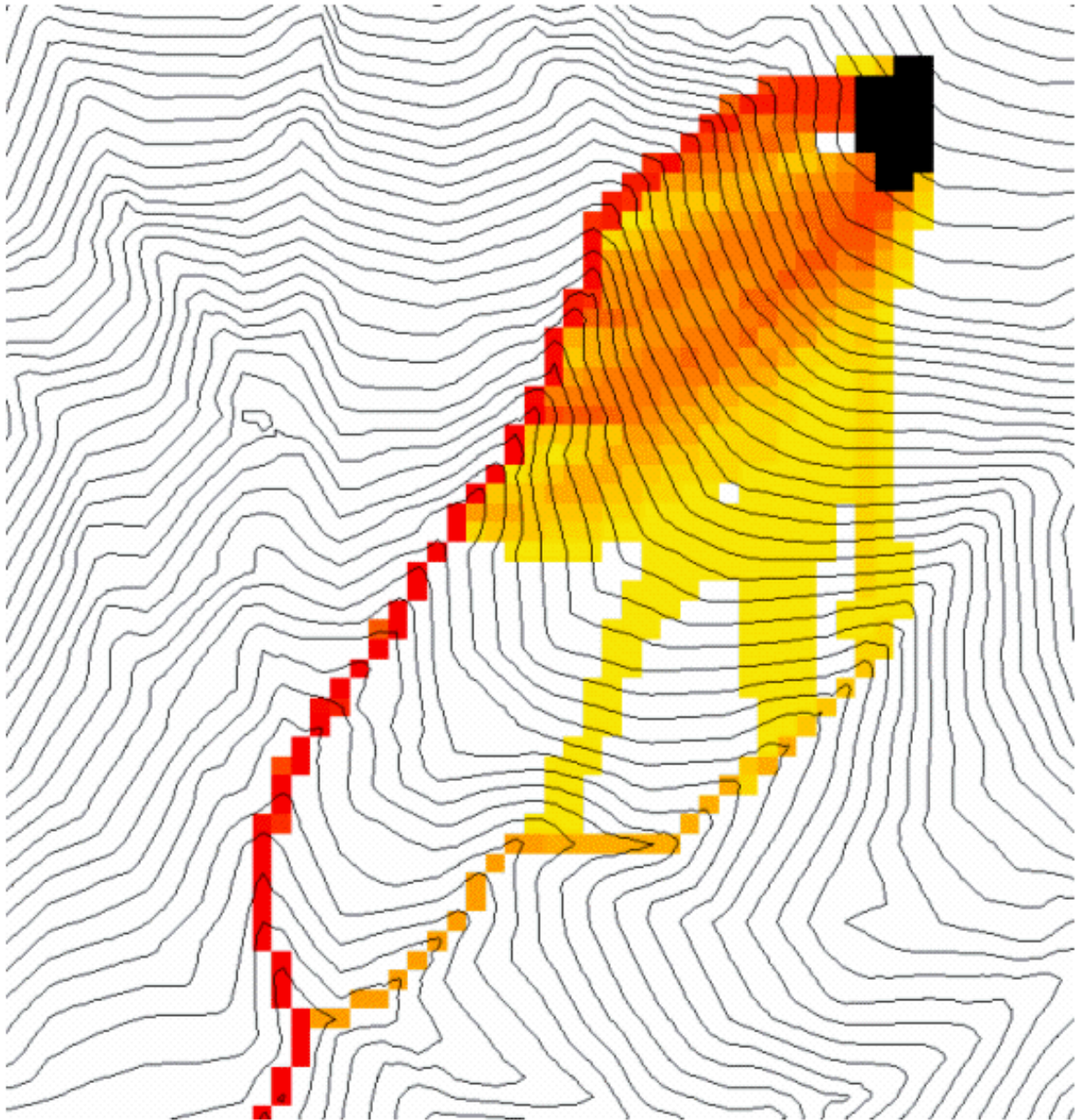
D-Infinity Decaying Accumulation

The D-Infinity Decaying Accumulation tool creates a grid of the accumulated quantity at each location in the domain where the quantity accumulates with the D-infinity flow field, but is subject to first order decay in moving from cell to cell. By default, the quantity contribution of each grid cell is the cell length to give a per unit width accumulation, but can optionally be expressed with a weight grid. The decay multiplier grid gives the fractional (first order) reduction in quantity in accumulating from grid cell x to the next downslope cell.

A decayed accumulation operator $DA[.]$ takes as input a mass loading field $m(x)$ expressed at each grid location as $m(i, j)$ that is assumed to move with the flow field but is subject to first order decay in moving from cell to cell. The output is the accumulated mass at each location $DA(x)$. The accumulation of m at each grid cell can be numerically evaluated.

$$DA[m(x)] = DA(i, j) = m(i, j) \Delta^2 + \sum_{k \text{ contributing neighbors}} p_k d(i_k, j_k) DA(i_k, j_k)$$

Here $d(x) = d(i, j)$ is a decay multiplier giving the fractional (first order) reduction in mass in moving from grid cell x to the next downslope cell. If travel (or residence) times $t(x)$ associated with flow between cells are available $d(x)$ may be evaluated as $\exp(-k t(x))$ where k is a first order decay parameter. The weight grid is used to represent the mass loading $m(x)$. If not specified this is taken as 1. If the outlets shapefile is used the function is only evaluated on that part of the domain that contributes flow to the locations given by the shapefile.



Useful for a tracking contaminant or compound subject to decay or attenuation.

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-infinity method. Flow direction is measured in radians, counter clock-wise from east. This grid can be created by the function "D-Infinity Flow Directions" .
Decay Multiplier Grid		[raster]	A grid giving the factor by which flow leaving each grid cell is multiplied before accumulation on downslope grid cells. This may be used to simulate the movement of an attenuating substance.
Weight Grid Optional		[raster]	A grid giving weights (loadings) to be used in the accumulation. If this optional grid is not specified, weights are taken as the linear grid cell size to give a per unit width accumulation.
Outlets Shapefile Optional		[vector: point]	This optional input is a point shapefile defining outlets of interest. If this file is used, the tool will only evaluate the area upslope of these outlets.
Check for edge contamination		[boolean] Default: True	This option determines whether the tool should check for edge contamination. Edge contamination is defined as the possibility that a value may be underestimated due to grid cells outside of the domain not being considered when determining contributing area.

Outputs

Label	Name	Type	Description
Decayed Specific Catchment Area Grid		[raster]	The D-Infinity Decaying Accumulation tool creates a grid of the accumulated mass at each location in the domain where mass moves with the D-infinity flow field, but is subject to first order decay in moving from cell to cell.

Python code

Algorithm ID: taudem:dinfdecayaccum

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

D-Infinity Distance Down

Calculates the distance downslope to a stream using the D-infinity flow model. The D-infinity flow model is a multiple flow direction model, because the outflow from each grid cell is proportioned between up to 2 downslope grid cells. As such, the distance from any grid cell to a stream is not uniquely defined. Flow that originates at a particular grid cell may enter the stream at a number of different cells. The statistical method may be selected as the longest, shortest or weighted average of the flow path distance to the stream. Also one of several ways of measuring distance may be selected: the total straight line path (Pythagoras), the horizontal component of the straight line path, the vertical component of the straight line path, or the total surface flow path.

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-infinity method. Flow direction is measured in radians, counter clockwise from east. This can be created by the tool "D-Infinity Flow Directions" .
Pit Filled Elevation Grid		[raster]	This input is a grid of elevation values. As a general rule, it is recommended that you use a grid of elevation values that have had the pits removed for this input. Pits are generally taken to be artifacts that interfere with the analysis of flow across them. This grid can be obtained as the output of the "Pit Remove" tool, in which case it contains elevation values where the pits have been filled to the point where they just drain.
Stream Raster Grid		[raster]	A grid indicating streams, by using a grid cell value of 1 on streams and 0 off streams. This is usually the output of one of the tools in the "Stream Network Analysis" toolset.
Weight Path Grid Optional		[raster]	A grid giving weights (loadings) to be used in the distance calculation. This might be used for example where only flow distance through a buffer is to be calculated. The weight is then 1 in the buffer and 0 outside it. Alternatively the weight may reflect some sort of cost function for travel over the surface, perhaps representing travel time or attenuation of a process. If this input file is not used, the loadings will assumed to be one for each grid cell.
Statistical Method		[enumeration] Default: 2	Statistical method used to calculate the distance down to the stream. In the D-Infinity flow model, the outflow from each grid cell is proportioned between two downslope grid cells. Therefore, the distance from any grid cell to a stream is not uniquely defined. Flow that originates at a particular grid cell may enter the stream at a number of cells. The distance to the stream may be defined as the longest (maximum), shortest (minimum) or weighted average of the distance down to the stream. Options: <ul style="list-style-type: none"> • 0 --- Minimum • 1 --- Maximum • 2 --- Average
Distance Method		[enumeration] Default: 1	Distance method used to calculate the distance down to the stream. One of several ways of measuring distance may be selected: the total straight line path (Pythagoras), the horizontal component of the straight line path (horizontal), the vertical component of the straight line path (vertical), or the total surface flow path (surface). Options: <ul style="list-style-type: none"> • 0 --- Pythagoras • 1 --- Horizontal • 2 --- Vertical • 3 --- Surface
Check for edge contamination		[boolean] Default: True	A flag that determines whether the tool should check for edge contamination. This is defined as the possibility that a value may be underestimated due to grid cells outside of the domain not being counted. In the context of Distance Down this occurs when part of a flow path traced downslope from a grid cell leaves the domain without reaching a stream grid cell. With edge contamination checking selected, the algorithm recognizes this and reports no data for the result. This is the
1254			Chapter 24 数据处理提供程序和算法

Outputs

Label	Name	Type	Description
D-Infinity Drop to Stream Grid		[raster]	Grid containing the distance to stream calculated using the D-infinity flow model and the statistical and path methods chosen.

Python code

Algorithm ID: taudem:dinfdistdown

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

D-Infinity Distance Up

This tool calculates the distance from each grid cell up to the ridge cells along the reverse D-infinity flow directions. Ridge cells are defined to be grid cells that have no contribution from grid cells further upslope. Given the convergence of multiple flow paths at any grid cell, any given grid cell can have multiple upslope ridge cells. There are three statistical methods that this tool can use: maximum distance, minimum distance and waited flow average over these flow paths. A variant on the above is to consider only grid cells that contribute flow with a proportion greater than a user specified threshold (t) to be considered as upslope of any given grid cell. Setting $t=0.5$ would result in only one flow path from any grid cell and would give the result equivalent to a D8 flow model, rather than D-infinity flow model, where flow is proportioned between two downslope grid cells. Finally there are several different optional paths that can be measured: the total straight line path (Pythagoras), the horizontal component of the straight line path, the vertical component of the straight line path, or the total surface flow path.

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-infinity method. Flow direction is measured in radians, counter clockwise from east. This can be created by the tool " D-Infinity Flow Directions ".
Pit Filled Elevation Grid		[raster]	This input is a grid of elevation values. As a general rule, it is recommended that you use a grid of elevation values that have had the pits removed for this input. Pits are generally taken to be artifacts that interfere with the analysis of flow across them. This grid can be obtained as the output of the " Pit Remove " tool, in which case it contains elevation values where the pits have been filled to the point where they just drain.
Slope Grid		[raster]	This input is a grid of slope values. This is measured as drop/distance and it is most often obtained as the output of the " D-Infinity Flow Directions " tool.
Statistical Method		[enumeration] Default: 2	Statistical method used to calculate the distance down to the stream. In the D-Infinity flow model, the outflow from each grid cell is proportioned between two downslope grid cells. Therefore, the distance from any grid cell to a stream is not uniquely defined. Flow that originates at a particular grid cell may enter the stream at a number of cells. The distance to the stream may be defined as the longest (maximum), shortest (minimum) or weighted average of the distance down to the stream. Options: <ul style="list-style-type: none"> • 0 --- Minimum • 1 --- Maximum • 2 --- Average
Distance Method		[enumeration] Default: 1	Distance method used to calculate the distance down to the stream. One of several ways of measuring distance may be selected: the total straight line path (Pythagoras), the horizontal component of the straight line path (horizontal), the vertical component of the straight line path (vertical), or the total surface flow path (surface). Options: <ul style="list-style-type: none"> • 0 --- Pythagoras • 1 --- Horizontal • 2 --- Vertical • 3 --- Surface
Proportion Threshold		[number] Default: 0.5	The proportion threshold parameter where only grid cells that contribute flow with a proportion greater than this user specified threshold (τ) is considered to be upslope of any given grid cell. Setting $\tau=0.5$ would result in only one flow path from any grid cell and would give the result equivalent to a D8 flow model, rather than D-Infinity flow model, where flow is proportioned between two downslope grid cells.
Check for edge contamination		[boolean] Default: True	A flag that determines whether the tool should check for edge contamination. This is defined as the possibility that a value may be underestimated due to grid cells outside of the domain not being counted.

Outputs

Label	Name	Type	Description
D-Infinity Distance Up		[raster]	Grid containing the distances up to the ridge calculated using the D-Infinity flow model and the statistical and path methods chosen.

Python code

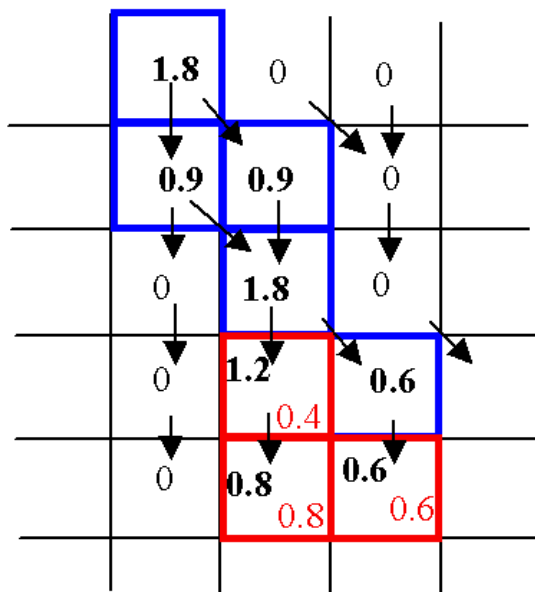
Algorithm ID: taudem:dinfdistup

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

D-Infinity Reverse Accumulation

This works in a similar way to evaluation of weighted Contributing area, except that the accumulation is by propagating the weight loadings upslope along the reverse of the flow directions to accumulate the quantity of weight loading downslope from each grid cell. The function also reports the maximum value of the weight loading downslope from each grid cell in the Maximum Downslope grid.



Reverse accumulation of field weights indicated in red



This function is designed to evaluate and map the hazard due to activities that may have an effect downslope. The example is land management activities that increase runoff. Runoff is sometimes a trigger for landslides or debris flows, so the weight grid here could be taken as a terrain stability map. Then the reverse accumulation provides a measure of the amount of unstable terrain downslope from each grid cell, as an indicator of the danger of activities that may increase runoff, even though there may be no potential for any local impact.

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-infinity method. Flow direction is measured in radians, counter clockwise from east. This can be created by the tool "D-Infinity Flow Directions" .
Weight Grid		[raster]	A grid giving weights (loadings) to be used in the accumulation.

Outputs

Label	Name	Type	Description
Reverse Accumulation Grid		[raster]	The grid giving the result of the "Reverse Accumulation" function. This works in a similar way to evaluation of weighted Contributing area, except that the accumulation is by propagating the weight loadings upslope along the reverse of the flow directions to accumulate the quantity of loading downslope from each grid cell.
Maximum Downslope Grid		[raster]	The grid giving the maximum of the weight loading grid downslope from each grid cell.

Python code

Algorithm ID: taudem:dinfrevaccum

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

D-Infinity Transport Limited Accumulation - 2

This function is designed to calculate the transport and deposition of a substance (e.g. sediment) that may be limited by both supply and the capacity of the flow field to transport it. This function accumulates substance flux (e.g. sediment transport) subject to the rule that transport out of any grid cell is the minimum between supply and transport capacity, T_{cap} . The total supply at a grid cell is calculated as the sum of the transport in from upslope grid cells, T_{in} , plus the local supply contribution, E (e.g. erosion). This function also outputs deposition, D , calculated as total supply minus actual transport.

$$T_{out} = \min(E + \sum T_{in}, T_{cap})$$

$$D = E + \sum T_{in} - T_{out}$$

Here E is the supply. T_{out} at each grid cell becomes T_{in} for downslope grid cells and is reported as Transport limited accumulation (t_{la}). D is deposition (t_{dep}). The function provides the option to evaluate concentration of a compound (contaminant) adhered to the transported substance. This is evaluated as follows:

$$L_{in} = \sum T_{in} C_{in}$$

Where L_{in} is the total incoming compound loading and C_{in} and T_{in} refer to the Concentration and Transport entering from each upslope grid cell.

$$T_{out} < \sum T_{in}$$

If

$$L_{out} = L_{in} \left(T_{out} / \sum T_{in} \right)$$

else

$$L_{out} = L_{in} + C_s \left(T_{out} - \sum T_{in} \right)$$

where C_s is the concentration supplied locally and the difference in the second term on the right represents the additional supply from the local grid cell. Then,

$$C_{out} = L_{out} / T_{out}$$

C_{out} at each grid cell comprises is the concentration grid output from this function.

If the outlets shapefile is used the tool only evaluates that part of the domain that contributes flow to the locations given by the shapefile.

Transport limited accumulation is useful for modeling erosion and sediment delivery, including the spatial dependence of sediment delivery ratio and contaminant that adheres to sediment.

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-infinity method. Flow direction is measured in radians, counter clockwise from east. This can be created by the tool "D-Infinity Flow Directions" .
Supply Grid		[raster]	A grid giving the supply (loading) of material to a transport limited accumulation function. In the application to erosion, this grid would give the erosion detachment, or sediment supplied at each grid cell.
Transport Capacity Grid		[raster]	A grid giving the transport capacity at each grid cell for the transport limited accumulation function. In the application to erosion this grid would give the transport capacity of the carrying flow.
Input Concentration Grid		[raster]	A grid giving the concentration of a compound of interest in the supply to the transport limited accumulation function. In the application to erosion, this grid would give the concentration of say phosphorous adhered to the eroded sediment.
Outlets Shapefile Optional		[vector: point]	This optional input is a point shapefile defining outlets of interest. If this file is used, the tool will only evaluate the area upslope of these outlets.
Check for edge contamination		[boolean] Default: True	This option determines whether the tool should check for edge contamination. Edge contamination is defined as the possibility that a value may be underestimated due to grid cells outside of the domain not being considered when determining the result.

Outputs

Label	Name	Type	Description
Transport Limited Accumulation Grid		[raster]	This grid is the weighted accumulation of supply accumulated respecting the limitations in transport capacity and reports the transport rate calculated by accumulating the substance flux subject to the rule that the transport out of any grid cell is the minimum of the total supply (local supply plus transport in) to that grid cell and the transport capacity.
Deposition Grid		[raster]	A grid giving the deposition resulting from the transport limited accumulation. This is the residual from the transport in to each grid cell minus the transport capacity out of the grid cell. The deposition grid is calculated as the transport in + the local supply - the transport out.
Output Concentration Grid		[raster]	If an input concentration in supply grid is given, then this grid is also output and gives the concentration of a compound (contaminant) adhered or bound to the transported substance (e.g. sediment) is calculated.

Python code

Algorithm ID: unknown

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

D-Infinity Transport Limited Accumulation

This function is designed to calculate the transport and deposition of a substance (e.g. sediment) that may be limited by both supply and the capacity of the flow field to transport it. This function accumulates substance flux (e.g. sediment transport) subject to the rule that transport out of any grid cell is the minimum between supply and transport capacity, T_{cap} . The total supply at a grid cell is calculated as the sum of the transport in from upslope grid cells, T_{in} , plus the local supply contribution, E (e.g. erosion). This function also outputs deposition, D , calculated as total supply minus actual transport.

$$T_{out} = \min(E + \sum T_{in}, T_{cap})$$

$$D = E + \sum T_{in} - T_{out}$$

Here E is the supply. T_{out} at each grid cell becomes T_{in} for downslope grid cells and is reported as Transport limited accumulation (t_{la}). D is deposition (t_{dep}). The function provides the option to evaluate concentration of a compound (contaminant) adhered to the transported substance. This is evaluated as follows:

$$L_{in} = \sum T_{in} C_{in}$$

Where L_{in} is the total incoming compound loading and C_{in} and T_{in} refer to the Concentration and Transport entering from each upslope grid cell.

$$T_{out} < \sum T_{in}$$

If

$$L_{out} = L_{in} \left(T_{out} / \sum T_{in} \right)$$

else

$$L_{out} = L_{in} + C_s \left(T_{out} - \sum T_{in} \right)$$

where C_s is the concentration supplied locally and the difference in the second term on the right represents the additional supply from the local grid cell. Then,

$$C_{out} = L_{out} / T_{out}$$

C_{out} at each grid cell comprises is the concentration grid output from this function.

If the outlets shapefile is used the tool only evaluates that part of the domain that contributes flow to the locations given by the shapefile.

Transport limited accumulation is useful for modeling erosion and sediment delivery, including the spatial dependence of sediment delivery ratio and contaminant that adheres to sediment.

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-infinity method. Flow direction is measured in radians, counter clockwise from east. This can be created by the tool " D-Infinity Flow Directions ".
Supply Grid		[raster]	A grid giving the supply (loading) of material to a transport limited accumulation function. In the application to erosion, this grid would give the erosion detachment, or sediment supplied at each grid cell.
Transport Capacity Grid		[raster]	A grid giving the transport capacity at each grid cell for the transport limited accumulation function. In the application to erosion this grid would give the transport capacity of the carrying flow.
Outlets Shapefile Optional		[vector: point]	This optional input is a point shapefile defining outlets of interest. If this file is used, the tool will only evaluate the area upslope of these outlets.
Check for edge contamination		[boolean] Default: True	This option determines whether the tool should check for edge contamination. Edge contamination is defined as the possibility that a value may be underestimated due to grid cells outside of the domain not being considered when determining the result.

Outputs

Label	Name	Type	Description
Transport Limited Accumulation Grid		[raster]	This grid is the weighted accumulation of supply accumulated respecting the limitations in transport capacity and reports the transport rate calculated by accumulating the substance flux subject to the rule that the transport out of any grid cell is the minimum of the total supply (local supply plus transport in) to that grid cell and the transport capacity.
Deposition Grid		[raster]	A grid giving the deposition resulting from the transport limited accumulation. This is the residual from the transport in to each grid cell minus the transport capacity out of the grid cell. The deposition grid is calculated as the transport in + the local supply - the transport out.

Python code

Algorithm ID: taudem:dinftranslimaccum

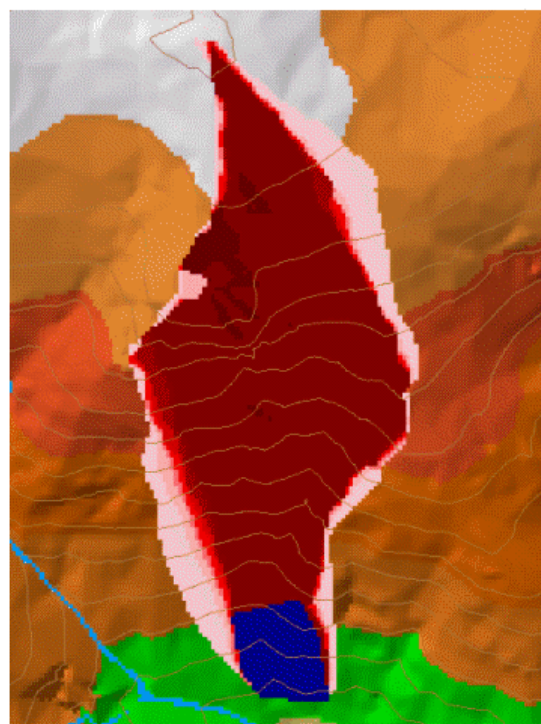
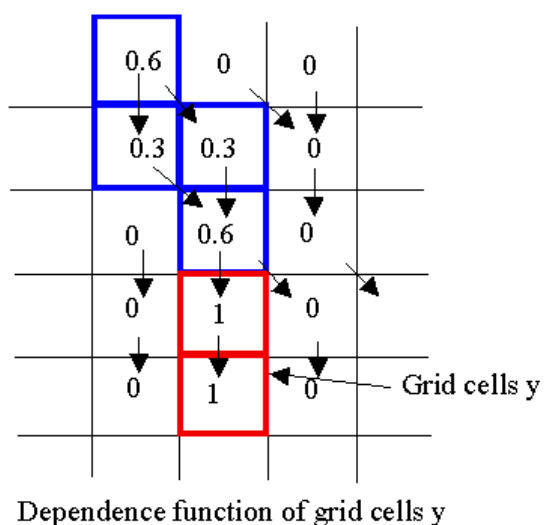
```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

D-Infinity Upslope Dependence

The D-Infinity Upslope Dependence tool quantifies the amount each grid cell in the domain contributes to a destination set of grid cells. D-Infinity flow directions proportion flow from each grid cell between multiple downslope grid cells. Following this flow field downslope the amount of flow originating at each grid cell that reaches the destination zone is defined. Upslope influence is evaluated using a downslope recursion, examining grid cells downslope from each grid cell, so that the map produced identifies the area upslope where flow through the destination zone originates, or the area it depends on, for its flow.

The figures below illustrate the amount each source point in the domain x (blue) contributes to the destination point or zone y (red). If the indicator weighted contributing area function is denoted $I(y; x)$ giving the weighted contribution using a unit value (1) from specific grid cells y to grid cells x , then the upslope dependence is: $D(x; y) = I(y; x)$.



This is useful for example to track where flow or a flow related substance or contaminant that enters a destination area may come from.

Parameters

Label	Name	Type	Description
D-Infinity Flow Direction Grid		[raster]	A grid giving flow direction by the D-Infinity method where the flow direction angle is determined as the direction of the steepest downward slope on the eight triangular facets formed in a 3x3 grid cell window centered on the grid cell of interest. This grid can be produced using the "D-Infinity Flow Direction" tool.
Destination Grid		[raster]	A grid that encodes the destination zone that may receive flow from upslope. This grid must be 1 inside the zone y and 0 over the rest of the domain.

Outputs

Label	Name	Type	Description
Output Upslope Dependence Grid		[raster]	A grid quantifying the amount each source point in the domain contributes to the zone defined by the destination grid.

Python code

Algorithm ID: taudem:dinfupdependence

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Slope Average Down

This tool computes slope in a D8 downslope direction averaged over a user selected distance. Distance should be specified in horizontal map units.

Parameters

Label	Name	Type	Description
D8 Flow Direction Grid		[raster]	This input is a grid of flow directions that are encoded using the D8 method where all flow from a cells goes to a single neighboring cell in the direction of steepest descent. This grid can be obtained as the output of the "D8 Flow Directions" tool.
Pit Filled Elevation Grid		[raster]	This input is a grid of elevation values. As a general rule, it is recommended that you use a grid of elevation values that have had the pits removed for this input. Pits are generally taken to be artifacts that interfere with the analysis of flow across them. This grid can be obtained as the output of the "Pit Remove" tool, in which case it contains elevation values where the pits have been filled to the point where they just drain.
Downslope Distance		[number] Default: 50	Input parameter of downslope distance over which to calculate the slope (in horizontal map units).

Outputs

Label	Name	Type	Description
Slope Average Down Grid		[raster]	This output is a grid of slopes calculated in the D8 downslope direction, averaged over the selected distance.

Python code

Algorithm ID: taudem:slopeavedown

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Slope Over Area Ratio

Calculates the ratio of the slope to the specific catchment area (contributing area). This is algebraically related to the more common $\ln(a/\tan \beta)$ wetness index, but contributing area is in the denominator to avoid divide by 0 errors when slope is 0.

Parameters

Label	Name	Type	Description
Slope Grid		[raster]	A grid of slope. This grid can be generated using either the "D8 Flow Directions" tool or the "D-Infinity Flow Directions" tool.
Specific Catchment Area Grid		[raster]	A grid giving the contributing area value for each cell taken as its own contribution plus the contribution from upslope neighbors that drain in to it. Contributing area is counted in terms of the number of grid cells (or summation of weights). This grid can be generated using either the "D8 Contributing Area" tool or the "D-Infinity Contributing Area" tool.

Outputs

Label	Name	Type	Description
Slope Divided By Area Ratio Grid		[raster]	A grid of the ratio of slope to specific catchment area (contributing area). This is algebraically related to the more common $\ln(a/\tan \beta)$ wetness index, but contributing area is in the denominator to avoid divide by 0 errors when slope is 0.

Python code

Algorithm ID: taudem:slopearearatio

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Topographic wetness index

Calculates the topographic wetness index (TWI).

Parameters

Label	Name	Type	Description
Slope		[raster]	A grid of slope. This grid can be generated using either the "D8 Flow Directions" tool or the "D-Infinity Flow Directions" tool.
Specific catchment area		[raster]	A grid giving the contributing area value for each cell taken as its own contribution plus the contribution from upslope neighbors that drain in to it. Contributing area is counted in terms of the number of grid cells (or summation of weights). This grid can be generated using either the "D8 Contributing Area" tool or the "D-Infinity Contributing Area" tool.

Outputs

Label	Name	Type	Description
Wetness index		[raster]	A grid of the wetness index (TWI).

Python code

Algorithm ID: taudem:twi

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.4.3 Stream Network Analysis

Connect down

For each zone in a raster entered (e.g. HUC converted to grid) it identifies the point with largest AreaD8. This is taken to be the outlet. A OGR file is created. Using flow directions each outlet is moved downflow a specified number of grid cells which is user controllable (Default is 1). The ID of the location the point has moved to is taken as iddown. Two OGR files are created one with the initial points and one with the moved points. Both contain id, iddown and AreaD8.

Parameters

Label	Name	Type	Description
D8 flow directions		[raster]	A grid of flow directions that are encoded using the D8 method where all flow from a cells goes to a single neighboring cell in the direction of steepest descent
D8 contribution area		[raster]	A grid giving the contributing area value in terms of the number of grid cells (or the summation of weights) for each cell taken as its own contribution plus the contribution from upslope neighbors that drain in to it using the D8 algorithm. This is usually the output of the " D8 Contributing Area " tool.
Watershed		[raster]	Watershed grid delineated from gage watershed function or streamreachwatershed function. Other watershed (e.g. HUC) raster also can be used as watershed grid.
Grid cells move to downstream		[number]	Number of grid cells move to downstream based on flow directions.

Outputs

Label	Name	Type	Description
Outlets		[vector: point]	A point OGR file where each point is created from watershed grid having the largest contributing area for each zone.
Moved Outlets		[vector: point]	A point OGR file defining moved outlets of interest. where each outlet is moved downflow a specified number of grid cells using flow directions.

Python code

Algorithm ID: taudem:connectdown

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

D8 Extreme Upslope Value

Evaluates the extreme (either maximum or minimum) upslope value from an input grid based on the D8 flow model. This is intended initially for use in stream raster generation to identify a threshold of the slope times area product that results in an optimum (according to drop analysis) stream network.

If the optional outlet point shapefile is used, only the outlet cells and the cells upslope (by the D8 flow model) of them are in the domain to be evaluated.

By default, the tool checks for edge contamination. This is defined as the possibility that a result may be underestimated due to grid cells outside of the domain not being counted. This occurs when drainage is inwards from the boundaries or areas with "no data" values for elevation. The algorithm recognizes this and reports "no data" for the result for these grid cells. It is common to see streaks of "no data" values extending inwards from boundaries along flow paths that enter the domain at a boundary. This is the desired effect and indicates that the result for these grid cells is unknown due to it being dependent on terrain outside of the domain of data available. Edge contamination checking may be turned off in cases where you know this is not an issue or want to ignore these problems, if for example, the DEM has been clipped along a watershed outline.

Parameters

Label	Name	Type	Description
D8 Flow Directions Grid		[raster]	A grid of D8 flow directions which are defined, for each cell, as the direction of the one of its eight adjacent or diagonal neighbors with the steepest downward slope. This grid can be obtained as the output of the " D8 Flow Directions " tool.
Upslope Values Grid		[raster]	This is the grid of values of which the maximum or minimum upslope value is selected. The values most commonly used are the slope times area product needed when generating stream rasters according to drop analysis.
Outlets Shapefile Optional		[vector: point]	A point shape file defining outlets of interest. If this input file is used, only the area upslope of these outlets will be evaluated by the tool.
Check for edge contamination		[boolean] Default: True	A flag that indicates whether the tool should check for edge contamination.
Use max upslope value		[boolean] Default: True	A flag to indicate whether the maximum or minimum upslope value is to be calculated.

Outputs

Label	Name	Type	Description
Extreme Upslope Values Grid		[raster]	A grid of the maximum/minimum upslope values.

Python code

Algorithm ID: taudem:d8flowpathextremeup

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Gage Watershed

Calculates Gage Watersheds Grid. Each grid cell is labeled with the identifier (from column *id*) of the gage to which it drains directly without passing through any other gages.

Parameters

Label	Name	Type	Description
A suppr D-infinity flow directions	DINF_FLOWDIR	[raster]	A grid of flow directions based on the D-infinity flow method
D8 Flow Directions Grid		[raster]	A grid of D8 flow directions which are defined, for each cell, as the direction of the one of its eight adjacent or diagonal neighbors with the steepest downward slope. This grid can be obtained as the output of the "D8 Flow Directions" tool.
Gages Shapefile		[vector: point]	A point shapefile defining the gages to which watersheds will be delineated. This shapefile should have a column <i>id</i> . Grid cells draining directly to each point in this shapefile will be labeled with this <i>id</i> .

Outputs

Label	Name	Type	Description
Gage Watershed Grid		[raster]	A grid identifies each gage watershed. Each grid cell is labeled with the identifier (from column <i>id</i>) of the gage to which it drains directly without passing through any other gages.
Downstream Identifiers File		[file]	Text file giving watershed downslope connectivity

Python code

Algorithm ID: taudem:gagewatershed

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Length Area Stream Source

Creates an indicator grid (1, 0) that evaluates $A \geq (M) (L^y)$ based on upslope path length, D8 contributing area grid inputs, and parameters M and y. This grid indicates likely stream source grid cells. This is an experimental method with theoretical basis in Hack's law which states that for streams $L \sim A^{0.6}$. However for hillslopes with parallel flow $L \sim A$. So a transition from hillslopes to streams may be represented by $L \sim A^{0.8}$ suggesting identifying grid cells as stream cells if $A > M (L^{1/0.8})$.

Parameters

Label	Name	Type	Description
Length Grid		[raster]	A grid of the maximum upslope length for each cell. This is calculated as the length of the flow path from the furthest cell that drains to each cell. Length is measured between cell centers taking into account cell size and whether the direction is adjacent or diagonal. It is this length (L) that is used in the formula, $A > (M) (L^y)$, to determine which cells are considered stream cells. This grid can be obtained as an output from the "Grid Network" tool.
Contributing Area Grid		[raster]	A grid of contributing area values for each cell that were calculated using the D8 algorithm. The contributing area for a cell is the sum of its own contribution plus the contribution from all upslope neighbors that drain to it, measured as a number of cells. This grid is typically obtained as the output of the "D8 Contributing Area" tool. In this tool, it is the contributing area (A) that is compared in the formula $A > (M) (L^y)$ to determine the transition to a stream.
Threshold		[number] Default: 0.03	The multiplier threshold (M) parameter which is used in the formula: $A > (M) (L^y)$, to identify the beginning of streams.
Exponent		[number] Default: 1.3	The exponent (y) parameter which is used in the formula: $A > (M) (L^y)$, to identify the beginning of streams. In branching systems, Hack's law suggests that $L = 1/M A^{(1/y)}$ with $1/y = 0.6$ (or 0.56) (y about 1.7). In parallel flow systems L is proportional to A (y about 1). This method tries to identify the transition between these two paradigms by using an exponent y somewhere in between (y about 1.3).

Outputs

Label	Name	Type	Description
Stream Source Grid		[raster]	An indicator grid (1,0) that evaluates $A \geq (M)(L^y)$, based on the maximum upslope path length, the D8 contributing area grid inputs, and parameters M and y. This grid indicates likely stream source grid cells.

Python code

Algorithm ID: taudem:lengtharea

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Move Outlets To Streams

Moves outlet points that are not aligned with a stream cell from a stream raster grid, downslope along the D8 flow direction until a stream raster cell is encountered, the "max_dist" number of grid cells are examined, or the flow path exits the domain (i.e. a "no data" value is encountered for the D8 flow direction). The output file is a new outlets shapefile where each point has been moved to coincide with the stream raster grid, if possible. A field "dist_moved" is added to the new outlets shapefile to indicate the changes made to each point. Points that are already on a stream cell are not moved and their "dist_moved" field is assigned a value 0. Points that are initially not on a stream cell are moved by sliding them downslope along the D8 flow direction until one of the following occurs: a) A stream raster grid cell is encountered before traversing the "max_dist" number of grid cells. In which case, the point is moved and the "dist_moved" field is assigned a value indicating how many grid cells the point was moved. b) More than the "max_number" of grid cells are traversed, or c) the traversal ends up going out of the domain (i.e., a "no data" D8 flow direction value is encountered). In which case, the point is not moved and the "dist_moved" field is assigned a value of -1.

Parameters

Label	Name	Type	Description
D8 Flow Direction Grid		[raster]	A grid of D8 flow directions which are defined, for each cell, as the direction of the one of its eight adjacent or diagonal neighbors with the steepest downward slope. This grid can be obtained as the output of the " D8 Flow Directions " tool.
Stream Raster Grid		[raster]	This output is an indicator grid (1, 0) that indicates the location of streams, with a value of 1 for each of the stream cells and 0 for the remainder of the cells. This file is produced by several different tools in the " Stream Network Analysis " toolset.
Outlets Shapefile		[vector: point]	A point shape file defining points of interest or outlets that should ideally be located on a stream, but may not be exactly on the stream due to the fact that the shapefile point locations may not have been accurately registered with respect to the stream raster grid.
Maximum Number of Grid Cells to traverse		[number] Default: 50	This input parameter is the maximum number of grid cells that the points in the input outlet shapefile will be moved before they are saved to the output outlet shapefile.

Outputs

Label	Name	Type	Description
Output Outlet Shapefile		[vector: point]	A point shape file defining points of interest or outlets. This file has one point in it for each point in the input outlet shapefile. If the original point was located on a stream, then the point was not moved. If the original point was not on a stream, the point was moved downslope according to the D8 flow direction until it reached a stream or the maximum distance had been reached. This file has an additional field "dist_moved" added to it which is the number of cells that the point was moved. This field is 0 if the cell was originally on a stream, -1 if it was not moved because there was not a stream within the maximum distance, or some positive value if it was moved.

Python code

Algorithm ID: taudem:moveoutletstostreams

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

道格拉斯-普克算法

Creates an indicator grid (1, 0) of upward curved grid cells according to the Peuker and Douglas algorithm.

With this tool, the DEM is first smoothed by a kernel with weights at the center, sides, and diagonals. The Peuker and Douglas (1975) method (also explained in Band, 1986), is then used to identify upwardly curving grid cells. This technique flags the entire grid, then examines in a single pass each quadrant of 4 grid cells, and unflags the highest. The remaining flagged cells are deemed "upwardly curved", and when viewed, resemble a channel network. This proto-channel network generally lacks connectivity and requires thinning, issues that were discussed in detail by Band (1986).

Parameters

Label	Name	Type	Description
Elevation Grid		[raster]	A grid of elevation values. This is usually the output of the " Pit Remove " tool, in which case it is elevations with pits removed.
Center Smoothing Weight		[number] Default: 0.4	The center weight parameter used by a kernel to smooth the DEM before the tool identifies upwardly curved grid cells.
Side Smoothing Weight		[number] Default: 0.1	The side weight parameter used by a kernel to smooth the DEM before the tool identifies upwardly curved grid cells.

下页继续

表 24.234 – 续上页

Diagonal Smoothing Weight		[number] Default: 0.05	The diagonal weight parameter used by a kernel to smooth the DEM before the tool identifies upwardly curved grid cells.
----------------------------------	--	---------------------------	---

Outputs

Label	Name	Type	Description
Stream Source Grid		[raster]	An indicator grid (1, 0) of upward curved grid cells according to the Peuker and Douglas algorithm, and if viewed, resembles a channel network. This proto-channel network generally lacks connectivity and requires thinning, issues that were discussed in detail by Band (1986).

Python code

Algorithm ID: taudem:peukerdouglas

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

See also

- Band, L. E., (1986), "Topographic partition of watersheds with digital elevation models", Water Resources Research, 22(1): 15-24.
- Peuker, T. K. and D. H. Douglas, (1975), "Detection of surface-specific points by local parallel processing of discrete terrain elevation data", Comput. Graphics Image Process., 4: 375-387.

Peuker Douglas stream

Combines the functionality of the "Peuker Douglas", "D8 Contributing Area", "Stream Drop Analysis" and "Stream Definition by Threshold" tools in order to generate a stream indicator grid (1,0) where the streams are located using a DEM curvature-based method. With this method, the DEM is first smoothed by a kernel with weights at the center, sides, and diagonals. The Peuker and Douglas (1975) method (also explained in Band, 1986), is then used to identify upwardly curving grid cells. This technique flags the entire grid, then examines in a single pass each quadrant of 4 grid cells, and unflags the highest. The remaining flagged cells are deemed 'upwardly curved', and when viewed, resemble a channel network. This proto-channel network sometimes lacks connectivity, and/or requires thinning, issues that were discussed in detail by Band (1986). The thinning and connecting of these grid cells is achieved here by computing the D8 contributing area using only these upwardly curving cells. An accumulation threshold on the number of these cells is then used to map the channel network where this threshold is optionally set by the user, or determined via drop analysis.

If drop analysis is used, then instead of providing a value for the accumulation threshold, the accumulation threshold value is determined by searching the range between the Drop Analysis Parameters "Lowest" and "Highest", using the number of steps in the parameter "Number". For the science behind drop analysis, see Tarboton, et al. (1991, 1992), and Tarboton and Ames (2001). The value of accumulation threshold that is selected is the smallest value where the absolute value of the t-statistic is less than 2. This is written to the drop analysis table text file. Drop analysis is only possible when outlets have been specified, because if an entire grid domain is analyzed, as the threshold varies, shorter

streams draining off the edge may not meet the threshold criterion and be excluded from the analysis. This makes defining drainage density problematic and it is somewhat inconsistent to compare statistics evaluated over differing domains.

Parameters

Outputs

Label	Name	Type	Description
Stream source		[raster]	An indicator grid (1, 0) of upward curved grid cells according to the Peuker and Douglas algorithm, and if viewed, resembles a channel network. This proto-channel network generally lacks connectivity and requires thinning, issues that were discussed in detail by Band (1986).

Python code

Algorithm ID: taudem:peukerdouglasstreamdef

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Slope Area Combination

Creates a grid of slope-area values = $(S_m) (A_n)$ based on slope and specific catchment area grid inputs, and parameters *m* and *n*. This tool is intended for use as part of the slope-area stream raster delineation method.

Parameters

Label	Name	Type	Description
Slope Grid		[raster]	This input is a grid of slope values. This grid can be obtained from the "D-Infinity Flow Directions" tool.
Contributing Area Grid		[raster]	A grid giving the specific catchment area for each cell taken as its own contribution (grid cell length or summation of weights) plus the proportional contribution from upslope neighbors that drain in to it. This grid is typically obtained from the "D-Infinity Contributing Area" tool.
Slope Exponent		[number] Default: 2	The slope exponent (<i>m</i>) parameter which will be used in the formula: $(S_m) (A_n)$, that is used to create the slope-area grid.
Area Exponent		[number] Default: 1	The area exponent (<i>n</i>) parameter which will be used in the formula: $(S_m) (A_n)$, that is used to create the slope-area grid.

Outputs

Label	Name	Type	Description
Slope Area Grid		[raster]	A grid of slope-area values = $(S_m) (A_n)$ calculated from the slope grid, specific catchment area grid, m slope exponent parameter, and n area exponent parameter.

Python code

Algorithm ID: taudem:slopearea

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Slope area stream definition

Creates a grid of slope-area values = $(S_m) (A_n)$ based on slope and specific catchment area grid inputs, and parameters m and n . This tool is intended for use as part of the slope-area stream raster delineation method.

Parameters

Label	Name	Type	Description
D8 flow directions		[raster]	
D-infinity Contributing Area		[raster]	A grid giving the specific catchment area for each cell taken as its own contribution (grid cell length or summation of weights) plus the proportional contribution from upslope neighbors that drain in to it. This grid is typically obtained from the "D-Infinity Contributing Area" tool.
Slope		[raster]	This input is a grid of slope values. This grid can be obtained from the "D-Infinity Flow Directions" tool.
Mask grid		[raster]	
Outlets		[vector: point]	
Pit-filled grid for drop analysis		[raster]	
D8 contributing area for drop analysis		[raster]	
Slope Exponent		[number] Default: 2	The slope exponent (m) parameter which will be used in the formula: $(S_m) (A_n)$, that is used to create the slope-area grid.
Area Exponent		[number] Default: 1	The area exponent (n) parameter which will be used in the formula: $(S_m) (A_n)$, that is used to create the slope-area grid.
Accumulation threshold		[number]	

下页继续

表 24.239 - 续上页

Minimum threshold		[number]	
Maximum threshold		[number]	
Number of drop thresholds		[number]	
Type of threshold step		[enumeration] Default: 0	Options: <ul style="list-style-type: none"> • 0 --- Logarithmic • 1 --- Linear
Check for edge contamination		[boolean]	
Select threshold by drop analysis		[boolean]	

Outputs

Label	Name	Type	Description
Stream raster		[raster]	
Slope area		[raster]	A grid of slope-area values = $(S_m)(A_n)$ calculated from the slope grid, specific catchment area grid, m slope exponent parameter, and n area exponent parameter.
Maximum upslope		[raster]	
Drop analysis		[file]	

Python code

Algorithm ID: `taudem:slopeareastreamdef`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Stream Definition By Threshold

Operates on any grid and outputs an indicator (1, 0) grid identifying cells with input values \geq the threshold value. The standard use is to use an accumulated source area grid to as the input grid to generate a stream raster grid as the output. If you use the optional input mask grid, it limits the domain being evaluated to cells with mask values \geq 0. When you use a D-infinity contributing area grid (**sca*) as the mask grid, it functions as an edge contamination mask. The threshold logic is:

```
src = ((ssa >= thresh) & (mask >= s0)) ? 1:0
```

Parameters

Label	Name	Type	Description
Accumulated Stream Source Grid		[raster]	This grid nominally accumulates some characteristic or combination of characteristics of the watershed. The exact characteristic(s) varies depending on the stream network raster algorithm being used. This grid needs to have the property that grid cell values are monotonically increasing downslope along D8 flow directions, so that the resulting stream network is continuous. While this grid is often from an accumulation, other sources such as a maximum upslope function will also produce a suitable grid.
Threshold		[number] Default: 100	This parameter is compared to the value in the Accumulated Stream Source grid (<i>*ssa</i>) to determine if the cell should be considered a stream cell. Streams are identified as grid cells for which ssa value is \geq this threshold.
Mask Grid Optional		[raster]	This optional input is a grid that is used to mask the domain of interest and output is only provided where this grid is ≥ 0 . A common use of this input is to use a D-Infinity contributing area grid as the mask so that the delineated stream network is constrained to areas where D-infinity contributing area is available, replicating the functionality of an edge contamination mask.

Outputs

Label	Name	Type	Description
Stream Raster Grid		[raster]	This is an indicator grid (1, 0) that indicates the location of streams, with a value of 1 for each of the stream cells and 0 for the remainder of the cells.

Python code

Algorithm ID: taudem:threshold

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

Stream definition with drop analysis

Combines the function of the "Stream Drop Analysis" tool and the "Stream Definition by Threshold" tools. It applies a series of thresholds (determined from the input parameters) to the input accumulated stream source grid (*ssa*) grid and outputs the results in the stream drop statistics table (*drp.txt*). Then it outputs a stream raster grid, which is an indicator (1,0) grid of stream cells. Stream cells are defined as those cells where the accumulated stream source value is \geq the optimal threshold as determined from the stream drop statistics. There is an option to include a mask input to replicate the functionality for using the **sca* file as an edge contamination mask. The threshold logic should be: $\text{src} = ((\text{ssa} \geq \text{thresh}) \ \& \ (\text{mask} \geq 0)) \ ? \ 1:0$

Parameters

Outputs

Python code

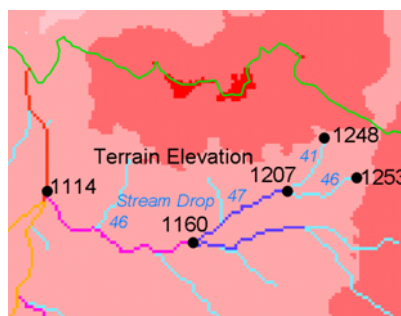
Algorithm ID: `taudem:streamdefdropanalysis`

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See [Using processing algorithms from the console](#) for details on how to run processing algorithms from the Python console.

Stream Drop Analysis

Applies a series of thresholds (determined from the input parameters) to the input accumulated stream source grid (**ssa*) grid and outputs the results in the **drp.txt* file the stream drop statistics table. This function is designed to aid in the determination of a geomorphologically objective threshold to be used to delineate streams. Drop Analysis attempts to select the right threshold automatically by evaluating a stream network for a range of thresholds and examining the constant drop property of the resulting Strahler streams. Basically it asks the question: Is the mean stream drop for first order streams statistically different from the mean stream drop for higher order streams, using a T-test. Stream drop is the difference in elevation from the beginning to the end of a stream defined as the sequence of links of the same stream order. If the T-test shows a significant difference then the stream network does not obey this "law" so a larger threshold needs to be chosen. The smallest threshold for which the T-test does not show a significant difference gives the highest resolution stream network that obeys the constant stream drop "law" from geomorphology, and is the threshold chosen for the "objective" or automatic mapping of streams from the DEM. This function can be used in the development of stream network rasters, where the exact watershed characteristic(s) that were accumulated in the accumulated stream source grid vary based on the method being used to determine the stream network raster.



The constant stream drop "law" was identified by Broscoe (1959). For the science behind using this to determine a stream delineation threshold, see Tarboton et al. (1991, 1992), Tarboton and Ames (2001).

Parameters

Label	Name	Type	Description
D8 Contributing Area Grid		[raster]	A grid of contributing area values for each cell that were calculated using the D8 algorithm. The contributing area for a cell is the sum of its own contribution plus the contribution from all upslope neighbors that drain to it, measured as a number of cells or the sum of weight loadings. This grid can be obtained as the output of the "D8 Contributing Area" tool. This grid is used in the evaluation of drainage density reported in the stream drop table.
D8 Flow Direction Grid		[raster]	A grid of D8 flow directions which are defined, for each cell, as the direction of the one of its eight adjacent or diagonal neighbors with the steepest downward slope. This grid can be obtained as the output of the "D8 Flow Directions" tool.
Pit Filled Elevation Grid		[raster]	A grid of elevation values. This is usually the output of the "Pit Remove" tool, in which case it is elevations with pits removed.
Accumulated Stream Source Grid		[raster]	This grid must be monotonically increasing along the downslope D8 flow directions. It is compared to a series of thresholds to determine the beginning of the streams. It is often generated by accumulating some characteristic or combination of characteristics of the watershed with the "D8 Contributing Area" tool, or using the maximum option of the "D8 Flow Path Extreme" tool. The exact method varies depending on the algorithm being used.
Outlets Shapefile		[vector: point]	A point shapefile defining the outlets upstream of which drop analysis is performed.
Minimum Threshold		[number] Default: 5	This parameter is the lowest end of the range searched for possible threshold values using drop analysis. This technique looks for the smallest threshold in the range where the absolute value of the t-statistic is less than 2. For the science behind the drop analysis see Tarboton et al. (1991, 1992), Tarboton and Ames (2001).
Maximum Threshold		[number] Default: 500	This parameter is the highest end of the range searched for possible threshold values using drop analysis. This technique looks for the smallest threshold in the range where the absolute value of the t-statistic is less than 2. For the science behind the drop analysis see Tarboton et al. (1991, 1992), Tarboton and Ames (2001).
Number of Threshold Values		[number] Default: 10	The parameter is the number of steps to divide the search range into when looking for possible threshold values using drop analysis. This technique looks for the smallest threshold in the range where the absolute value of the t-statistic is less than 2. For the science behind the drop analysis see Tarboton et al. (1991, 1992), Tarboton and Ames (2001).

下页继续

表 24.243 – 续上页

Spacing for Threshold Values		[enumeration] Default: 0	This parameter indicates whether logarithmic or linear spacing should be used when looking for possible threshold values using drop analysis. Options: <ul style="list-style-type: none"> • 0 --- Logarithmic • 1 --- Linear
-------------------------------------	--	-----------------------------	---

Outputs

Label	Name	Type	Description
D-Infinity Drop to Stream Grid		[file]	This is a comma delimited text file with the following header line: Threshold,DrainDen,NoFirstOrd, ↔NoHighOrd,MeanDFirstOrd,MeanDHighOrd, ↔StdDevFirstOrd,StdDevHighOrd,T The file then contains one line of data for each threshold value examined, and then a summary line that indicates the optimum threshold value. This technique looks for the smallest threshold in the range where the absolute value of the t-statistic is less than 2. For the science behind the drop analysis, see Tarboton et al. (1991, 1992), Tarboton and Ames (2001).

Python code

Algorithm ID: taudem:dropanalysis

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMEs and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

See also

- Broscocoe, A. J., (1959), "Quantitative analysis of longitudinal stream profiles of small watersheds", Office of Naval Research, Project NR 389-042, Technical Report No. 18, Department of Geology, Columbia University, New York.
- Tarboton, D. G., R. L. Bras and I. Rodriguez-Iturbe, (1991), "On the Extraction of Channel Networks from Digital Elevation Data", *Hydrologic Processes*, 5(1): 81-100.
- Tarboton, D. G., R. L. Bras and I. Rodriguez-Iturbe, (1992), "A Physical Basis for Drainage Density", *Geomorphology*, 5(1/2): 59-76.
- Tarboton, D. G. and D. P. Ames, (2001), "Advances in the mapping of flow networks from digital elevation data", World Water and Environmental Resources Congress, Orlando, Florida, May 20-24, ASCE, https://www.researchgate.net/publication/2329568_Advances_in_the_Mapping_of_Flow_Networks_From_Digital_Elevation_Data.

Stream Reach and Watershed

This tool produces a vector network and shapefile from the stream raster grid. The flow direction grid is used to connect flow paths along the stream raster. The Strahler order of each stream segment is computed. The subwatershed draining to each stream segment (reach) is also delineated and labeled with the value identifier that corresponds to the WSNO (watershed number) attribute in the Stream Reach Shapefile.

This tool orders the stream network according to the Strahler ordering system. Streams that don't have any other streams draining in to them are order 1. When two stream reaches of different order join the order of the downstream reach is the order of the highest incoming reach. When two reaches of equal order join the downstream reach order is increased by 1. When more than two reaches join the downstream reach order is calculated as the maximum of the highest incoming reach order or the second highest incoming reach order + 1. This generalizes the common definition to cases where more than two reaches join at a point. The network topological connectivity is stored in the Stream Network Tree file, and coordinates and attributes from each grid cell along the network are stored in the Network Coordinates file.

The stream raster grid is used as the source for the stream network, and the flow direction grid is used to trace connections within the stream network. Elevations and contributing area are used to determine the elevation and contributing area attributes in the network coordinate file. Points in the outlets shapefile are used to logically split stream reaches to facilitate representing watersheds upstream and downstream of monitoring points. The program uses the attribute field "id" in the outlets shapefile as identifiers in the Network Tree file. This tool then translates the text file vector network representation in the Network Tree and Coordinates files into a shapefile. Further attributes are also evaluated. The program has an option to delineate a single watershed by representing the entire area draining to the Stream Network as a single value in the output watershed grid.

Parameters

Label	Name	Type	Description
Pit Filled Elevation Grid		[raster]	A grid of elevation values. This is usually the output of the " Pit Remove " tool, in which case it is elevations with pits removed.
D8 Flow Direction Grid		[raster]	A grid of D8 flow directions which are defined, for each cell, as the direction of the one of its eight adjacent or diagonal neighbors with the steepest downward slope. This grid can be obtained as the output of the " D8 Flow Directions " tool.
D8 Drainage Area		[raster]	A grid giving the contributing area value in terms of the number of grid cells (or the summation of weights) for each cell taken as its own contribution plus the contribution from upslope neighbors that drain in to it using the D8 algorithm. This is usually the output of the " D8 Contributing Area " tool and is used to determine the contributing area attribute in the Network Coordinate file.
Stream Raster Grid		[raster]	An indicator grid indicating streams, by using a grid cell value of 1 on streams and 0 off streams. Several of the " Stream Network Analysis " tools produce this type of grid. The Stream Raster Grid is used as the source for the stream network.

下页继续

表 24.245 - 续上页

Outlets Shapefile as Network Nodes Optional		[vector: point]	A point shape file defining points of interest. If this file is used, the tool will only delineate the stream network upstream of these outlets. Additionally, points in the Outlets Shapefile are used to logically split stream reaches to facilitate representing watersheds upstream and downstream of monitoring points. This tool REQUIRES THAT THERE BE an integer attribute field "id" in the Outlets Shapefile, because the "id" values are used as identifiers in the Network Tree file.
Delineate Single Watershed		[boolean] Default: True	This option causes the tool to delineate a single watershed by representing the entire area draining to the Stream Network as a single value in the output watershed grid. Otherwise a separate watershed is delineated for each stream reach. Default is <i>False</i> (separate watershed).

Outputs

Label	Name	Type	Description
Stream Order Grid		[raster]	The Stream Order Grid has cells values of streams ordered according to the Strahler order system. The Strahler ordering system defines order 1 streams as stream reaches that don't have any other reaches draining in to them. When two stream reaches of different order join the order of the downstream reach is the order of the highest incoming reach. When two reaches of equal order join the downstream reach order is increased by 1. When more than two reaches join the downstream reach order is calculated as the maximum of the highest incoming reach order or the second highest incoming reach order + 1. This generalizes the common definition to cases where more than two flow paths reaches join at a point.
Watershed Grid		[raster]	This output grid identified each reach watershed with a unique ID number, or in the case where the delineate single watershed option was checked, the entire area draining to the stream network is identified with a single ID.

下页继续

表 24.246 – 续上页

<p>Stream Shapefile</p>	<p>Reach</p>	<p>[vector: line]</p>	<p>This output is a polyline shapefile giving the links in a stream network. The columns in the attribute table are:</p> <ul style="list-style-type: none"> • LINKNO --- Link Number. A unique number associated with each link (segment of channel between junctions). This is arbitrary and will vary depending on number of processes used • DSLINKNO --- Link Number of the downstream link. -1 indicates that this does not exist • USLINKNO1 --- Link Number of first upstream link. (-1 indicates no link upstream, i.e. for a source link) • USLINKNO2 --- Link Number of second upstream link. (-1 indicates no second link upstream, i.e. for a source link or an internal monitoring point where the reach is logically split but the network does not bifurcate) • DSNODEID --- Node identifier for node at downstream end of stream reach. This identifier corresponds to the "id" attribute from the Outlets shapefile used to designate nodes • Order --- Strahler Stream Order • Length --- Length of the link. The units are the horizontal map units of the underlying DEM grid • Magnitude --- Shreve Magnitude of the link. This is the total number of sources upstream • DS_Cont_Ar --- Drainage area at the downstream end of the link. Generally this is one grid cell upstream of the downstream end because the drainage area at the downstream end grid cell includes the area of the stream being joined • Drop --- Drop in elevation from the start to the end of the link • Slope --- Average slope of the link (computed as drop/length) • Straight_L --- Straight line distance from the start to the end of the link • US_Cont_Ar --- Drainage area at the upstream end of the link • WSNO --- Watershed number. Cross reference to the *w.shp and *w grid files giving the identification number of the watershed draining directly to the link • DOUT_END --- Distance to the eventual outlet (i.e. the most downstream point in the stream network) from the downstream end of the link • DOUT_START --- Distance to the eventual outlet from the upstream end of the link • DOUT_MID --- Distance to the eventual outlet from the midpoint of the link
--------------------------------	---------------------	-----------------------	--

下页继续

表 24.246 – 续上页

<p>Network Connectivity Tree</p>		<p>[file]</p>	<p>This output is a text file that details the network topological connectivity is stored in the Stream Network Tree file. Columns are as follows:</p> <ul style="list-style-type: none"> • Link Number (Arbitrary --- will vary depending on number of processes used) • Start Point Number in Network coordinates (*coord.dat) file (Indexed from 0) • End Point Number in Network coordinates (*coord.dat) file (Indexed from 0) • Next (Downstream) Link Number. Points to Link Number. -1 indicates no links downstream, i.e. a terminal link • First Previous (Upstream) Link Number. Points to Link Number. -1 indicates no upstream links • Second Previous (Upstream) Link Numbers. Points to Link Number. -1 indicates no upstream links. Where only one previous link is -1, it indicates an internal monitoring point where the reach is logically split, but the network does not bifurcate • Strahler Order of Link • Monitoring point identifier at downstream end of link. -1 indicates downstream end is not a monitoring point • Network magnitude of the link, calculated as the number of upstream sources (following Shreve)
<p>Network Coordinates</p>		<p>[file]</p>	<p>This output is a text file that contains the coordinates and attributes of points along the stream network. Columns are as follows:</p> <ul style="list-style-type: none"> • 横坐标 • Y Coordinate • Distance along channels to the downstream end of a terminal link • Elevation • Contributing area

Python code

Algorithm ID: taudem:streamnet

```
import processing
processing.run("algorithm_id", {parameter_dictionary})
```

The *algorithm id* is displayed when you hover over the algorithm in the Processing Toolbox. The *parameter dictionary* provides the parameter NAMES and values. See *Using processing algorithms from the console* for details on how to run processing algorithms from the Python console.

24.5 OTB 应用程序提供者

OTB (Orfeo ToolBox) 是用于遥感数据的图像处理库，它还提供图像处理功能的应用程序。应用程序列表及其文档可在 [OTB CookBook](#) 找到。

25.1 QGIS 插件


QGIS has been designed with a plugin architecture. This allows many new features and functions to be easily added to the application. Some of the features in QGIS are actually implemented as plugins.

25.1.1 Core and External plugins

QGIS plugins are implemented either as **Core Plugins** or **External Plugins**.

Core Plugins are maintained by the QGIS Development Team and are automatically part of every QGIS distribution. They are written in one of two languages: **C++** or **Python**.

Most of External Plugins are currently written in Python. They are stored either in the 'Official' QGIS Repository at <https://plugins.qgis.org/plugins/> or in external repositories and are maintained by the individual authors. Detailed documentation about the usage, minimum QGIS version, home page, authors, and other important information are provided for the plugins in the Official repository. For other external repositories, documentation might be available with the external plugins themselves. External plugins documentation is not included in this manual.

To install or activate a plugin, go to *Plugins* menu and select  *Manage and install plugins...* Installed external python plugins are placed under the `python/plugins` folder of the active *user profile* path.


Paths to Custom C++ plugins libraries can also be added under *Settings* ► *Options* ► *System*.

注解: According to the *plugin manager settings*, QGIS main interface can display an icon on the right of the status bar to inform you that there are updates for your installed plugins or new plugins available.


25.1.2 The Plugins Dialog

The tabs in the Plugins dialog allow the user to install, uninstall and upgrade plugins in different ways. Each plugin has some metadata displayed in the right panel:

- information on whether the plugin is experimental
- description
- rating vote(s) (you can vote for your preferred plugin!)
- tags
- some useful links to the home page, tracker and code repository
- author(s)
- version available

At the top of the dialog, a *Search* function helps you find any plugin using metadata information (author, name, description...). It is available in nearly every tab (except  *Settings*).

The Settings tab

The  *Settings* tab is the main place you can configure which plugins can be displayed in your application. You can use the following options:

- *Check for updates on startup*. Whenever a new plugin or a plugin update is available, QGIS will inform you 'every time QGIS starts', 'once a day', 'every 3 days', 'every week', 'every 2 weeks' or 'every month'.
- *Show also experimental plugins*. QGIS will show you plugins in early stages of development, which are generally unsuitable for production use.
- *Show also deprecated plugins*. Because they use functions that are no longer available in QGIS, these plugins are set deprecated and generally unsuitable for production use. They appear among invalid plugins list.

By default, QGIS provides you with its official plugin repository with the URL <https://plugins.qgis.org/plugins/plugins.xml?qgis=3.0> (in case of QGIS 3.0) in the *Plugin repositories* section. To add external author repositories, click *Add...* and fill in the *Repository Details* form with a name and the URL. The URL can be of `http://` or `file://` protocol type.

The default QGIS repository is an open repository and you don't need any authentication to access it. You can however deploy your own plugin repository and require an authentication (basic authentication, PKI). You can get more information on QGIS authentication support in *Authentication* chapter.

If you do not want one or more of the added repositories, they can be disabled from the Settings tab via the *Edit...* button, or completely removed with the *Delete* button.

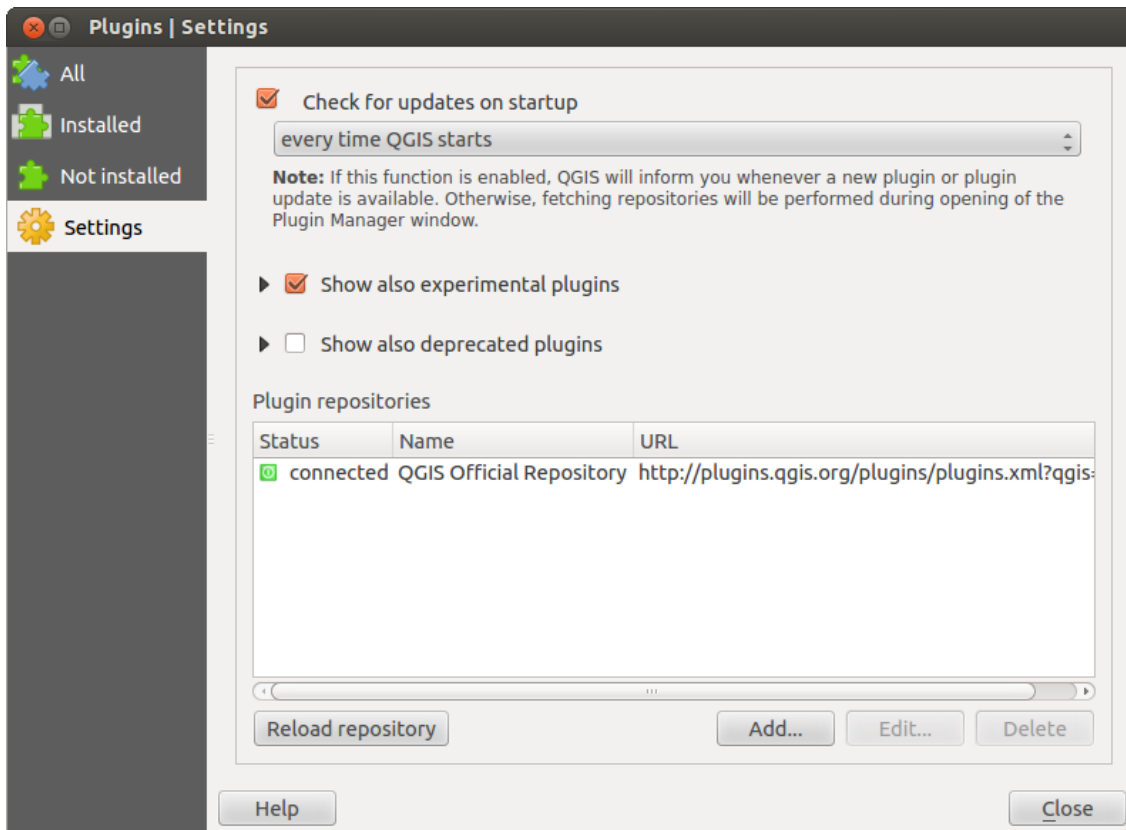



图 25.1: The  Settings tab

The All tab

In the  *All* tab, all the available plugins are listed, including both core and external plugins. Use *Upgrade All* to look for new versions of the plugins. Furthermore, you can use *Install Plugin* if a plugin is listed but not installed, *Uninstall Plugin* as well as *Reinstall Plugin* if a plugin is installed. An installed plugin can be temporarily de/activated using the checkbox.

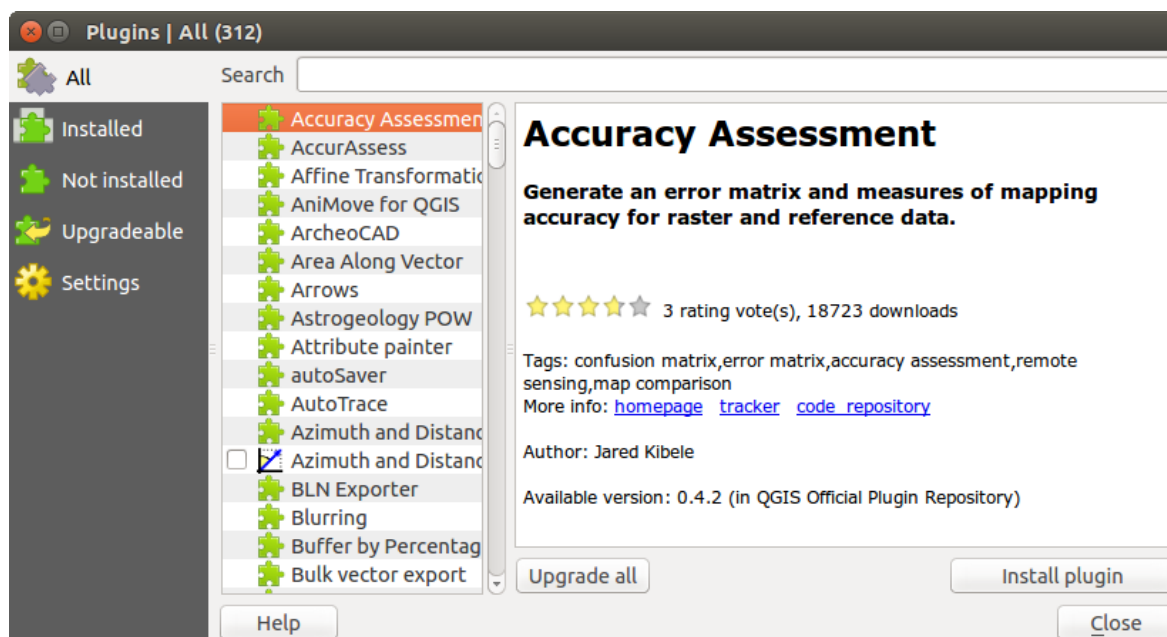


图 25.2: The  All tab

The Installed tab


In the  *Installed* tab, you'll find listed the Core plugins, that you can not uninstall. You can extend this list with external plugins that can be uninstalled and reinstalled any time, using the *Uninstall Plugin* and *Reinstall Plugin* buttons. You can *Upgrade All* the plugins here as well.



图 25.3: The  *Installed* tab

The Not installed tab





The  *Not installed* tab lists all plugins available that are not installed. You can use the *Install Plugin* button to implement a plugin into QGIS.



图 25.4: The  *Not installed* tab

The Upgradeable and New tabs

The  *Upgradeable* and  *New* tabs are enabled when new plugins are added to the repository or a new version of an installed plugin is released. If you activated *Show also experimental plugins* in the  *Settings* menu, those also appear in the list giving you opportunity to early test upcoming tools.

Installation can be done with the *Install Plugin*, *Upgrade Plugin* or *Upgrade All* buttons.

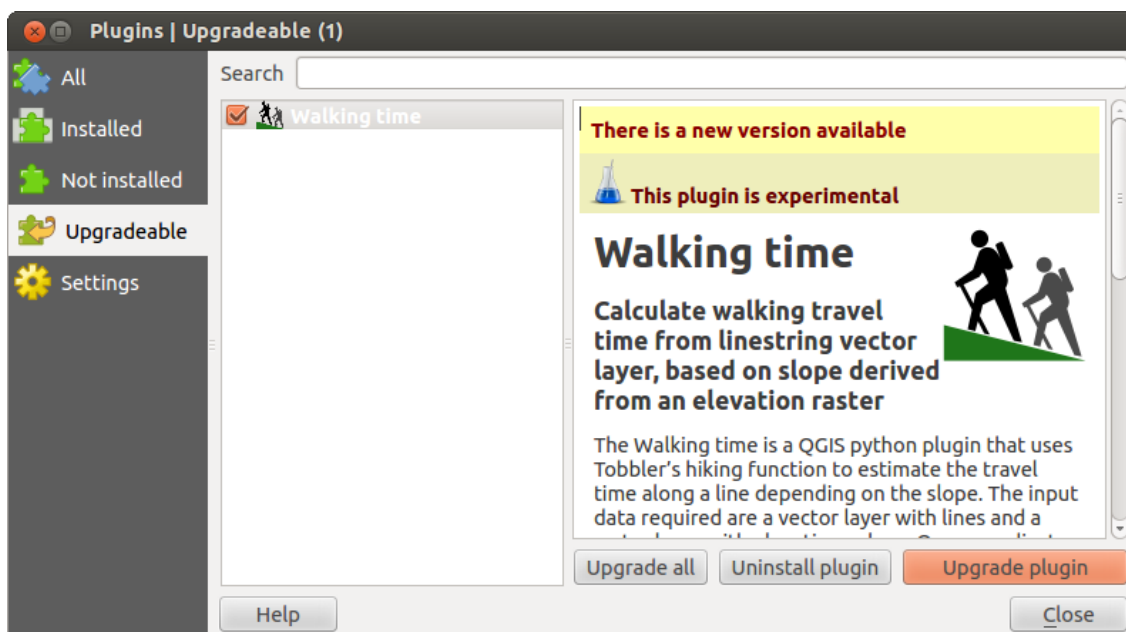



图 25.5: The  Upgradeable tab

The Invalid tab

The  *Invalid* tab lists all installed plugins that are currently broken for any reason (missing dependency, errors while loading, incompatible functions with QGIS version...). You can try the *Reinstall Plugin* button to fix an invalidated plugin but most of the times the fix will be elsewhere (install some libraries, look for another compatible plugin or help to upgrade the broken one).

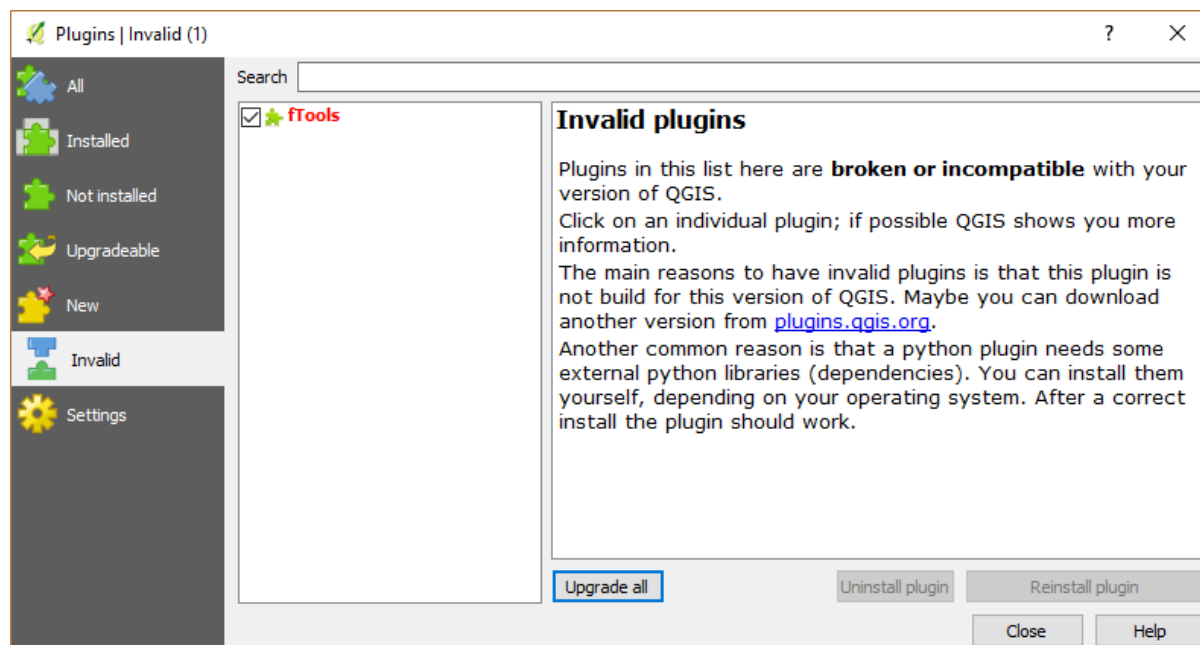



图 25.6: The  Invalid tab

The Install from ZIP tab

The  *Install from ZIP* tab provides a file selector widget to import plugins in a zipped format, e.g. plugins downloaded directly from their repository.

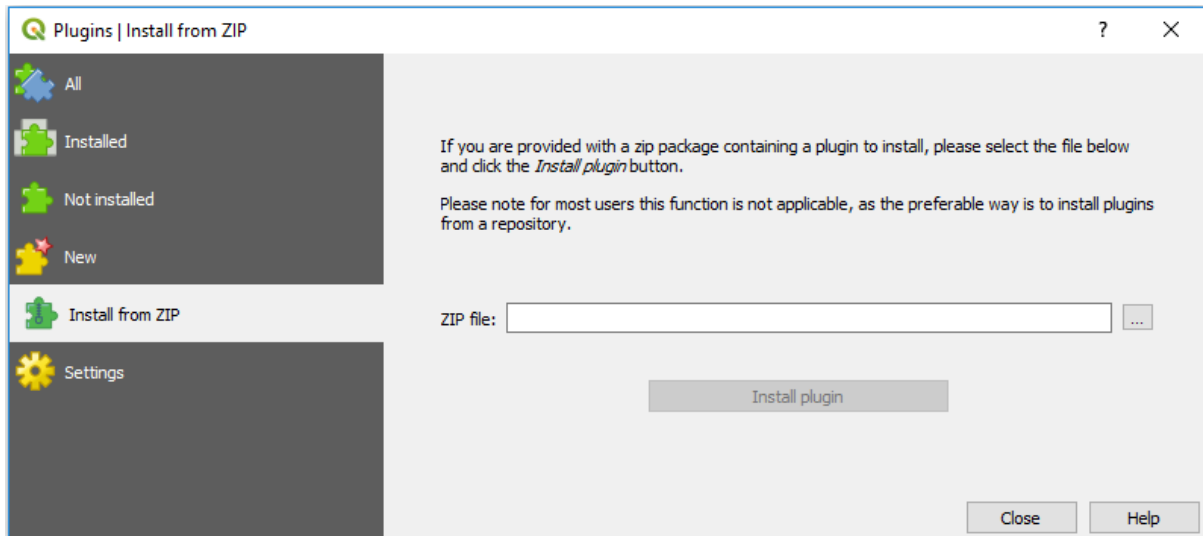



图 25.7: The  *Install from zip* tab

25.2 使用 QGIS 核心插件

25.2.1 DB Manager Plugin

The DB Manager Plugin is intended to be the main tool to integrate and manage spatial database formats supported by QGIS (PostGIS, SpatiaLite, GeoPackage, Oracle Spatial, Virtual layers) in one user interface. The  DB Manager Plugin provides several features. You can drag layers from the QGIS Browser into the DB Manager, and it will import your layer into your spatial database. You can drag and drop tables between spatial databases and they will get imported.

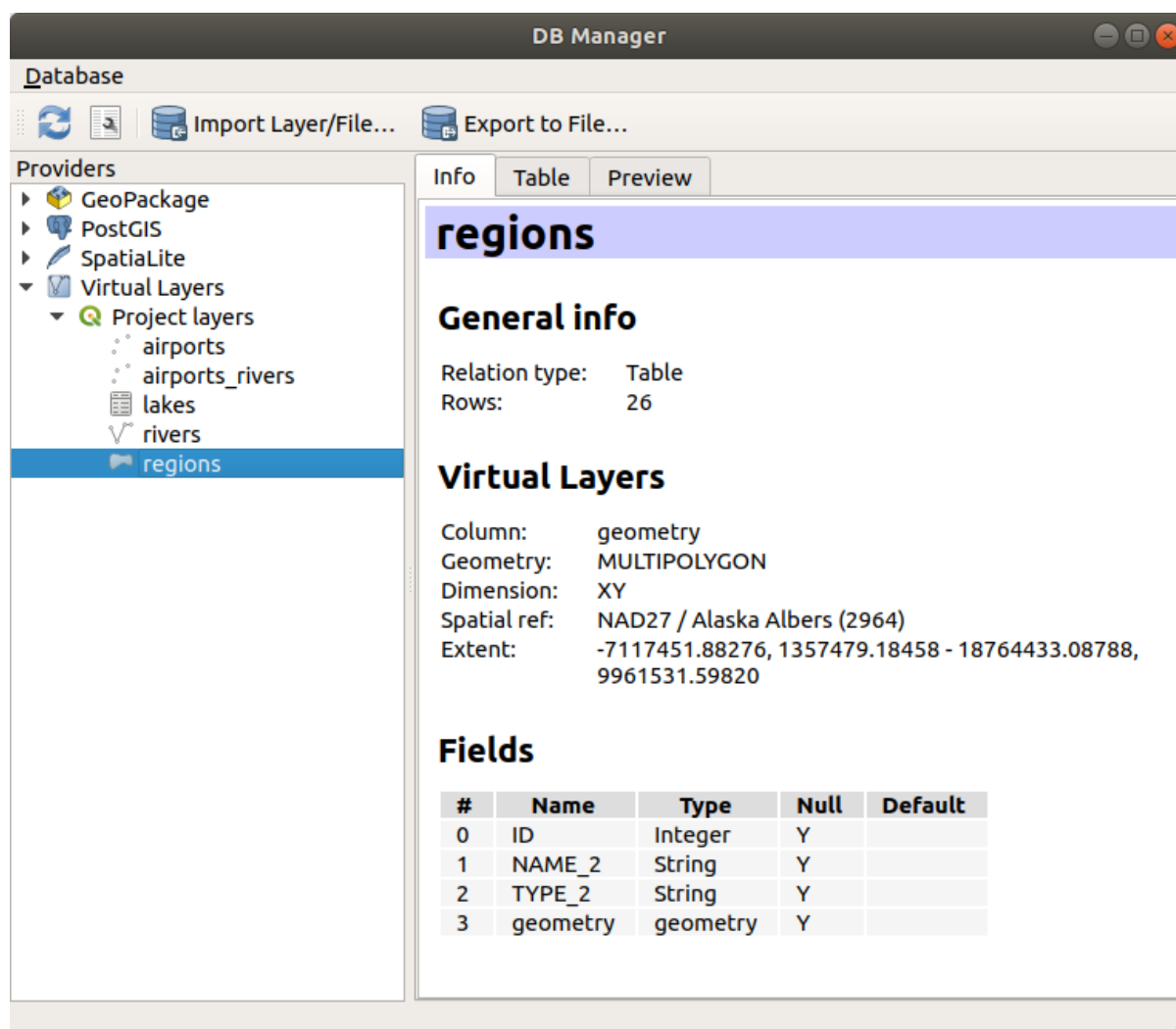


图 25.8: DB Manager dialog

The *Database* menu allows you to connect to an existing database, to start the SQL window and to exit the DB Manager Plugin. Once you are connected to an existing database, the menus *Schema* (relevant for DBMSs, such as PostGIS / PostgreSQL) and *Table* will appear.

The *Schema* menu includes tools to create and delete (only if empty) schemas and, if topology is available (e.g. with PostGIS topology), to start a *TopoViewer*.

The *Table* menu allows you to create and edit tables and to delete tables and views. It is also possible to empty tables and to move tables between schemas. You can *Run Vacuum Analyze* for the selected table. *Vacuum* reclaims space and makes it available for reuse, and *analyze* updates statistics that is used to determine the most efficient way to execute a query. *Change Logging...* allows you to add change logging support to a table. Finally, you can *Import Layer/File...* and *Export to File...*

The *Providers* window lists all existing databases supported by QGIS. With a double-click, you can connect to the database. With the right mouse button, you can rename and delete existing schemas and tables. Tables can also be added to the QGIS canvas with the context menu.

If connected to a database, the **main** window of the DB Manager offers four tabs. The *Info* tab provides information about the table and its geometry, as well as about existing fields, constraints and indexes. It allows you to create a spatial index on a the selected table. The *Table* tab shows the table, and the *Preview* tab renders the geometries as preview. When you open an *SQL Window*, it will be placed in a new tab.

Working with the SQL Window

You can use the DB Manager to execute SQL queries against your spatial database. Queries can be saved and loaded, and there the *SQL Query Builder* will help you formulate your queries. You can even view spatial output by checking *Load as new layer* and specifying *Column(s) with unique values* (IDs), *Geometry column* and *Layer name (prefix)*. It is possible to highlight a portion of the SQL to only execute that portion when pressing `Ctrl+R` or clicking the *Execute* button.

The *Query History* button stores the last 20 queries of each database and provider.

Double clicking on an entry will add the string to the SQL window.

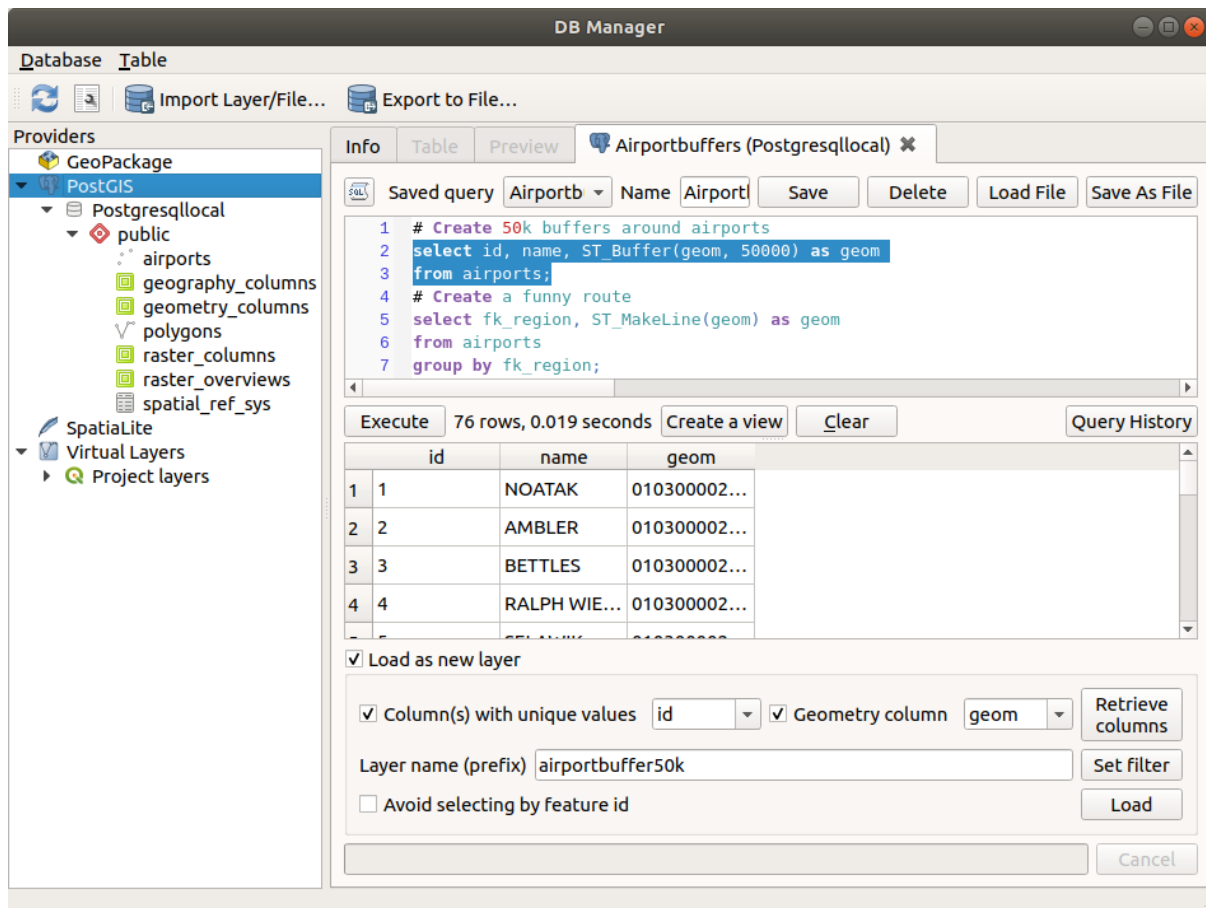



图 25.9: Executing SQL queries in the DB Manager SQL window

注解: The SQL Window can also be used to create Virtual Layers. In that case, instead of selecting a database, select **QGIS Layers** under **Virtual Layers** before opening the SQL Window. See *Creating virtual layers* for instructions on the SQL syntax to use.



25.2.2 Geometry Checker Plugin

Geometry Checker is a powerful core plugin to check and fix the geometry validity of a layer. It is available from the *Vector* menu ( *Check Geometries...*).

Configuring the checks

The *Check Geometries* dialog shows different grouped settings in the first tab (*Setup*):

- *Input vector layers*: to select the layers to check. A *Only selected features* checkbox can be used to restrict the checking to the geometries of the selected features.
- *Allowed geometry types* gives the chance to restrict the geometry type of the input layer(s) to:
 - Point
 - Multipoint
 - Line
 - Multiline
 - Polygon
 - Multipolygon
- *Geometry validity*. Depending on geometry types you can choose between:
 - *Self intersections*
 - *Duplicate nodes*
 - *Self contacts*
 - *Polygon with less than 3 nodes.*
- *Geometry properties*. Depending on geometry types, different options are available:
 - *Polygons and multipolygons may not contain any holes*
 - *Multipart objects must consist of more than one part*
 - *Lines must not have dangles*
- *Geometry conditions*. Allows you to add some condition to validate the geometries with:
 - *Minimal segment length (map units)*
 - *Minimum angle between segment (deg)*
 - *Minimal polygon area (map units sqr.)*
 - *No sliver polygons with a Maximum thinness* and a *Max. area (map units sqr.)*
- *Topology checks*. Depending on geometry types, many different options are available:
 - *Checks for duplicates*
 - *Checks for features within other features*
 - *Checks for overlaps smaller than*
 - *Checks for gaps smaller than*

- *Points must be covered by lines*
- *Points must properly lie inside a polygon*
- *Lines must not intersect any other lines*
- *Lines must not intersect with features of layer* 
- *Polygons must follow boundaries of layer* 
- *Tolerance*. You can define the tolerance of the check in map layer units.
- *Output vector layer* gives the choice to:
 - *Modify input layer*
 - *Create new layers*

When you are happy with the configuration, you can click on the *Run* button.

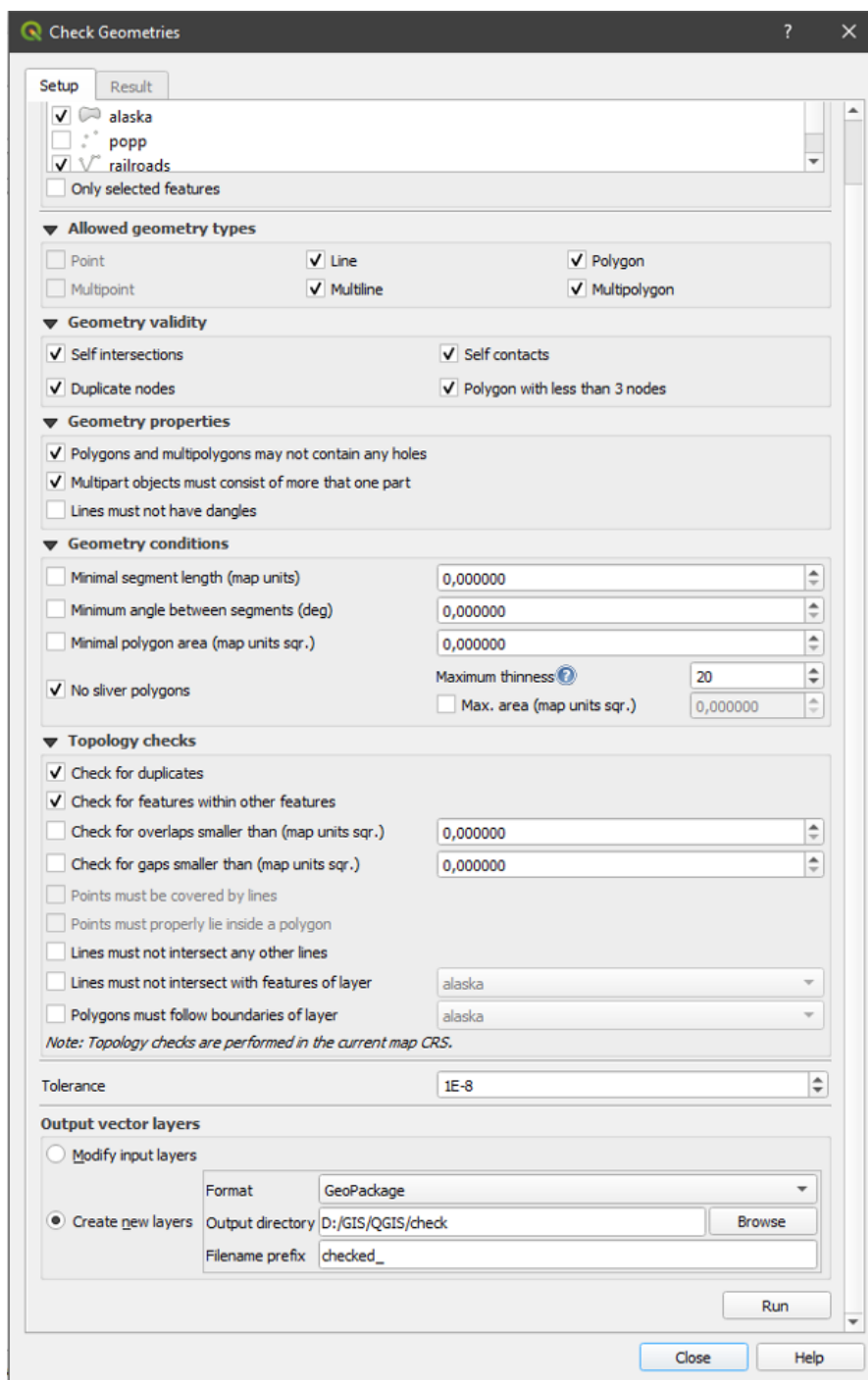


图 25.10: The Geometry Checker Plugin

The *Geometry Checker Plugin* can find the following errors:

- Self intersections: a polygon with a self intersection
- Duplicate nodes: two duplicates nodes in a segment
- Holes: hole in a polygon
- Segment length: a segment length lower than a threshold
- Minimum angle: two segments with an angle lower than a threshold
- Minimum area: polygon area lower than a threshold

- Silver polygon: this error come from very small polygon (with small area) with a large perimeter
- Duplicates features
- Feature within feature
- Overlaps: polygon overlapping
- Gaps: gaps between polygons

The following figure shows the different checks made by the plugin.

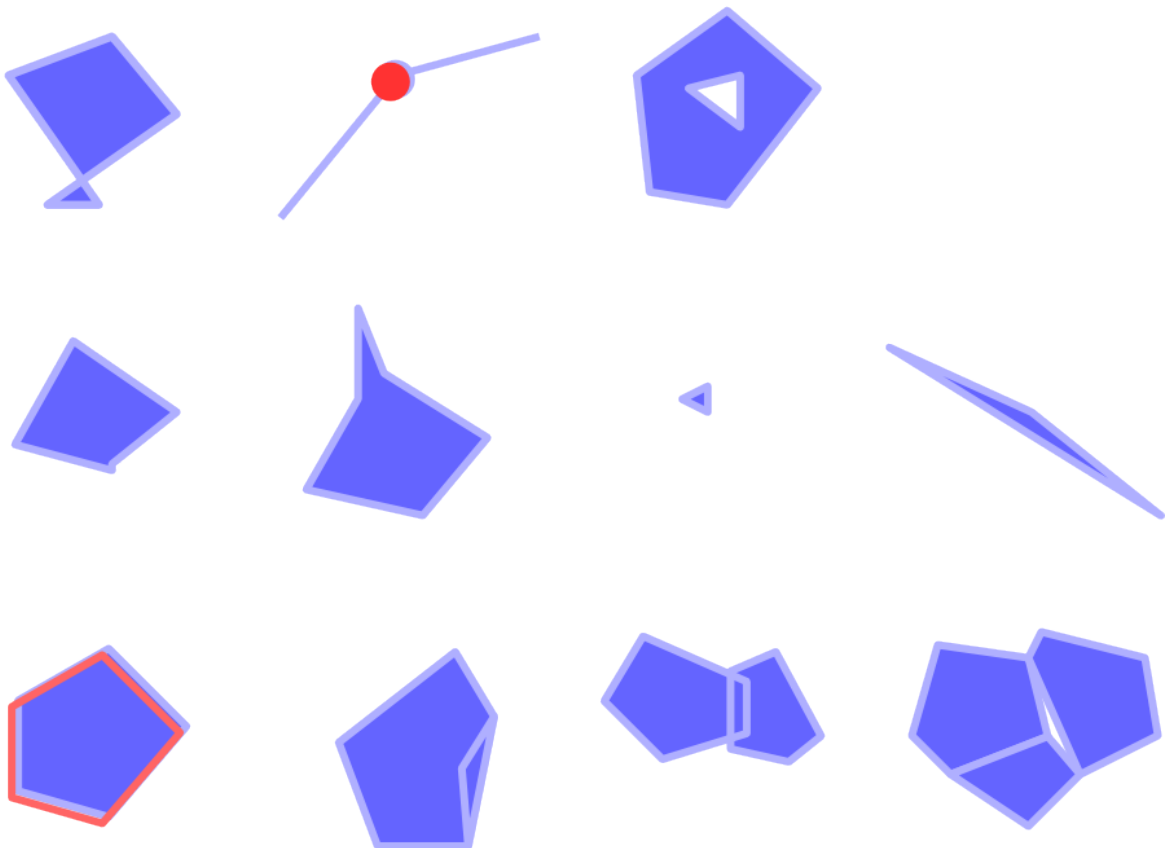


图 25.11: Some checks supported by the plugin

Analysing the results


The results appear in the second tab (*Result*) and as an overview layer of the errors in the canvas (its name has the default prefix *checked_*). A table lists the *Geometry check result* with one error per row and columns containing: the layer name, an ID, the error type, then the coordinates of the error, a value (depending on the type of the error) and finally the resolution column which indicates the resolution of the error. At the bottom of this table, you can *Export* the error into different file formats. You also have a counter with the number of total errors and fixed ones.

You can select a row to see the location of the error. You can change this behavior by selecting another action between

Error (default), *Feature*, *Don't move*, and *Highlight contour of selected features*.

Below the zoom action when clicking on the table row, you can:

- *Show selected features in attribute table*
- *Fix selected errors using default resolution*
- *Fix selected errors, prompt for resolution method* You will see a window to choose the resolution's method among which:

- Merge with neighboring polygon with longest shared edge
- Merge with neighboring polygon with largest area
- Merge with neighboring polygon with identical attribute value, if any, or leave as is
- Delete feature
- No action
-  *Error resolution settings* allows you to change the default resolution method depending on the error type

小技巧: Fix multiple errors

You can fix multiple errors by selecting more than one row in the table with the *CTRL + click* action.

Finally, you can choose which *Attribute to use when merging features by attribute value*.

25.2.3 MetaSearch Catalog Client

简介

MetaSearch is a QGIS plugin to interact with metadata catalog services, supporting the OGC Catalog Service for the Web (CSW) standard.

MetaSearch provides an easy and intuitive approach and user-friendly interface to searching metadata catalogs within QGIS.

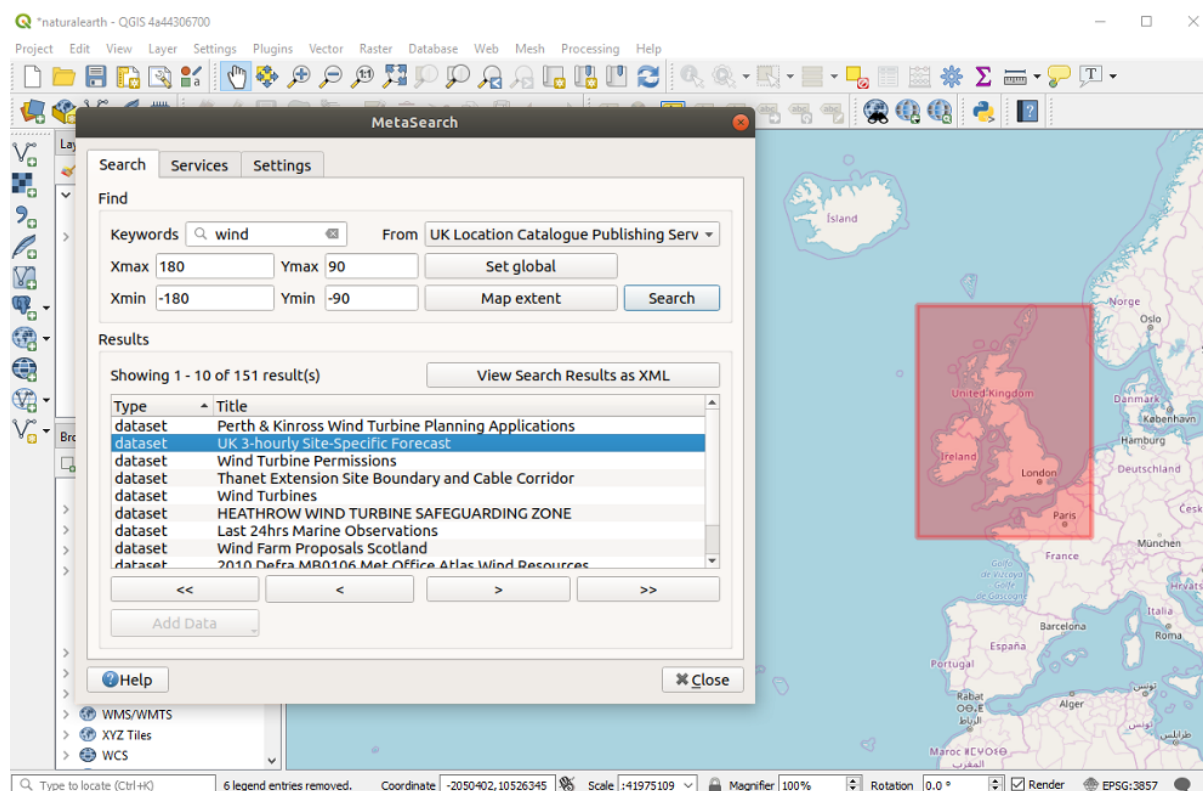


图 25.12: Search and results of Services in MetaSearch


Working with Metadata Catalogs in QGIS

MetaSearch is included by default in QGIS, with all of its dependencies, and can be enabled from the QGIS Plugin Manager.

CSW (Catalog Service for the Web)

CSW (Catalog Service for the Web) is an OGC (Open Geospatial Consortium) specification that defines common interfaces to discover, browse and query metadata about data, services, and other potential resources.

Startup

To start MetaSearch, click the  icon or select *Web* ► *MetaSearch* ► *MetaSearch* via the QGIS main menu. The MetaSearch dialog will appear. The main GUI consists of three tabs: *Services*, *Search* and *Settings*.

Managing Catalog Services

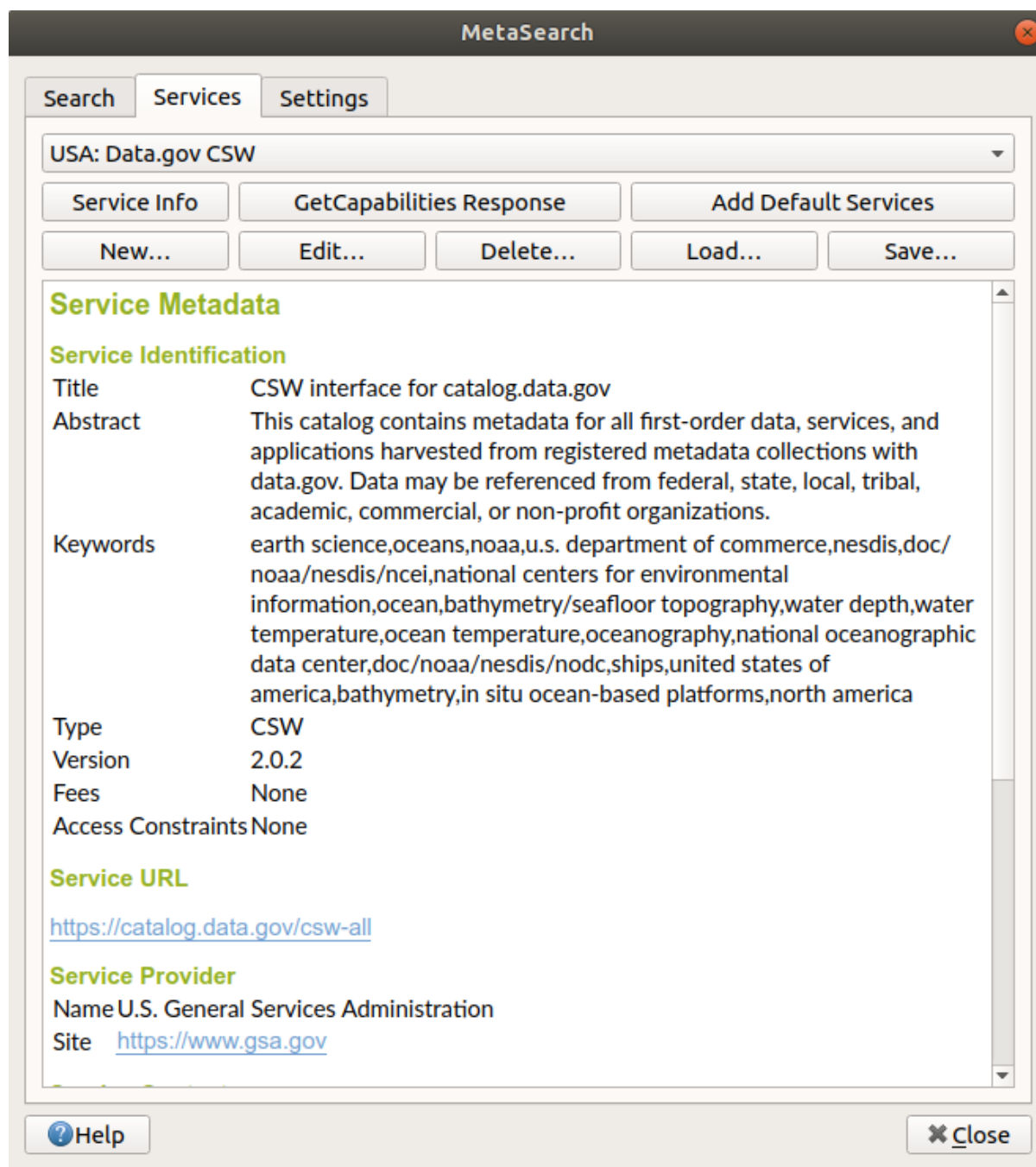


图 25.13: Managing Catalog Services

The *Services* tab allows the user to manage all available catalog services. MetaSearch provides a default list of Catalog Services, which can be added by pressing the *Add Default Services* button.

To find all listed Catalog Service entries, click the dropdown select box.

To add a Catalog Service entry:

1. Click the *New* button
2. Enter a *Name* for the service, as well as the *URL* (endpoint). Note that only the base URL is required (not a full GetCapabilities URL).

3. If the CSW requires authentication, enter the appropriate *User name* and *Password* credentials.
4. Click *OK* to add the service to the list of entries.

To edit an existing Catalog Service entry:

1. Select the entry you would like to edit
2. Click the *Edit* button
3. And modify the *Name* or *URL* values
4. Click *OK*.

To delete a Catalog Service entry, select the entry you would like to delete and click the *Delete* button. You will be asked to confirm deleting the entry.

MetaSearch allows loading and saving connections to an XML file. This is useful when you need to share settings between applications. Below is an example of the XML file format.

```
<?xml version="1.0" encoding="UTF-8"?>
<qgsCSWConnections version="1.0">
  <csw name="Data.gov CSW" url="https://catalog.data.gov/csw-all"/>
  <csw name="Geonorge - National CSW service for Norway" url="https://www.
↵geonorge.no/geonetwork/srv/eng/csw"/>
  <csw name="Geoportale Nazionale - Servizio di ricerca Italiano" url="http://
↵www.pcn.minambiente.it/geoportal/csw"/>
  <csw name="LINZ Data Service" url="http://data.linz.govt.nz/feeds/csw"/>
  <csw name="Nationaal Georegister (Nederland)" url="http://www.
↵nationaalgeoregister.nl/geonetwork/srv/eng/csw"/>
  <csw name="RNDT - Repertorio Nazionale dei Dati Territoriali - Servizio di
↵ricerca" url="http://www.rndt.gov.it/RNDT/CSW"/>
  <csw name="UK Location Catalogue Publishing Service" url="http://csw.data.gov.
↵uk/geonetwork/srv/en/csw"/>
  <csw name="UNEP/GRID-Geneva Metadata Catalog" url="http://metadata.grid.unep.
↵ch:8080/geonetwork/srv/eng/csw"/>
</qgsCSWConnections>
```

To load a list of entries:

1. Click the *Load* button. A new window will appear.
2. Click the *Browse* button and navigate to the XML file of entries you wish to load.
3. Click *Open*. The list of entries will be displayed.
4. Select the entries you wish to add from the list and click *Load*.

Click the *Service Info* button to display information about the selected Catalog Service such as service identification, service provider and contact information. If you would like to view the raw XML response, click the *GetCapabilities Response* button. A separate window will open displaying the Capabilities XML.

Searching Catalog Services

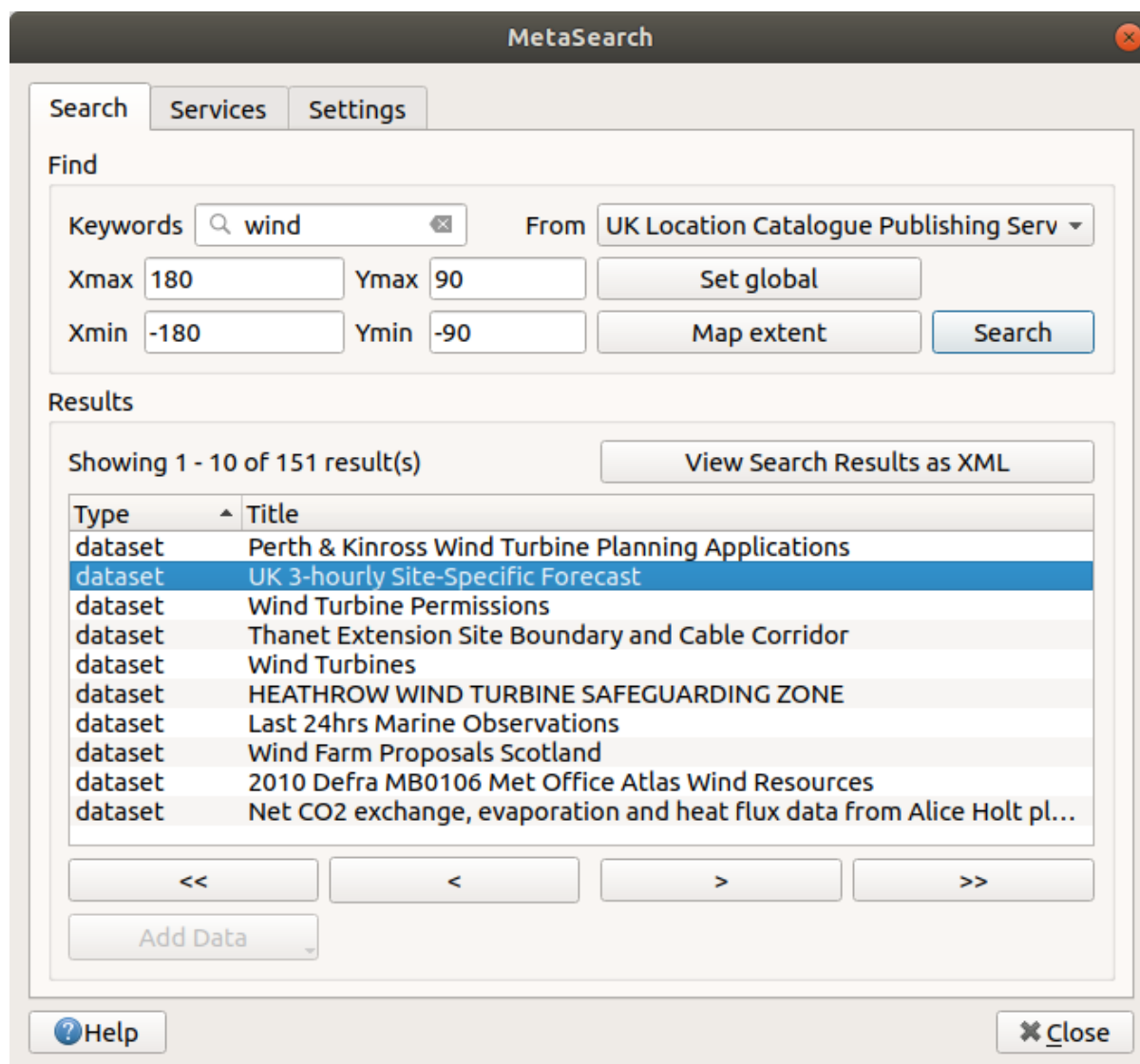


图 25.14: Searching catalog services

The *Search* tab allows the user to query Catalog Services for data and services, set various search parameters and view results.

The following search parameters are available:

- *Keywords*: free text search keywords;
- *From*: the Catalog Service to perform the query against;
- **Bounding box**: the spatial area of interest to filter, defined by *Xmax*, *Xmin*, *Ymax*, and *Ymin*. Click *Set Global* to do a global search, click *Map Extent* to do a search in the visible area, or enter values manually.

Clicking the *Search* button will search the selected Metadata Catalog. Search results are displayed in a list, and can be sorted by clicking on the column header. You can navigate through search results with the directional buttons below the search results.

Select a result and:

- Click the *View Search Results as XML* button to open a window with the service response in raw XML format.

- If the metadata record has an associated bounding box, a footprint of the bounding box will be displayed on the map.
- Double-click the record to display the record metadata with any associated access links. Clicking a link opens the link in the user's web browser.
- If the record is a supported web service (WMS/WMTS, WFS, WCS, ArcGIS Map Service, ArcGIS Feature Service, etc.), the *Add Data* button will be enabled. When clicking this button, MetaSearch will verify if this is a valid OWS. The service will then be added to the appropriate QGIS connection list, and the appropriate connection dialog will appear.

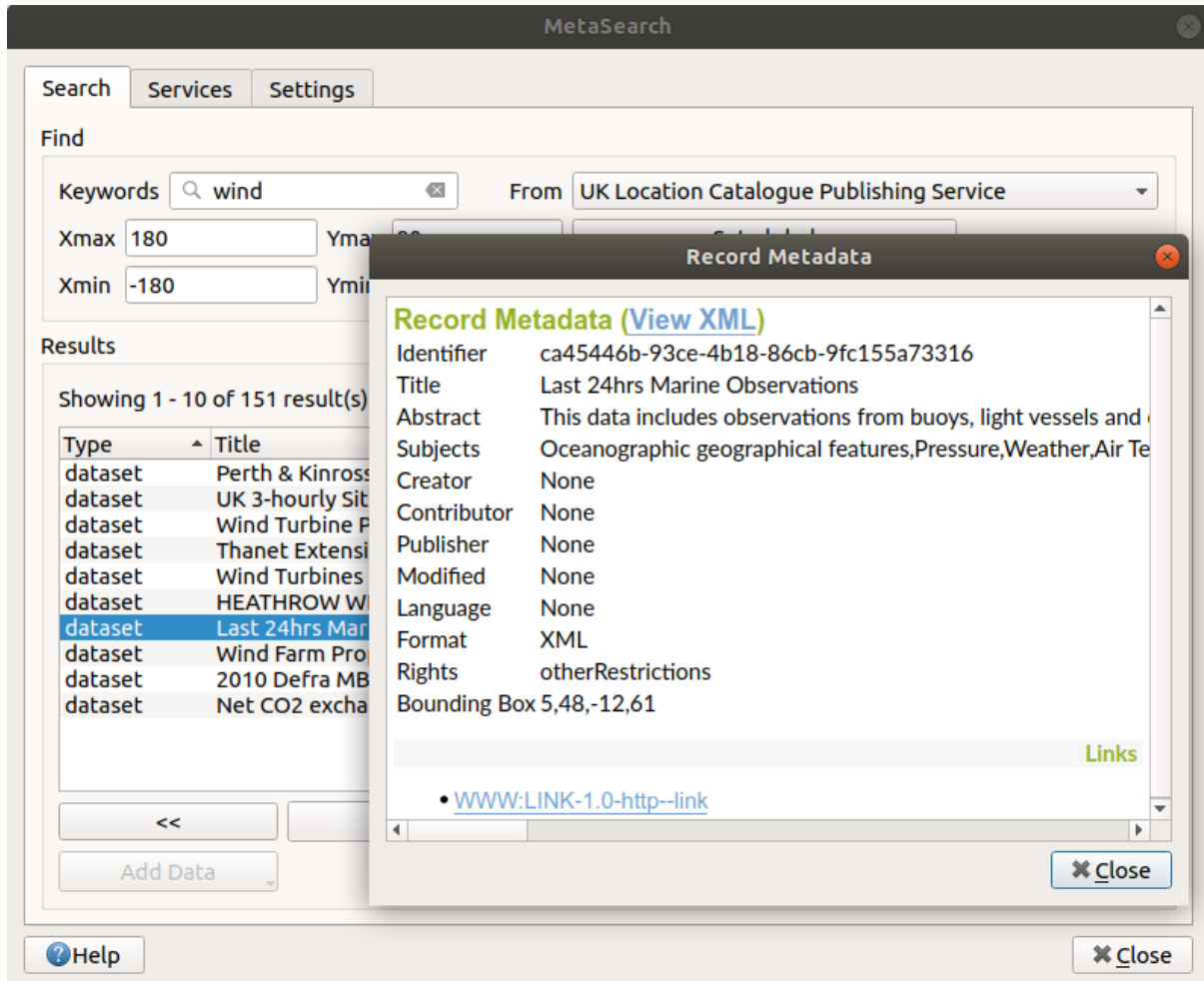


图 25.15: Metadata record display

Settings

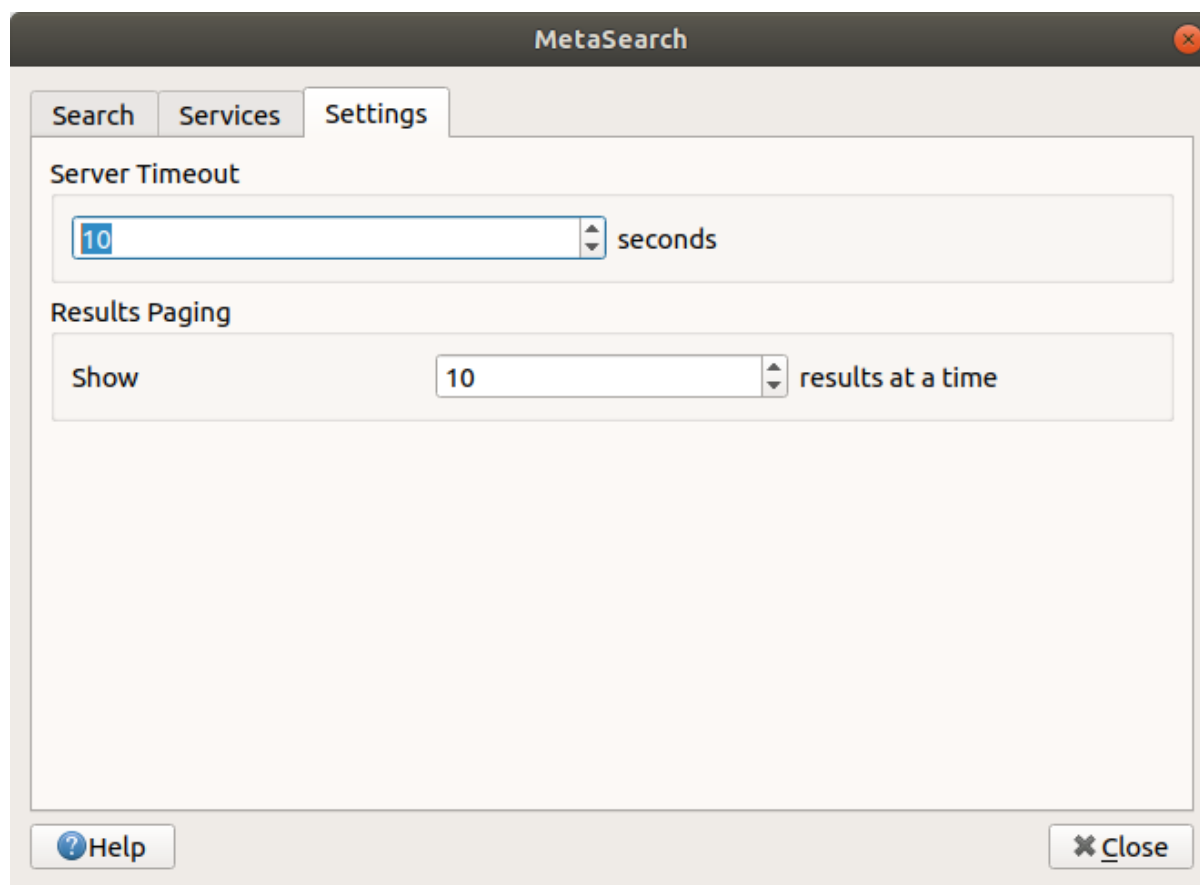


图 25.16: MetaSearch settings

You can fine tune MetaSearch with the following *Settings*:


- *Server Timeout*: when searching metadata catalogs, the number of seconds for blocking connection attempt. Default value is 10.
- *Results paging*: when searching metadata catalogs, the number of results to show per page. Default value is 10.

CSW Server Errors



In some cases, the CSW will work in a web browser, but not in MetaSearch. This may be due to the CSW server's configuration/setup. CSW server providers should ensure URLs are consistent and up to date in their configuration (this is common in HTTP -> HTTPS redirection scenarios). Please see the [pycsw FAQ item](#) for a deeper explanation of the issue and fix. Although the FAQ item is pycsw specific it can also apply in general to other CSW servers.

25.2.4 Offline Editing Plugin

For data collection, it is a common situation to work with a laptop or a cell phone offline in the field. Upon returning to the network, the changes need to be synchronized with the master datasource (e.g., a PostGIS database). If several persons are working simultaneously on the same datasets, it is difficult to merge the edits by hand, even if people don't change the same features.

The  Offline Editing Plugin automates the synchronisation by copying the content of a datasource (usually PostGIS or WFS-T) to a SpatiaLite or GeoPackage database and storing the offline edits to dedicated tables. After being connected to the network again, it is possible to apply the offline edits to the master dataset.

To use the plugin:

1. Open a project with some vector layers (e.g., from a PostGIS or WFS-T datasource).
2. Assuming you have already enabled the plugin (see *Core and External plugins*) go to *Database ► Offline Editing ►  Convert to offline project*. The eponym dialog opens.
3. Select the *Storage type*. It can be of *GeoPackage* or *SpatiaLite* database type.
4. Use the *Browse* button to indicate the location of the database in which to store the *Offline data*. It can be an existing file or one to create.
5. In the *Select remote layers* section, check the layers you'd like to save. The content of the layers is saved to database tables.
6. You can check *Only synchronize selected features if a selection is present* allowing to only save and work on a subset. It can be invaluable in case of large layers.
This is all!
7. Save your project and bring it on the field.
8. Edit the layers offline.
9. After being connected again, upload the changes using *Database ► Offline Editing ►  Synchronize*.

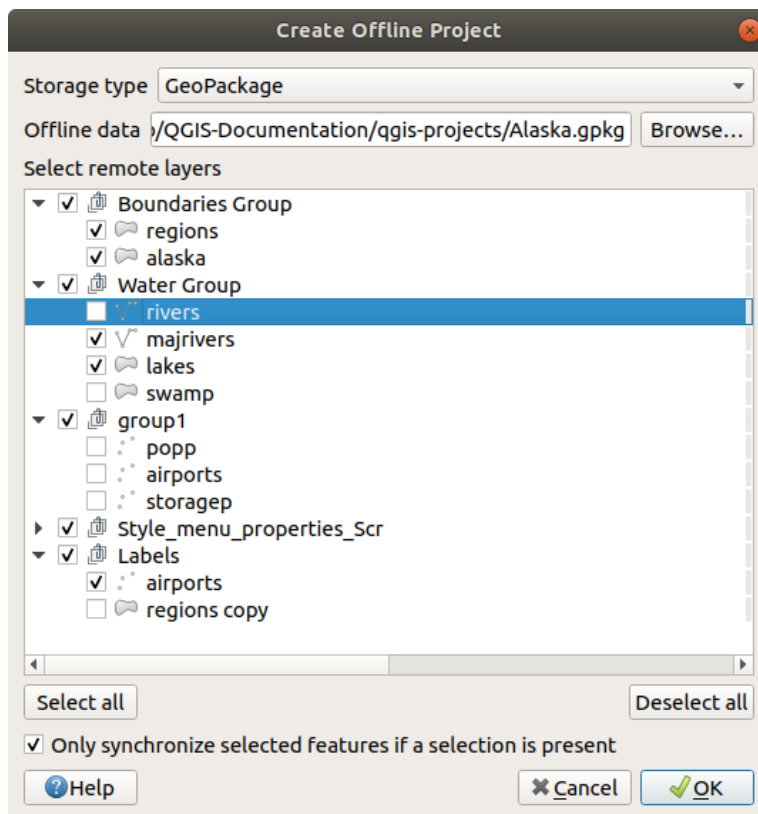


图 25.17: Create an offline project

25.2.5 Topology Checker Plugin

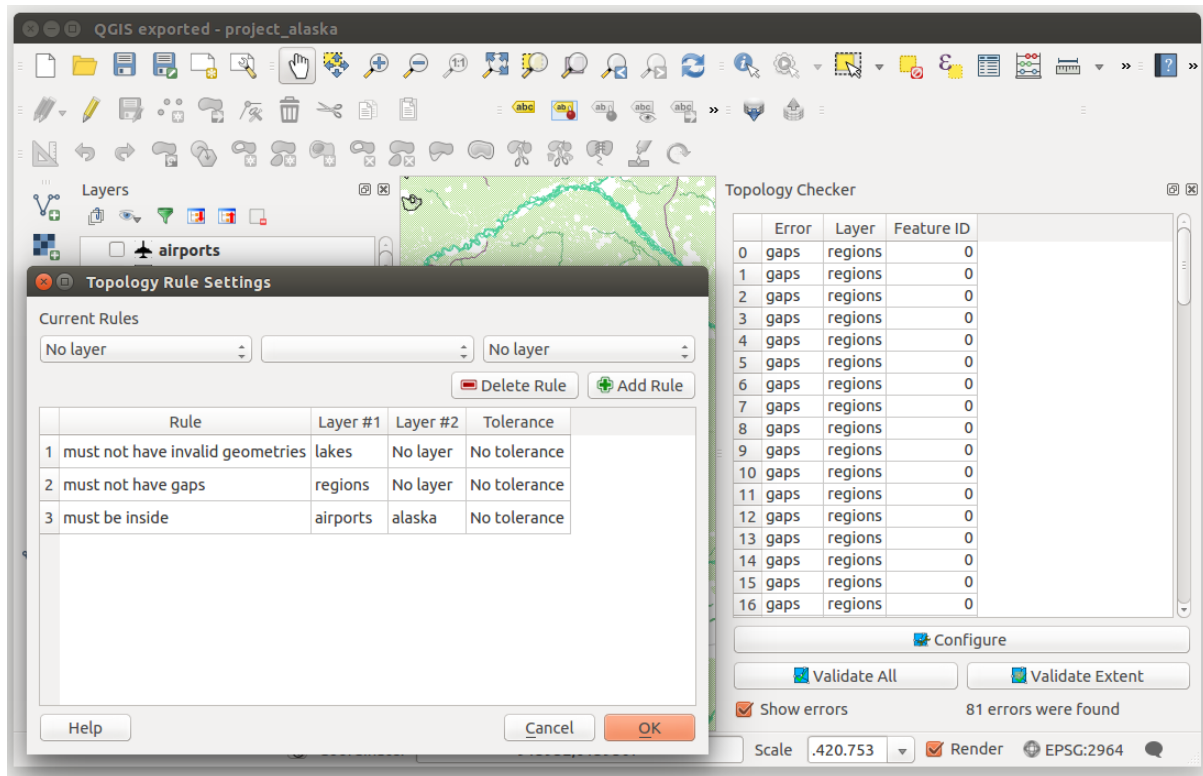


图 25.18: The Topology Checker Plugin

Topology describes the relationships between points, lines and polygons that represent the features of a geographic region. With the Topology Checker plugin, you can look over your vector files and check the topology with several topology rules. These rules check with spatial relations whether your features 'Equal', 'Contain', 'Cover', are 'CoveredBy', 'Cross', are 'Disjoint', 'Intersect', 'Overlap', 'Touch' or are 'Within' each other. It depends on your individual questions which topology rules you apply to your vector data (e.g., normally you won't accept overshoots in line layers, but if they depict dead-end streets you won't remove them from your vector layer).

QGIS has a built-in topological editing feature, which is great for creating new features without errors. But existing data errors and user-induced errors are hard to find. This plugin helps you find such errors through a list of rules.

It is very simple to create topology rules with the Topology Checker plugin.

On **point layers** the following rules are available:

- **Must be covered by:** Here you can choose a vector layer from your project. Points that aren't covered by the given vector layer occur in the 'Error' field.
- **Must be covered by endpoints of:** Here you can choose a line layer from your project.
- **Must be inside:** Here you can choose a polygon layer from your project. The points must be inside a polygon. Otherwise, QGIS writes an 'Error' for the point.
- **Must not have duplicates:** Whenever a point is represented twice or more, it will occur in the 'Error' field.
- **Must not have invalid geometries:** Checks whether the geometries are valid.
- **Must not have multi-part-geometries:** All multi-part points are written into the 'Error' field.

On **line layers**, the following rules are available:

- **End points must be covered by:** Here you can select a point layer from your project.
- **Must not have dangles:** This will show the overshoots in the line layer.

- **Must not have duplicates:** Whenever a line feature is represented twice or more, it will occur in the 'Error' field.
- **Must not have invalid geometries:** Checks whether the geometries are valid.
- **Must not have multi-part geometries:** Sometimes, a geometry is actually a collection of simple (single-part) geometries. Such a geometry is called multi-part geometry. If it contains just one type of simple geometry, we call it multi-point, multi-linestring or multi-polygon. All multi-part lines are written into the 'Error' field.
- **Must not have pseudos:** A line geometry's endpoint should be connected to the endpoints of two other geometries. If the endpoint is connected to only one other geometry's endpoint, the endpoint is called a pseudo node.

On **polygon layers**, the following rules are available:

- **Must contain:** Polygon layer must contain at least one point geometry from the second layer.
- **Must not have duplicates:** Polygons from the same layer must not have identical geometries. Whenever a polygon feature is represented twice or more it will occur in the 'Error' field.
- **Must not have gaps:** Adjacent polygons should not form gaps between them. Administrative boundaries could be mentioned as an example (US state polygons do not have any gaps between them...).
- **Must not have invalid geometries:** Checks whether the geometries are valid. Some of the rules that define a valid geometry are:
 - Polygon rings must close.
 - Rings that define holes should be inside rings that define exterior boundaries.
 - Rings may not self-intersect (they may neither touch nor cross one another).
 - Rings may not touch other rings, except at a point.
- **Must not have multi-part geometries:** Sometimes, a geometry is actually a collection of simple (single-part) geometries. Such a geometry is called multi-part geometry. If it contains just one type of simple geometry, we call it multi-point, multi-linestring or multi-polygon. For example, a country consisting of multiple islands can be represented as a multi-polygon.
- **Must not overlap:** Adjacent polygons should not share common area.
- **Must not overlap with:** Adjacent polygons from one layer should not share common area with polygons from another layer.

下面是 QGIS 提供的核心插件列表。默认情况下，它们不一定启用。

图标	插件	描述	手册参考
	数据库管理器	在 QGIS 中管理您的数据库	<i>DB Manager Plugin</i>
	几何图形检查器	检查并修复矢量几何图形中的错误	<i>Geometry Checker Plugin</i>
	GPS 工具	用于加载和导入 GPS 数据的工具	<i>GPS Plugin</i>
	GRASS	GRASS 功能	<i>GRASS GIS Integration</i>
	MetaSearch Catalog Client	与元数据目录服务交互 (CSW)	<i>MetaSearch Catalog Client</i>
	离线编辑	离线编辑并与数据库同步	<i>Offline Editing Plugin</i>
	数据处理	空间数据处理架构	<i>QGIS 处理框架</i>
	拓扑检查器	在矢量图层中查找拓扑错误	<i>Topology Checker Plugin</i>

25.3 QGIS Python console

As you will see later in this chapter, QGIS has been designed with a plugin architecture. Plugins can be written in Python, a very famous language in the geospatial world.

QGIS brings a Python API (see [PyQGIS Developer Cookbook](#) for some code sample) to let the user interact with its objects (layers, feature or interface). QGIS also has a Python console.






The QGIS Python Console is an interactive shell for the python command executions. It also has a python file editor that allows you to edit and save your python scripts. Both console and editor are based on PyQScintilla2 package. To open the console go to *Plugins* ► *Python Console* (Ctrl+Alt+P).

25.3.1 The Interactive Console

The interactive console is composed of a toolbar, an input area and an output one.

Toolbar

The toolbar proposes the following tools:

-  Clear Console to wipe the output area;
-  Run Command available in the input area: same as pressing Enter;
-  Show Editor: toggles *The Code Editor* visibility;
-  Options...: opens a dialog to configure console properties (see [Python 控制台设置](#));
-  Help...: browses the current documentation.

Console

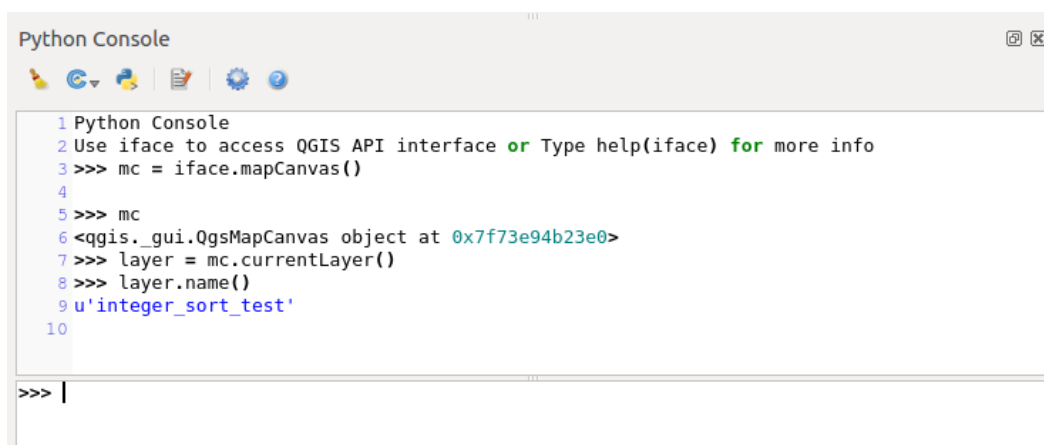
The console main features are:

- Code completion, highlighting syntax and calltips for the following APIs:
 - Python
 - PyQGIS
 - PyQt5
 - QScintilla2
 - osgeo-gdal-ogr
- Ctrl+Alt+Space to view the auto-completion list if enabled in the [Python 控制台设置](#);
- Execute code snippets from the input area by typing and pressing Enter or *Run Command*;
- Execute code snippets from the output area using the *Enter Selected* from the contextual menu or pressing Ctrl+E;
- Browse the command history from the input area using the Up and Down arrow keys and execute the command you want;
- Ctrl+Shift+Space to view the command history: double-clicking a row will execute the command. The *Command History* dialog can also be accessed from context menu of input area;
- Save and clear the command history. The history will be saved into the `console_history.txt` file under the active *user profile* folder;
- Open [QGIS C++ API](#) documentation by typing `_api`;

- Open [QGIS Python API](#) documentation by typing `_pyqgis`.
- Open [PyQGIS Cookbook](#) by typing `_cookbook`.

小技巧: Reuse executed commands from the output panel


You can execute code snippets from the output panel by selecting some text and pressing `Ctrl+E`. No matter if selected text contains the interpreter prompt (`>>>`, `...`).





```
Python Console
1 Python Console
2 Use iface to access QGIS API interface or Type help(iface) for more info
3 >>> mc = iface.mapCanvas()
4
5 >>> mc
6 <qgis._gui.QgsMapCanvas object at 0x7f73e94b23e0>
7 >>> layer = mc.currentLayer()
8 >>> layer.name()
9 u'integer_sort_test'
10
>>> |
```

图 25.19: The Python Console

25.3.2 The Code Editor

Use the  `Show Editor` button to enable the editor widget. It allows editing and saving Python files and offers advanced functionalities to manage your code (comment and uncomment code, check syntax, share the code via [codepad.org](#) and much more). Main features are:

- Code completion, highlighting syntax and calltips for the following APIs:
 - Python
 - PyQGIS
 - PyQt5
 - QScintilla2
 - osgeo-gdal-ogr
- `Ctrl+Space` to view the auto-completion list.
- Sharing code snippets via [codepad.org](#).
- `Ctrl+4` Syntax check.
- Search bar (open it with the default Desktop Environment shortcut, usually `Ctrl+F`):
 - Use the default Desktop Environment shortcut to find next/previous (`Ctrl+G` and `Shift+Ctrl+G`);
 - Automatically find first match when typing in find box;
 - Set initial find string to selection when opening find;
 - Pressing `Esc` closes the find bar.
- Object inspector: a class and function browser;
- Go to an object definition with a mouse click (from Object inspector);

- Execute code snippets with the  *Run Selected* command in contextual menu;
- Execute the whole script with the  *Run Script* command (this creates a byte-compiled file with the extension `.pyc`).

注解: Running partially or totally a script from the *Code Editor* outputs the result in the Console output area.

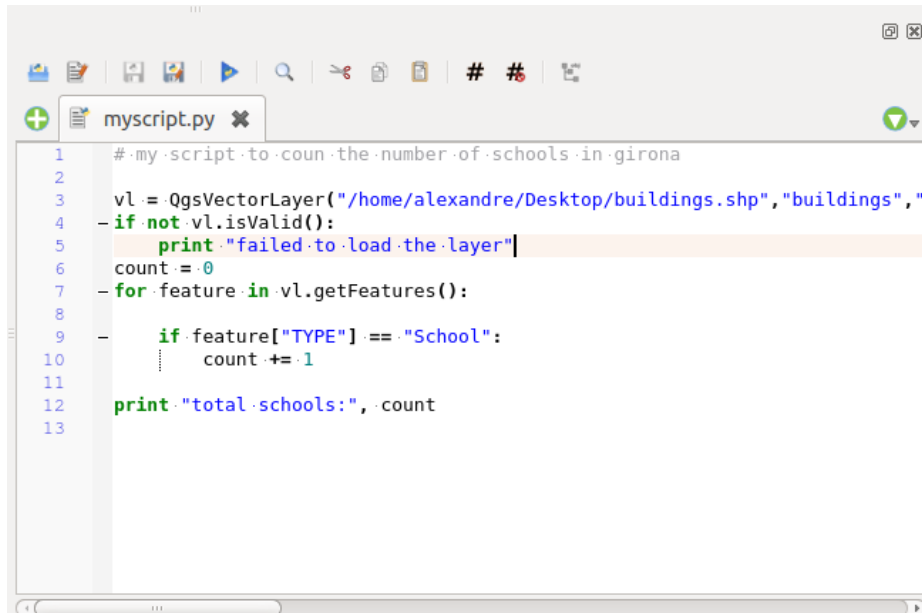


图 25.20: The Python Console editor

小技巧: Save the options

To save the state of console's widgets you have to close the Python Console from the close button. This allows you to save the geometry to be restored to the next start.

26.1 Mailing lists

QGIS is under active development and as such it won't always work like you expect it to. The preferred way to get help is by joining the qgis-users mailing list. Your questions will reach a broader audience and answers will benefit others.

26.1.1 QGIS Users

This mailing list is used for discussion about QGIS in general, as well as specific questions regarding its installation and use. You can subscribe to the qgis-users mailing list by visiting the following URL: <https://lists.osgeo.org/mailman/listinfo/qgis-user>

26.1.2 QGIS Developers

If you are a developer facing problems of a more technical nature, you may want to join the qgis-developer mailing list. This list is also a place where people can chime in and collect and discuss QGIS related UX (User Experience) / usability issues. It's here: <https://lists.osgeo.org/mailman/listinfo/qgis-developer>

26.1.3 QGIS Community Team

This list deals with topics like documentation, context help, user guide, web sites, blog, mailing lists, forums, and translation efforts. If you would like to work on the user guide as well, this list is a good starting point to ask your questions. You can subscribe to this list at: <https://lists.osgeo.org/mailman/listinfo/qgis-community-team>

26.1.4 QGIS Translations

This list deals with the translation efforts. If you like to work on the translation of the website, manuals or the graphical user interface (GUI), this list is a good starting point to ask your questions. You can subscribe to this list at: <https://lists.osgeo.org/mailman/listinfo/qgis-tr>

26.1.5 QGIS Project Steering Committee (PSC)

This list is used to discuss Steering Committee issues related to overall management and direction of QGIS. You can subscribe to this list at: <https://lists.osgeo.org/mailman/listinfo/qgis-psc>

26.1.6 QGIS User groups

In order to locally promote QGIS and contribute to its development, some QGIS communities are organized into QGIS User Groups. These groups are places to discuss local topics, organize regional or national user meetings, organize sponsoring of features... The list of current user groups is available at <https://qgis.org/en/site/forusers/usergroups.html>

You are welcome to subscribe to any of the lists. Please remember to contribute to the list by answering questions and sharing your experiences.

26.2 IRC

We also maintain a presence on IRC - visit us by joining the #qgis channel on irc.freenode.net. Please wait for a response to your question, as many folks on the channel are doing other things and it may take a while for them to notice your question. If you missed a discussion on IRC, not a problem! We log all discussion, so you can easily catch up. Just go to <http://irclogs.geoapt.com/qgis/> and read the IRC-logs.

26.3 Commercial support

Commercial support for QGIS is also available. Check the website https://qgis.org/en/site/forusers/commercial_support.html for more information.

26.4 BugTracker

While the qgis-users mailing list is useful for general 'How do I do XYZ in QGIS?'-type questions, you may wish to notify us about bugs in QGIS. You can submit bug reports using the [QGIS bug tracker](#).

Please bear in mind that your bug may not always enjoy the priority you might hope for (depending on its severity). Some bugs may require significant developer effort to remedy, and the manpower is not always available for this.

Feature requests can be submitted as well using the same ticket system as for bugs. Please make sure to select the type `Feature request`.

If you have found a bug and fixed it yourself, you can submit a Pull Request on the [Github QGIS Project](#).

Read [Bugs, Features and Issues](#) and `submit_patch` for more details.

26.5 Blog

The QGIS community also runs a weblog at <https://plugins.qgis.org/planet/>, which has some interesting articles for users and developers. Many other QGIS blogs exist, and you are invited to contribute with your own QGIS blog!

26.6 Plugins

The website <https://plugins.qgis.org> is the official QGIS plugins web portal. Here, you find a list of all stable and experimental QGIS plugins available via the 'Official QGIS Plugin Repository'.

26.7 Wiki

Lastly, we maintain a WIKI web site at <https://github.com/qgis/QGIS/wiki> where you can find a variety of useful information relating to QGIS development, release plans, links to download sites, message-translation hints and more. Check it out, there are some goodies inside!

QGIS 是由具有奉献精神志愿者和组织开发的开源项目，我们力争成为所有种族、信仰、性别及各行各业人士欢迎的社区，您可以随时加入我们。

27.1 作者

以下为投入大量时间和精力编写、审查和更新整个 QGIS 文档的人员。

Tim Sutton	Yves Jacolin	Jacob Lanstorp	Gary E. Sherman	Richard Duivenvoorde
Tara Athan	Anita Graser	Arnaud Morvan	Gavin Macaulay	Luca Casagrande
K. Koy	Hugo Mercier	Akbar Gumbira	Marie Silvestre	Jürgen E. Fischer
Fran Raga	Eric Goddard	Martin Dobias	Diethard Jansen	Saber Razmjooei
Ko Nagase	Nyall Dawson	Matthias Kuhn	Andreas Neumann	Harrissou Sant-anna
Manel Clos	David Willis	Larissa Junek	Paul Blottière	Sebastian Dietrich
Chris Mayo	Stephan Holl	Magnus Homann	Bernhard Ströbl	Alessandro Pasotti
N. Horning	Radim Blazek	Joshua Arnott	Luca Manganelli	Marco Hugentobler
Andre Mano	Mie Winstrup	Frank Sokolic	Vincent Picavet	Jean-Roc Morreale
Andy Allan	Victor Olaya	Tyler Mitchell	René-Luc D'Hont	Marco Bernasocchi
Ilkka Rinne	Werner Macho	Chris Berkhout	Nicholas Duggan	Jonathan Willitts
David Adler	Lars Luthman	Brendan Morely	Raymond Nijssen	Carson J.Q. Farmer
Jaka Kranjc	Mezene Worku	Patrick Sunter	Steven Cordwell	Stefan Blumentrath
Andy Schmid	Vincent Mora	Alexandre Neto	Hien Tran-Quang	Alexandre Busquets
João Gaspar	Tom Kralidis	Alexander Bruy	Paolo Cavallini	Milo Van der Linden
Peter Ersts	Ujaval Gandhi	Dominic Keller	Giovanni Manghi	Maximilian Krumbach
Anne Ghisla	Dick Groskamp	Uros Preloznik	Stéphane Brunner	QGIS Korean Translator
Zoltan Siki	Håvard Tveite	Mattheo Ghetta	Salvatore Larosa	Konstantinos Nikolaou
Tom Chadwin	Larry Shaffer	Nathan Woodrow	Martina Savarese	Godofredo Contreras
Astrid Emde	Luigi Pirelli	Thomas Gratier	Giovanni Allegri	GiordanoPezzola
Paolo Corti	Tudor Băräscu	Maning Sambale	Claudia A. Engel	Yoichi Kayama
Otto Dassau	Denis Rouzaud	Nick Bearman	embelding	ajazepk
Ramon	Andrei	zstadler	icephale	Rosa Aguilar

27.2 翻译者

QGIS 是一个多语言应用程序，并且还发布了翻译成多种语言的帮助文档。许多其他语言正在翻译中，一旦达到合理的翻译百分比就会发布。如果您希望帮助改进或申请新的语言翻译，请参阅 <https://qgis.org/en/site/getinvolved/index.html>。

参与当前翻译的人员：

语言	贡献者
印度尼西亚语	Emir Hartato, I Made Anombawa, Januar V. Simarmata, Muhammad Iqnaul Haq Siregar, Trias Aditya
中文 (繁体)	Calvin Ngei, Zhang Jun, Richard Xie
荷兰语	Carlo van Rijswijk, Dick Groskamp, Diethard Jansen, Raymond Nijssen, Richard Duivenvoorde, Willem Hoffman
芬兰语	Matti Mäntynen, Kari Mikkonen
法语	Arnaud Morvan, Augustin Roche, Didier Vanden Berghe, Dofabien, Etienne Trimaille, Francis Gasc, Harrissou Sant-anna, Jean-Roc Morreale, Jérémy Garniaux, Loïc Buscoz, Lsam, Marc-André Saia, Marie Silvestre, Mathieu Bossaert, Mathieu Lattes, Mayeul Kauffmann, Médéric Ribreux, Mehdi Semchaoui, Michael Douchin, Nicolas Boisteault, Nicolas Rochard, Pascal Obstetar, Robin Prest, Rod Bera, Stéphane Henriod, Stéphane Possamai, sylther, Sylvain Badey, Sylvain Maillard, Vincent Picavet, Xavier Tardieu, Yann Leveille-Menez, yoda89
加利西亚语	Xan Vieiro
德语	Jürgen E. Fischer, Otto Dassau, Stephan Holl, Werner Macho
印地语	Harish Kumar Solanki
意大利语	Alessandro Fanna, Anne Ghisla, Flavio Rigolon, Giuliano Curti, Luca Casagrande, Luca Delucchi, Marco Braidà, Matteo Ghetta, Maurizio Napolitano, Michele Beneventi, Michele Ferretti, Roberto Angeletti, Paolo Cavallini, Stefano Campus
日语	Baba Yoshihiko, Minoru Akagi, Norihiro Yamate, Takayuki Mizutani, Takayuki Nuimura, Yoichi Kayama
韩语	OSGeo Korean Chapter
波兰语	Andrzej Świąder, Borys Jurgiel, Ewelina Krawczak, Jakub Bobrowski, Mateusz Łoskot, Michał Kułach, Michał Smoczyk, Milena Nowotarska, Radosław Pasiok, Robert Szczepanek, Tomasz Paul
葡萄牙语	Alexandre Neto, Duarte Carreira, Giovanni Manghi, João Gaspar, Joana Simões, Leandro Infantini, Nelson Silva, Pedro Palheiro, Pedro Pereira, Ricardo Sena
葡萄牙语 (巴西)	Arthur Nanni, Felipe Sodrê Barros, Leônidas Descovi Filho, Marcelo Soares Souza, Narcélio de Sá Pereira Filho, Sidney Schaberle Goveia
罗马尼亚语	Alex Bădescu, Bogdan Pacurar, Georgiana Ioanovici, Lonut Losifescu-Enescu, Sorin Călinică, Tudor Bărăscu
俄语	Alexander Bruy, Artem Popov
西班牙语	Carlos Dávila, Diana Galindo, Edwin Amado, Gabriela Awad, Javier César Aldariz, Mayeul Kauffmann, Fran Raga
乌克兰语	Alexander Bruy

27.3 翻译统计

QGIS 3.16 长期发行版翻译工作的完成情况。

(last update: 2022-03-11)

字符串数	目标语言的数量	全部翻译率
32361	59	12.3%

语言	翻译率 (%)	语言	翻译率 (%)	语言	翻译率 (%)
阿尔巴尼亚语	0.23	阿拉伯语	4.02	阿塞拜疆语	0.02
巴斯克语	1.42	孟加拉语	0.19	保加利亚语	2.59
缅甸语	0.1	加泰罗尼亚语	1.51	简体中文	8.53
繁体中文	0.69	克罗地亚语	0.12	捷克语	6.0
丹麦语	0.66	荷兰语	100.0	爱沙尼亚语	1.3
芬兰语	1.81	法语	98.49	加利西亚语	0.59
格鲁吉亚语	0.11	德语	21.73	希腊语	0.37
希伯来语	0.74	印地语	0.31	匈牙利语	9.3
伊博语	0.01	印度尼西亚语	2.77	意大利语	88.87
日语	71.07	卡拜尔语	0.11	韩语	88.58
老挝语	0.0	立陶宛语	6.06	马其顿语	0.13
马来语	0.05	马拉雅拉姆语	0.1	马拉地语	0.19
蒙古语	0.11	曼德族语	1.82	挪威语	3.32
旁遮普语 (Punjabi)	0.0	波斯语	0.48	波兰语	1.85
葡萄牙语 (巴西)	37.01	葡萄牙语 (葡萄牙)	8.5	罗马尼亚语	30.61
俄语	14.94	塞尔维亚语	0.11	斯洛伐克语	1.55
斯洛维尼亚语	3.2	西班牙语	96.0	瑞典语	1.19
塔加洛语	0.1	泰米尔语	0.52	泰卢固语	0.03
泰国语	0.11	土耳其语	2.82	乌克兰语	2.37
乌尔都语	0.0	越南语	0.33		

28.1 附录 A: GNU 通用公共许可证 (GNU GPL)

版本 2, 1991 年 6 月

Copyright (C) 1989, 1991 Free Software Foundation, Inc. 59 Temple Place - Suite 330, Boston, MA 02111-1307, USA

任何人都可以复制和分发本授权文件的完整副本，但不允许更改。

序言

大多数软件的许可证都是为了剥夺你分享和修改软件的自由而设计的。相比之下，GNU 通用公共许可证旨在保障您共享和更改自由软件的自由，以确保软件对所有用户都是免费的。本通用公共许可证适用于自由软件基金会的大多数软件，以及其作者承诺使用它的任何其他程序。（一些其他的自由软件基金会软件被 GNU 库通用公共许可证所覆盖。）你也可以将它应用到你的程序中。

当我们谈到自由软件（free software）时，我们指的是自由而不是价格。我们的 GNU 通用公共许可证决意保证你有发布自由软件的自由（如果你愿意，你可以对此项服务收取一定的费用）；保证你能收到源程序或者在你需要时能够得到它；保证你能修改软件或将它的一部分用于新的自由软件；而且还保证你知道你能做这些事情。

为了保护你的权利，我们需要作出规定：禁止任何人不承认你的权利，或者要求你放弃这些权利。如果你修改了库或者发布了库的副本，这些规定就转化为你的责任。

例如，如果你发布这样一个软件的副本，不管是收费的还是免费的，你必须将你具有的一切权利给予你的接受者。你必须保证他们能收到或得到源程序。并且你必须将这些条款给他们看，使他们知道他们有这样的权利。

我们采取两项措施来保护你的权利：（1）给软件以版权保护。（2）给你提供许可证。它给你复制，发布和修改这些软件的法律许可。

同样，为了保护发布者，我们需要清楚地让每个人明白，自由库软件没有担保。如果由于其他某个人修改了库，并继续加以传播。我们需要它的接受者明白：他们所得到的并不是原来的版本。任何由其他人引入的任何问题，不应损害原作者的声誉。

最后，任何自由软件不断受到软件专利的威胁。我们希望避免这样的风险，自由软件的再发布者以个人名义获得专利许可证。从而在事实上将软件变为私有。为防止这一点，我们必须明确：任何专利必须以允许每个人自由使用为前提，否则就不准许有专利。

以下是关于复制、分发和修改的详细条款和条件。复制、分发和修改的条款和条件

0. This License applies to any program or other work which contains a notice placed by the copyright holder saying it may be distributed under the terms of this General Public License. The "Program", below, refers to any such program or work, and a "work based on the Program" means either the Program or any derivative work under copyright law: that is to say, a work containing the Program or a portion of it, either verbatim or with modifications and/or translated into another language. (Hereinafter, translation is included without limitation in the term "modification".) Each licensee is addressed as "you".

Activities other than copying, distribution and modification are not covered by this License; they are outside its scope. The act of running the Program is not restricted, and the output from the Program is covered only if its contents constitute a work based on the Program (independent of having been made by running the Program). Whether that is true depends on what the Program does.

1. You may copy and distribute verbatim copies of the Program's source code as you receive it, in any medium, provided that you conspicuously and appropriately publish on each copy an appropriate copyright notice and disclaimer of warranty; keep intact all the notices that refer to this License and to the absence of any warranty; and give any other recipients of the Program a copy of this License along with the Program.

You may charge a fee for the physical act of transferring a copy, and you may at your option offer warranty protection in exchange for a fee.

2. You may modify your copy or copies of the Program or any portion of it, thus forming a work based on the Program, and copy and distribute such modifications or work under the terms of Section 1 above, provided that you also meet all of these conditions:

- a) You must cause the modified files to carry prominent notices stating that you changed the files and the date of any change.
- b) You must cause any work that you distribute or publish, that in whole or in part contains or is derived from the Program or any part thereof, to be licensed as a whole at no charge to all third parties under the terms of this License.
- c) If the modified program normally reads commands interactively when run, you must cause it, when started running for such interactive use in the most ordinary way, to print or display an announcement including an appropriate copyright notice and a notice that there is no warranty (or else, saying that you provide a warranty) and that users may redistribute the program under these conditions, and telling the user how to view a copy of this License. (Exception: if the Program itself is interactive but does not normally print such an announcement, your work based on the Program is not required to print an announcement.)

These requirements apply to the modified work as a whole. If identifiable sections of that work are not derived from the Program, and can be reasonably considered independent and separate works in themselves, then this License, and its terms, do not apply to those sections when you distribute them as separate works. But when you distribute the same sections as part of a whole which is a work based on the Program, the distribution of the whole must be on the terms of this License, whose permissions for other licensees extend to the entire whole, and thus to each and every part regardless of who wrote it.

Thus, it is not the intent of this section to claim rights or contest your rights to work written entirely by you; rather, the intent is to exercise the right to control the distribution of derivative or collective works based on the Program.

In addition, mere aggregation of another work not based on the Program with the Program (or with a work based on the Program) on a volume of a storage or distribution medium does not bring the other work under the scope of this License.

3. You may copy and distribute the Program (or a work based on it, under Section 2) in object code or executable form under the terms of Sections 1 and 2 above provided that you also do one of the following:
 - a) Accompany it with the complete corresponding machine-readable source code, which must be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange; or,
 - b) Accompany it with a written offer, valid for at least three years, to give any third party, for a charge no more than your cost of physically performing source distribution, a complete machine-readable copy of the corresponding source code, to be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange; or,

- c) Accompany it with the information you received as to the offer to distribute corresponding source code. (This alternative is allowed only for noncommercial distribution and only if you received the program in object code or executable form with such an offer, in accord with Subsection b above.)

The source code for a work means the preferred form of the work for making modifications to it. For an executable work, complete source code means all the source code for all modules it contains, plus any associated interface definition files, plus the scripts used to control compilation and installation of the executable. However, as a special exception, the source code distributed need not include anything that is normally distributed (in either source or binary form) with the major components (compiler, kernel, and so on) of the operating system on which the executable runs, unless that component itself accompanies the executable.

If distribution of executable or object code is made by offering access to copy from a designated place, then offering equivalent access to copy the source code from the same place counts as distribution of the source code, even though third parties are not compelled to copy the source along with the object code.

4. You may not copy, modify, sublicense, or distribute the Program except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense or distribute the Program is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.
5. You are not required to accept this License, since you have not signed it. However, nothing else grants you permission to modify or distribute the Program or its derivative works. These actions are prohibited by law if you do not accept this License. Therefore, by modifying or distributing the Program (or any work based on the Program), you indicate your acceptance of this License to do so, and all its terms and conditions for copying, distributing or modifying the Program or works based on it.
6. Each time you redistribute the Program (or any work based on the Program), the recipient automatically receives a license from the original licensor to copy, distribute or modify the Program subject to these terms and conditions. You may not impose any further restrictions on the recipients' exercise of the rights granted herein. You are not responsible for enforcing compliance by third parties to this License.
7. If, as a consequence of a court judgment or allegation of patent infringement or for any other reason (not limited to patent issues), conditions are imposed on you (whether by court order, agreement or otherwise) that contradict the conditions of this License, they do not excuse you from the conditions of this License. If you cannot distribute so as to satisfy simultaneously your obligations under this License and any other pertinent obligations, then as a consequence you may not distribute the Program at all. For example, if a patent license would not permit royalty-free redistribution of the Program by all those who receive copies directly or indirectly through you, then the only way you could satisfy both it and this License would be to refrain entirely from distribution of the Program.

If any portion of this section is held invalid or unenforceable under any particular circumstance, the balance of the section is intended to apply and the section as a whole is intended to apply in other circumstances.

It is not the purpose of this section to induce you to infringe any patents or other property right claims or to contest validity of any such claims; this section has the sole purpose of protecting the integrity of the free software distribution system, which is implemented by public license practices. Many people have made generous contributions to the wide range of software distributed through that system in reliance on consistent application of that system; it is up to the author/donor to decide if he or she is willing to distribute software through any other system and a licensee cannot impose that choice.

This section is intended to make thoroughly clear what is believed to be a consequence of the rest of this License.

8. If the distribution and/or use of the Program is restricted in certain countries either by patents or by copyrighted interfaces, the original copyright holder who places the Program under this License may add an explicit geographical distribution limitation excluding those countries, so that distribution is permitted only in or among countries not thus excluded. In such case, this License incorporates the limitation as if written in the body of this License.
9. The Free Software Foundation may publish revised and/or new versions of the General Public License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns.

Each version is given a distinguishing version number. If the Program specifies a version number of this License which applies to it and "any later version", you have the option of following the terms and conditions either of that version or of any later version published by the Free Software Foundation. If the Program does not specify a version number of this License, you may choose any version ever published by the Free Software Foundation.

10. If you wish to incorporate parts of the Program into other free programs whose distribution conditions are different, write to the author to ask for permission. For software which is copyrighted by the Free Software Foundation, write to the Free Software Foundation; we sometimes make exceptions for this. Our decision will be guided by the two goals of preserving the free status of all derivatives of our free software and of promoting the sharing and reuse of software generally.

NO WARRANTY

11. BECAUSE THE PROGRAM IS LICENSED FREE OF CHARGE, THERE IS NO WARRANTY FOR THE PROGRAM, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE PROGRAM "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE PROGRAM IS WITH YOU. SHOULD THE PROGRAM PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.
12. IN NO EVENT UNLESS REQUIRED BY APPLICABLE LAW OR AGREED TO IN WRITING WILL ANY COPYRIGHT HOLDER, OR ANY OTHER PARTY WHO MAY MODIFY AND/OR REDISTRIBUTE THE PROGRAM AS PERMITTED ABOVE, BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE PROGRAM (INCLUDING BUT NOT LIMITED TO LOSS OF DATA OR DATA BEING RENDERED INACCURATE OR LOSSES SUSTAINED BY YOU OR THIRD PARTIES OR A FAILURE OF THE PROGRAM TO OPERATE WITH ANY OTHER PROGRAMS), EVEN IF SUCH HOLDER OR OTHER PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

QGIS Qt exception for GPL

In addition, as a special exception, the QGIS Development Team gives permission to link the code of this program with the Qt library, including but not limited to the following versions (both free and commercial): Qt/Non-commercial Windows, Qt/Windows, Qt/X11, Qt/Mac, and Qt/Embedded (or with modified versions of Qt that use the same license as Qt), and distribute linked combinations including the two. You must obey the GNU General Public License in all respects for all of the code used other than Qt. If you modify this file, you may extend this exception to your version of the file, but you are not obligated to do so. If you do not wish to do so, delete this exception statement from your version.

28.2 Appendix B: GNU Free Documentation License

Version 1.3, 3 November 2008

Copyright 2000, 2001, 2002, 2007, 2008 Free Software Foundation, Inc

<http://fsf.org/>

任何人都可以复制和分发本授权文件的完整副本，但不允许更改。

Preamble

The purpose of this License is to make a manual, textbook, or other functional and useful document "free" in the sense of freedom: to assure everyone the effective freedom to copy and redistribute it, with or without modifying it, either commercially or noncommercially. Secondly, this License preserves for the author and publisher a way to get credit for their work, while not being considered responsible for modifications made by others.

This License is a kind of "copyleft", which means that derivative works of the document must themselves be free in the same sense. It complements the GNU General Public License, which is a copyleft license designed for free software.

We have designed this License in order to use it for manuals for free software, because free software needs free documentation: a free program should come with manuals providing the same freedoms that the software does. But this License is not limited to software manuals; it can be used for any textual work, regardless of subject matter or whether it is published as a printed book. We recommend this License principally for works whose purpose is instruction or reference.

1. APPLICABILITY AND DEFINITIONS

This License applies to any manual or other work, in any medium, that contains a notice placed by the copyright holder saying it can be distributed under the terms of this License. Such a notice grants a world-wide, royalty-free license, unlimited in duration, to use that work under the conditions stated herein. The **Document**, below, refers to any such manual or work. Any member of the public is a licensee, and is addressed as **"you"**. You accept the license if you copy, modify or distribute the work in a way requiring permission under copyright law.

A **"Modified Version"** of the Document means any work containing the Document or a portion of it, either copied verbatim, or with modifications and/or translated into another language.

A **"Secondary Section"** is a named appendix or a front-matter section of the Document that deals exclusively with the relationship of the publishers or authors of the Document to the Document's overall subject (or to related matters) and contains nothing that could fall directly within that overall subject. (Thus, if the Document is in part a textbook of mathematics, a Secondary Section may not explain any mathematics.) The relationship could be a matter of historical connection with the subject or with related matters, or of legal, commercial, philosophical, ethical or political position regarding them.

The **"Invariant Sections"** are certain Secondary Sections whose titles are designated, as being those of Invariant Sections, in the notice that says that the Document is released under this License. If a section does not fit the above definition of Secondary then it is not allowed to be designated as Invariant. The Document may contain zero Invariant Sections. If the Document does not identify any Invariant Sections then there are none.

The **"Cover Texts"** are certain short passages of text that are listed, as Front-Cover Texts or Back-Cover Texts, in the notice that says that the Document is released under this License. A Front-Cover Text may be at most 5 words, and a Back-Cover Text may be at most 25 words.

A **"Transparent"** copy of the Document means a machine-readable copy, represented in a format whose specification is available to the general public, that is suitable for revising the document straightforwardly with generic text editors or (for images composed of pixels) generic paint programs or (for drawings) some widely available drawing editor, and that is suitable for input to text formatters or for automatic translation to a variety of formats suitable for input to text formatters. A copy made in an otherwise Transparent file format whose markup, or absence of markup, has been arranged to thwart or discourage subsequent modification by readers is not Transparent. An image format is not Transparent if used for any substantial amount of text. A copy that is not "Transparent" is called **Opaque**.

Examples of suitable formats for Transparent copies include plain ASCII without markup, Texinfo input format, LaTeX input format, SGML or XML using a publicly available DTD, and standard-conforming simple HTML, PostScript or PDF designed for human modification. Examples of transparent image formats include PNG, XCF and JPG. Opaque formats include proprietary formats that can be read and edited only by proprietary word processors, SGML or XML for which the DTD and/or processing tools are not generally available, and the machine-generated HTML, PostScript or PDF produced by some word processors for output purposes only.

The **"Title Page"** means, for a printed book, the title page itself, plus such following pages as are needed to hold, legibly, the material this License requires to appear in the title page. For works in formats which do not have any title page as such, "Title Page" means the text near the most prominent appearance of the work's title, preceding the beginning of the body of the text.

The **"publisher"** means any person or entity that distributes copies of the Document to the public.

A section **"Entitled XYZ"** means a named subunit of the Document whose title either is precisely XYZ or contains XYZ in parentheses following text that translates XYZ in another language. (Here XYZ stands for a specific section name mentioned below, such as **"Acknowledgements"**, **"Dedications"**, **"Endorsements"**, or **"History"**.) To **"Preserve the Title"** of such a section when you modify the Document means that it remains a section "Entitled XYZ" according to this definition.

The Document may include Warranty Disclaimers next to the notice which states that this License applies to the Document. These Warranty Disclaimers are considered to be included by reference in this License, but only as

regards disclaiming warranties: any other implication that these Warranty Disclaimers may have is void and has no effect on the meaning of this License.

2. VERBATIM COPYING

You may copy and distribute the Document in any medium, either commercially or noncommercially, provided that this License, the copyright notices, and the license notice saying this License applies to the Document are reproduced in all copies, and that you add no other conditions whatsoever to those of this License. You may not use technical measures to obstruct or control the reading or further copying of the copies you make or distribute. However, you may accept compensation in exchange for copies. If you distribute a large enough number of copies you must also follow the conditions in section 3.

You may also lend copies, under the same conditions stated above, and you may publicly display copies.

3. COPYING IN QUANTITY

If you publish printed copies (or copies in media that commonly have printed covers) of the Document, numbering more than 100, and the Document's license notice requires Cover Texts, you must enclose the copies in covers that carry, clearly and legibly, all these Cover Texts: Front-Cover Texts on the front cover, and Back-Cover Texts on the back cover. Both covers must also clearly and legibly identify you as the publisher of these copies. The front cover must present the full title with all words of the title equally prominent and visible. You may add other material on the covers in addition. Copying with changes limited to the covers, as long as they preserve the title of the Document and satisfy these conditions, can be treated as verbatim copying in other respects.

If the required texts for either cover are too voluminous to fit legibly, you should put the first ones listed (as many as fit reasonably) on the actual cover, and continue the rest onto adjacent pages.

If you publish or distribute Opaque copies of the Document numbering more than 100, you must either include a machine-readable Transparent copy along with each Opaque copy, or state in or with each Opaque copy a computer-network location from which the general network-using public has access to download using public-standard network protocols a complete Transparent copy of the Document, free of added material. If you use the latter option, you must take reasonably prudent steps, when you begin distribution of Opaque copies in quantity, to ensure that this Transparent copy will remain thus accessible at the stated location until at least one year after the last time you distribute an Opaque copy (directly or through your agents or retailers) of that edition to the public.

It is requested, but not required, that you contact the authors of the Document well before redistributing any large number of copies, to give them a chance to provide you with an updated version of the Document.

4. MODIFICATIONS

You may copy and distribute a Modified Version of the Document under the conditions of sections 2 and 3 above, provided that you release the Modified Version under precisely this License, with the Modified Version filling the role of the Document, thus licensing distribution and modification of the Modified Version to whoever possesses a copy of it. In addition, you must do these things in the Modified Version:

- A. Use in the Title Page (and on the covers, if any) a title distinct from that of the Document, and from those of previous versions (which should, if there were any, be listed in the History section of the Document). You may use the same title as a previous version if the original publisher of that version gives permission.
- B. List on the Title Page, as authors, one or more persons or entities responsible for authorship of the modifications in the Modified Version, together with at least five of the principal authors of the Document (all of its principal authors, if it has fewer than five), unless they release you from this requirement.
- C. State on the Title page the name of the publisher of the Modified Version, as the publisher.
- D. Preserve all the copyright notices of the Document.
- E. Add an appropriate copyright notice for your modifications adjacent to the other copyright notices.
- F. Include, immediately after the copyright notices, a license notice giving the public permission to use the Modified Version under the terms of this License, in the form shown in the Addendum below.
- G. Preserve in that license notice the full lists of Invariant Sections and required Cover Texts given in the Document's license notice.
- H. Include an unaltered copy of this License.

- I. Preserve the section Entitled "History", Preserve its Title, and add to it an item stating at least the title, year, new authors, and publisher of the Modified Version as given on the Title Page. If there is no section Entitled "History" in the Document, create one stating the title, year, authors, and publisher of the Document as given on its Title Page, then add an item describing the Modified Version as stated in the previous sentence.
- J. Preserve the network location, if any, given in the Document for public access to a Transparent copy of the Document, and likewise the network locations given in the Document for previous versions it was based on. These may be placed in the "History" section. You may omit a network location for a work that was published at least four years before the Document itself, or if the original publisher of the version it refers to gives permission.
- K. For any section Entitled "Acknowledgements" or "Dedications", Preserve the Title of the section, and preserve in the section all the substance and tone of each of the contributor acknowledgements and/or dedications given therein.
- L. Preserve all the Invariant Sections of the Document, unaltered in their text and in their titles. Section numbers or the equivalent are not considered part of the section titles.
- M. Delete any section Entitled "Endorsements". Such a section may not be included in the Modified Version.
- N. Do not retitle any existing section to be Entitled "Endorsements" or to conflict in title with any Invariant Section.
- O. Preserve any Warranty Disclaimers.

If the Modified Version includes new front-matter sections or appendices that qualify as Secondary Sections and contain no material copied from the Document, you may at your option designate some or all of these sections as invariant. To do this, add their titles to the list of Invariant Sections in the Modified Version's license notice. These titles must be distinct from any other section titles.

You may add a section Entitled "Endorsements", provided it contains nothing but endorsements of your Modified Version by various parties---for example, statements of peer review or that the text has been approved by an organization as the authoritative definition of a standard.

You may add a passage of up to five words as a Front-Cover Text, and a passage of up to 25 words as a Back-Cover Text, to the end of the list of Cover Texts in the Modified Version. Only one passage of Front-Cover Text and one of Back-Cover Text may be added by (or through arrangements made by) any one entity. If the Document already includes a cover text for the same cover, previously added by you or by arrangement made by the same entity you are acting on behalf of, you may not add another; but you may replace the old one, on explicit permission from the previous publisher that added the old one.

The author(s) and publisher(s) of the Document do not by this License give permission to use their names for publicity for or to assert or imply endorsement of any Modified Version.

5. COMBINING DOCUMENTS

You may combine the Document with other documents released under this License, under the terms defined in section 4 above for modified versions, provided that you include in the combination all of the Invariant Sections of all of the original documents, unmodified, and list them all as Invariant Sections of your combined work in its license notice, and that you preserve all their Warranty Disclaimers.

The combined work need only contain one copy of this License, and multiple identical Invariant Sections may be replaced with a single copy. If there are multiple Invariant Sections with the same name but different contents, make the title of each such section unique by adding at the end of it, in parentheses, the name of the original author or publisher of that section if known, or else a unique number. Make the same adjustment to the section titles in the list of Invariant Sections in the license notice of the combined work.

In the combination, you must combine any sections Entitled "History" in the various original documents, forming one section Entitled "History"; likewise combine any sections Entitled "Acknowledgements", and any sections Entitled "Dedications". You must delete all sections Entitled "Endorsements".

6. COLLECTIONS OF DOCUMENTS

You may make a collection consisting of the Document and other documents released under this License, and replace the individual copies of this License in the various documents with a single copy that is included in the collection, provided that you follow the rules of this License for verbatim copying of each of the documents in all other respects.

You may extract a single document from such a collection, and distribute it individually under this License, provided you insert a copy of this License into the extracted document, and follow this License in all other respects regarding verbatim copying of that document.

7. AGGREGATION WITH INDEPENDENT WORKS

A compilation of the Document or its derivatives with other separate and independent documents or works, in or on a volume of a storage or distribution medium, is called an "aggregate" if the copyright resulting from the compilation is not used to limit the legal rights of the compilation's users beyond what the individual works permit. When the Document is included in an aggregate, this License does not apply to the other works in the aggregate which are not themselves derivative works of the Document.

If the Cover Text requirement of section 3 is applicable to these copies of the Document, then if the Document is less than one half of the entire aggregate, the Document's Cover Texts may be placed on covers that bracket the Document within the aggregate, or the electronic equivalent of covers if the Document is in electronic form. Otherwise they must appear on printed covers that bracket the whole aggregate.

8. TRANSLATION

Translation is considered a kind of modification, so you may distribute translations of the Document under the terms of section 4. Replacing Invariant Sections with translations requires special permission from their copyright holders, but you may include translations of some or all Invariant Sections in addition to the original versions of these Invariant Sections. You may include a translation of this License, and all the license notices in the Document, and any Warranty Disclaimers, provided that you also include the original English version of this License and the original versions of those notices and disclaimers. In case of a disagreement between the translation and the original version of this License or a notice or disclaimer, the original version will prevail.

If a section in the Document is Entitled "Acknowledgements", "Dedications", or "History", the requirement (section 4) to Preserve its Title (section 1) will typically require changing the actual title.

9. TERMINATION

You may not copy, modify, sublicense, or distribute the Document except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense, or distribute it is void, and will automatically terminate your rights under this License.

However, if you cease all violation of this License, then your license from a particular copyright holder is reinstated (a) provisionally, unless and until the copyright holder explicitly and finally terminates your license, and (b) permanently, if the copyright holder fails to notify you of the violation by some reasonable means prior to 60 days after the cessation.

Moreover, your license from a particular copyright holder is reinstated permanently if the copyright holder notifies you of the violation by some reasonable means, this is the first time you have received notice of violation of this License (for any work) from that copyright holder, and you cure the violation prior to 30 days after your receipt of the notice.

Termination of your rights under this section does not terminate the licenses of parties who have received copies or rights from you under this License. If your rights have been terminated and not permanently reinstated, receipt of a copy of some or all of the same material does not give you any rights to use it.

10. FUTURE REVISIONS OF THIS LICENSE

The Free Software Foundation may publish new, revised versions of the GNU Free Documentation License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns. See <http://www.gnu.org/copyleft/>.

Each version of the License is given a distinguishing version number. If the Document specifies that a particular numbered version of this License "or any later version" applies to it, you have the option of following the terms and conditions either of that specified version or of any later version that has been published (not as a draft) by the Free Software Foundation. If the Document does not specify a version number of this License, you may choose any version ever published (not as a draft) by the Free Software Foundation. If the Document specifies that a proxy can decide which future versions of this License can be used, that proxy's public statement of acceptance of a version permanently authorizes you to choose that version for the Document.

11. RELICENSING

”Massive Multiauthor Collaboration Site” (or ”MMC Site”) means any World Wide Web server that publishes copyrightable works and also provides prominent facilities for anybody to edit those works. A public wiki that anybody can edit is an example of such a server. A ”Massive Multiauthor Collaboration” (or ”MMC”) contained in the site means any set of copyrightable works thus published on the MMC site.

”CC-BY-SA” means the Creative Commons Attribution-Share Alike 3.0 license published by Creative Commons Corporation, a not-for-profit corporation with a principal place of business in San Francisco, California, as well as future copyleft versions of that license published by that same organization.

”Incorporate” means to publish or republish a Document, in whole or in part, as part of another Document.

An MMC is ”eligible for relicensing” if it is licensed under this License, and if all works that were first published under this License somewhere other than this MMC, and subsequently incorporated in whole or in part into the MMC, (1) had no cover texts or invariant sections, and (2) were thus incorporated prior to November 1, 2008.

The operator of an MMC Site may republish an MMC contained in the site under CC-BY-SA on the same site at any time before August 1, 2009, provided the MMC is eligible for relicensing.

ADDENDUM: How to use this License for your documents

To use this License in a document you have written, include a copy of the License in the document and put the following copyright and license notices just after the title page:

```
Copyright © YEAR YOUR NAME.
Permission is granted to copy, distribute and/or modify this document
under the terms of the GNU Free Documentation License, Version 1.3
or any later version published by the Free Software Foundation;
with no Invariant Sections, no Front-Cover Texts, and no Back-Cover Texts.
A copy of the license is included in the section entitled "GNU
Free Documentation License".
```

If you have Invariant Sections, Front-Cover Texts and Back-Cover Texts, replace the ”with ... Texts.” line with this:

```
with the Invariant Sections being LIST THEIR TITLES, with the
Front-Cover Texts being LIST, and with the Back-Cover Texts being LIST.
```

If you have Invariant Sections without Cover Texts, or some other combination of the three, merge those two alternatives to suit the situation.

If your document contains nontrivial examples of program code, we recommend releasing these examples in parallel under your choice of free software license, such as the GNU General Public License, to permit their use in free software.

28.3 Appendix C: QGIS File Formats

28.3.1 QGS/QGZ - QGIS 工程文件格式

QGS 是一种用于保存 QGIS 工程的 XML 格式，QGZ 是一种包含 QGS 文件和 QGD 文件的压缩格式 (zip) 文件，QGD 是一个关联 sqlite 数据库的文件，里面包含了 QGIS 工程的辅助数据，如果一个工程没有辅助数据，QGD 文件将会为空。

QGIS 文件包含了任何存储 QGIS 工程需要的信息，包括

- 工程标题
- 工程参考坐标系
- 图层树
- 捕捉设置
- 关系
- 地图画布范围

- 工程模型
- 图例
- mapview docks (2D and 3D)
- the layers with links to the underlying datasets (data sources) and other layer properties including extent, SRS, joins, styles, renderer, blend mode, opacity and more.
- 工程属性

The figures below show the top level tags in a QGS file and the expanded `ProjectLayers` tag.

```
-<qgis version="3.4.13-Madeira" projectname="">
  <homePath path=""/>
  <title/>
  <autotransaction active="0"/>
  <evaluateDefaultValues active="0"/>
  <trust active="0"/>
  +<projectCrs></projectCrs>
  +<layer-tree-group></layer-tree-group>
  +<snapping-settings tolerance="12" unit="1" enabled="0" type="1" mode="2" intersection-snapping="0">
    </snapping-settings>
    <relations/>
  -<mapcanvas name="theMapCanvas" annotationsVisible="1">
    <units>meters</units>
    +<extent></extent>
    <rotation>0</rotation>
    +<destinationSRS></destinationSRS>
    <renderMapTile>0</renderMapTile>
  </mapcanvas>
  <projectModels/>
  +<legend updateDrawingOrder="true"></legend>
  <mapViewDocks/>
  <mapViewDocks3D/>
  +<projectLayers></projectLayers>
  +<layerOrder></layerOrder>
  +<properties></properties>
  <visibility-presets/>
  <transformContext/>
  +<projectMetadata></projectMetadata>
  <Annotations/>
  <Layouts/>
</qgis>
```

图 28.1: The top level tags in a QGS file


```

--<projectlayers>
--<maplayer styleCategories="AllStyleCategories" readOnly="0" autoRefreshTime="0" autoRefreshEnabled="0" refreshOnNotifyEnabled="0" maxScale="0"
geometry="Polygon" labelsEnabled="0" type="vector" simplifyDrawingHints="1" hasScaleBasedVisibilityFlag="0" simplifyDrawingTol="1"
simplifyMaxScale="1" minScale="1e+8" simplifyAlgorithm="0" simplifyLocal="1" refreshOnNotifyMessage="">
+<extent></extent>
<id>watersheds_b62efa19_8809_4406_b6ec_2951ac4c94c5</id>
--<datasource>
./QGIS-Training-Data-2.0/exercise_data/processing/generalize/watersheds.shp
</datasource>
+<keywordList></keywordList>
<layername>watersheds</layername>
+<srs></srs>
+<resourceMetadata></resourceMetadata>
<provider encoding="UTF-8">ogr</provider>
<vectorJoins/>
<layerDependencies/>
<dataDependencies/>
<legend type="default-vector"/>
<expressionFields/>
+<map-layer-style-manager current="default"></map-layer-style-manager>
<auxiliaryLayer/>
+<flags></flags>
+<renderer-v2 symbolLevels="0" enableOrderby="0" type="singleSymbol" forceRaster="0"></renderer-v2>
+<customProperties></customProperties>
<blendMode>0</blendMode>
<featureBlendMode>0</featureBlendMode>
<layerOpacity>1</layerOpacity>
+<SingleCategoryDiagramRenderer diagramType="Histogram" attributeLegend="1"></SingleCategoryDiagramRenderer>
+<DiagramLayerSettings priority="0" linePlacementFlags="18" dist="0" showAll="1" placement="1" obstacle="0" zIndex="0"></DiagramLayerSettings>
+<geometryOptions removeDuplicateNodes="0" geometryPrecision="0"></geometryOptions>
+<fieldConfiguration></fieldConfiguration>
+<aliases></aliases>
<excludeAttributesWMS/>
<excludeAttributesWFS/>
+<defaults></defaults>
+<constraints></constraints>
+<constraintExpressions></constraintExpressions>
<expressionFields/>
+<attributeactions></attributeactions>
+<attributableconfig actionWidgetStyle="dropDown" sortExpression="" sortOrder="0"></attributableconfig>
+<conditionalstyles></conditionalstyles>
<editform tolerant="1"/>
<editforminit/>
<editforminitcodesource>0</editforminitcodesource>
<editforminitfilepath/>
+<editforminitcode></editforminitcode>
<featformsuppress>0</featformsuppress>
<editorlayout>generatedlayout</editorlayout>
+<editable></editable>
+<labelOnTop></labelOnTop>
<widgets/>
<previewExpression>ID</previewExpression>
<mapTip/>
</maplayer>
</projectlayers>

```

图 28.2: The expanded top level ProjectLayers tag of a QGS file

28.3.2 QLR - QGIS 图层定义文件定义

图层定义文件（QLR）是一种包含图层数据源的指针和次图层的 QGIS 样式信息的 XML 文件

The use case for this file is simple: To have a single file for opening a data source and bringing in all the related style information. QLR files also allow you to mask the underlying datasource in an easy to open file.

An example of QLR usage is for opening MS SQL layers. Rather than having to go to the MS SQL connection dialog, connect, select, load and finally style, you can simply add a .qlr file that points to the correct MS SQL layer with all the necessary style included.

未来，一个 qlr 文件将会支持引用多个图层

```

-<qlr>
+<layer-tree-group name="" checked="Qt::Checked" expanded="1"></layer-tree-group>
-<maplayers>
  -<maplayer autoRefreshEnabled="0" labelsEnabled="0" autoRefreshTime="0" readOnly="0" refreshOnNotifyMessage=""
    geometry="Line" simplifyDrawingTol="1" simplifyMaxScale="1" styleCategories="AllStyleCategories" simplifyDrawingHints="1"
    maxScale="0" simplifyLocal="1" hasScaleBasedVisibilityFlag="0" type="vector" refreshOnNotifyEnabled="0" minScale="1e+8"
    simplifyAlgorithm="0">
    +<extent></extent>
      <id>inputnew_6740bb2e_0441_4af5_8dcf_305c5c4d8ca7</id>
    +<datasource></datasource>
    +<keywordList></keywordList>
      <layername>inputnew</layername>
    +<srs></srs>
    +<resourceMetadata></resourceMetadata>
      <provider encoding="UTF-8">ogr</provider>
      <vectorjoins/>
      <layerDependencies/>
      <dataDependencies/>
      <legend type="default-vector"/>
      <expressionfields/>
    +<map-layer-style-manager current="default"></map-layer-style-manager>
      <auxiliaryLayer/>
    +<flags></flags>
    +<renderer-v2 enableOrderby="0" type="singleSymbol" forceraster="0" symbollevels="0"></renderer-v2>
    +<customproperties></customproperties>
      <blendMode>0</blendMode>
      <featureBlendMode>0</featureBlendMode>
      <layerOpacity>1</layerOpacity>
    +<geometryOptions removeDuplicateNodes="0" geometryPrecision="0"></geometryOptions>
    +<fieldConfiguration></fieldConfiguration>
    +<aliases></aliases>
      <excludeAttributesWMS/>
      <excludeAttributesWFS/>
    +<defaults></defaults>
    +<constraints></constraints>
    +<constraintExpressions></constraintExpressions>
      <expressionfields/>
    +<attributeactions></attributeactions>
    +<attributableconfig sortExpression="" actionWidgetStyle="dropDown" sortOrder="0"></attributableconfig>
    +<conditionalstyles></conditionalstyles>
      <editform tolerant="1">../src/qgisplugins/qgisbostaskdepplugin/data</editform>
      <editforminit/>
      <editforminitcodesource>0</editforminitcodesource>
      <editforminitfilepath/>
      <editforminitcode></editforminitcode>
      <featformsuppress>0</featformsuppress>
      <editorlayout>generatedlayout</editorlayout>
      <editable/>
      <labelOnTop/>
      <widgets/>
      <previewExpression>"FID"</previewExpression>
      <mapTip/>
    </maplayer>
  </maplayers>
</qlr>

```

图 28.3: The top level tags of a QLR file

28.3.3 QML - QGIS 样式文件格式

QML 是一种用于存储图层样式的 XML 文件格式

QML 文件包含了 GIS 能够处理的所有信息 (用于渲染几何要素的信息), 包括符号定义、尺寸、旋转、标注、不透明度、混合模式等

The figure below shows the top level tags of a QML file (with only `renderer_v2` and its `symbol` tag expanded).

```

- <qgis version="3.4.13-Madeira" styleCategories="AllStyleCategories" readOnly="0" maxScale="0"
labelsEnabled="0" simplifyDrawingHints="1" hasScaleBasedVisibilityFlag="0" simplifyDrawingTol="1"
simplifyMaxScale="1" minScale="1e+8" simplifyAlgorithm="0" simplifyLocal="1">
+ <flags></flags>
- <renderer-v2 symbollevels="0" enableorderby="0" type="singleSymbol" forceraster="0">
  - <symbols>
    + <symbol clip_to_extent="1" name="0" alpha="1" type="fill" force_rhr="0"></symbol>
    </symbols>
    <rotation/>
    <sizescale/>
  </renderer-v2>
+ <customproperties></customproperties>
  <blendMode>0</blendMode>
  <featureBlendMode>0</featureBlendMode>
  <layerOpacity>1</layerOpacity>
+ <SingleCategoryDiagramRenderer diagramType="Histogram" attributeLegend="1">
</SingleCategoryDiagramRenderer>
+ <DiagramLayerSettings priority="0" linePlacementFlags="18" dist="0" showAll="1" placement="1"
obstacle="0" zIndex="0">
</DiagramLayerSettings>
+ <geometryOptions removeDuplicateNodes="0" geometryPrecision="0"></geometryOptions>
+ <fieldConfiguration></fieldConfiguration>
+ <aliases></aliases>
  <excludeAttributesWMS/>
  <excludeAttributesWFS/>
+ <defaults></defaults>
+ <constraints></constraints>
+ <constraintExpressions></constraintExpressions>
  <expressionfields/>
+ <attributeactions></attributeactions>
+ <attributetableconfig actionWidgetStyle="dropDown" sortExpression="" sortOrder="0">
</attributetableconfig>
+ <conditionalstyles></conditionalstyles>
  <editform tolerant="1"/>
  <editforminit/>
  <editforminitcodesource>0</editforminitcodesource>
  <editforminitfilepath/>
+ <editforminitcode></editforminitcode>
  <featformsuppress>0</featformsuppress>
  <editorlayout>generatedlayout</editorlayout>
+ <editable></editable>
+ <labelOnTop></labelOnTop>
  <widgets/>
  <previewExpression>ID</previewExpression>
  <mapTip/>
  <layerGeometryType>2</layerGeometryType>
</qgis>

```

图 28.4: The top level tags of a QML file (only the renderer_v2 tag with its symbol tag is expanded)

28.4 Appendix D: QGIS R script syntax

Contributed by Matteo Ghetta - funded by Scuola Superiore Sant'Anna

Writing R scripts in Processing is a bit tricky because of the special syntax.

A Processing R script starts with defining its **Inputs** and **Outputs**, each preceded with double hash characters (##).

Before the inputs, the group to place the algorithm in can be specified. If the group already exists, the algorithm will be added to it, if not, the group will be created. In the example below, the name of the group is *My group*:

```
##My Group=group
```

28.4.1 Inputs

All input data and parameters have to be specified. There are several types of inputs:

- vector: `##Layer = vector`
- vector field: `##F = Field Layer` (where *Layer* is the name of an input vector layer the field belongs to)
- raster: `##r = raster`
- table: `##t = table`
- number: `##Num = number`
- string: `##Str = string`
- boolean: `##Bol = boolean`
- elements in a dropdown menu. The items must be separated with semicolons `;; ##type=selection point;lines;point+lines`

28.4.2 Outputs

As for the inputs, each output has to be defined at the beginning of the script:

- vector: `##output= output vector`
- raster: `##output= output raster`
- table: `##output= output table`
- plots: `##output_plots_to_html (##showplots in earlier versions)`
- To show R output in the *Result Viewer*, put `>` in front of the command whose output you would like to show.

28.4.3 Syntax Summary for QGIS R scripts

A number of input and output parameter types are offered.

Input parameter types

Parameter	Syntax example	Returning objects
vector	Layer = vector	sf object (or SpatialDataFrame object, if <code>##load_vector_using_rgdal</code> is specified)
vector point	Layer = vector point	sf object (or SpatialDataFrame object, if <code>##load_vector_using_rgdal</code> is specified)
vector line	Layer = vector line	sf object (or SpatialDataFrame object, if <code>##load_vector_using_rgdal</code> is specified)
vector poly-gon	Layer = vector polygon	sf object (or SpatialPolygonsDataFrame object, if <code>##load_vector_using_rgdal</code> is used)
multiple vector	Layer = multiple vector	sf object (or SpatialDataFrame objects if <code>##load_vector_using_rgdal</code> is specified)
table	Layer = table	dataframe conversion from csv, default object of <code>read.csv</code> function
field	Field = Field Layer	name of the Field selected, e.g. "Area"
raster	Layer = raster	RasterBrick object, default object of <code>raster</code> package
multiple raster	Layer = multiple raster	RasterBrick objects, default object of <code>raster</code> package
number	N = number	integer or floating number chosen
string	S = string	string added in the box
longstring	LS = longstring	string added in the box, could be longer then the normal string
selection	S = selection first;second;third	string of the selected item chosen in the dropdown menu
crs	C = crs	string of the resulting CRS chosen, in the format: "EPSG:4326"
extent	E = extent	Extent object of the <code>raster</code> package, you can extract values as <code>E@xmin</code>
point	P = point	when clicked on the map, you have the coordinates of the point
file	F = file	path of the file chosen, e.g. <code>"/home/matteo/file.txt"</code>
folder	F = folder	path of the folder chosen, e.g. <code>"/home/matteo/Downloads"</code>

A parameter can be **OPTIONAL**, meaning that it can be ignored.

In order to set an input as optional, you add the string `optional` **before** the input, e.g:

```
##Layer = vector
##Field1 = Field Layer
##Field2 = optional Field Layer
```

Output parameter types

Parameter	Syntax example
vector	Output = output vector
raster	Output = output raster
table	Output = output table
file	Output = output file

注解: You can save plots as `png` from the *Processing Result Viewer*, or you can choose to save the plot directly from the algorithm interface.

Script body

The script body follows R syntax and the **Log** panel can help you if there is something wrong with your script.

Remember that you have to load all additional libraries in the script:

```
library(sp)
```

28.4.4 Examples

Example with vector output

Let's take an algorithm from the online collection that creates random points from the extent of an input layer:

```
##Point pattern analysis=group
##Layer=vector polygon
##Size=number 10
##Output=output vector
library(sp)
spatpoly = as(Layer, "Spatial")
pts=spsample(spatpoly, Size, type="random")
spdf=SpatialPointsDataFrame(pts, as.data.frame(pts))
Output=st_as_sf(spdf)
```

Explanation (per line in the script):

1. Point pattern analysis is the group of the algorithm
2. Layer is the input **vector** layer
3. Size is a **numerical** parameter with a default value of 10
4. Output is the **vector** layer that will be created by the algorithm
5. `library(sp)` loads the **sp** library
6. `spatpoly = as(Layer, "Spatial")` translate to an sp object
7. Call the `spsample` function of the sp library and run it using the input defined above (Layer and Size)
8. Create a *SpatialPointsDataFrame* object using the `SpatialPointsDataFrame` function
9. Create the output vector layer using the `st_as_sf` function

That's it! Just run the algorithm with a vector layer you have in the QGIS Legend, choose the number of random point. The resulting layer will be added to your map.

Example with raster output

The following script will perform basic ordinary kriging to create a raster map of interpolated values from a specified field of the input point vector layer by using the `autoKrige` function of the `automap` R package. It will first calculate the kriging model and then create a raster. The raster is created with the `raster` function of the `raster` R package:

```
##Basic statistics=group
##Layer=vector point
##Field=Field Layer
##Output=output raster
##load_vector_using_rgdal
require("automap")
require("sp")
require("raster")
```

(下页继续)

(续上页)

```

table=as.data.frame(Layer)
coordinates(table)= ~coords.x1+coords.x2
c = Layer[[Field]]
kriging_result = autoKrige(c~1, table)
prediction = raster(kriging_result$krige_output)
Output<-prediction

```

By using `##load_vector_using_rgdal`, the input vector layer will be made available as a `SpatialDataFrame` objects, so we avoid having to translate it from an `sf` object.

Example with table output

Let's edit the `Summary Statistics` algorithm so that the output is a table file (csv).

The script body is the following:

```

##Basic statistics=group
##Layer=vector
##Field=Field Layer
##Stat=Output table
Summary_statistics<-data.frame(rbind(
  sum(Layer[[Field]]),
  length(Layer[[Field]]),
  length(unique(Layer[[Field]])),
  min(Layer[[Field]]),
  max(Layer[[Field]]),
  max(Layer[[Field]])-min(Layer[[Field]]),
  mean(Layer[[Field]]),
  median(Layer[[Field]]),
  sd(Layer[[Field]]),
  row.names=c("Sum:", "Count:", "Unique values:", "Minimum value:", "Maximum value:",
  ↪"Range:", "Mean value:", "Median value:", "Standard deviation:"))
colnames(Summary_statistics)<-c(Field)
Stat<-Summary_statistics

```

The third line specifies the **Vector Field** in input and the fourth line tells the algorithm that the output should be a table.

The last line will take the `Stat` object created in the script and convert it into a `csv` table.

Example with console output

We can use the previous example and instead of creating a table, print the result in the **Result Viewer**:

```

##Basic statistics=group
##Layer=vector
##Field=Field Layer
Summary_statistics<-data.frame(rbind(
  sum(Layer[[Field]]),
  length(Layer[[Field]]),
  length(unique(Layer[[Field]])),
  min(Layer[[Field]]),
  max(Layer[[Field]]),
  max(Layer[[Field]])-min(Layer[[Field]]),
  mean(Layer[[Field]]),
  median(Layer[[Field]]),
  sd(Layer[[Field]]), row.names=c("Sum:", "Count:", "Unique values:", "Minimum value:",
  ↪"Maximum value:", "Range:", "Mean value:", "Median value:", "Standard deviation:"))
colnames(Summary_statistics)<-c(Field)
>Summary_statistics

```

The script is exactly the same as the one above except for two edits:

1. no output specified (the fourth line has been removed)
2. the last line begins with `>`, telling Processing to make the object available through the result viewer

Example with plot

To create plots, you have to use the `##output_plots_to_html` parameter as in the following script:

```
##Basic statistics=group
##Layer=vector
##Field=Field Layer
##output_plots_to_html
####output_plots_to_html
qqnorm(Layer[[Field]])
qqline(Layer[[Field]])
```

The script uses a field (`Field`) of a vector layer (`Layer`) as input, and creates a *QQ Plot* (to test the normality of the distribution).

The plot is automatically added to the Processing *Result Viewer*.

Literature and Web References

GDAL-SOFTWARE-SUITE. Geospatial data abstraction library. <https://gdal.org>, 2013.

GRASS-PROJECT. Geographic resource analysis support system. <https://grass.osgeo.org>, 2013.

NETELER, M., AND MITASOVA, H. Open source gis: A grass gis approach, 2008.

OGR-SOFTWARE-SUITE. Geospatial data abstraction library. <https://gdal.org>, 2013.

OPEN-GEOSPATIAL-CONSORTIUM. Web map service (1.1.1) implementation specification. <https://portal.opengeospatial.org>, 2002.

OPEN-GEOSPATIAL-CONSORTIUM. Web map service (1.3.0) implementation specification. <https://portal.opengeospatial.org>, 2004.

POSTGIS-PROJECT. Spatial support for postgresql. <http://postgis.refractions.net/>, 2013.