QGIS Interface for SWAT+: QSWAT+

Version 1.3.2
December 2019

Prepared by: Yihun Dile, R. Srinivasan and Chris George

This version describes QSWAT+ version 1.3.2, which needs SWAT+ Editor version 1.3.0 or later.

QSWAT+
Step by Step Setup for the Robit Watershed, Lake Tana basin Ethiopia

Contents

1. Notes for QSWAT users 5
2. Environment and Tools Required 6
   2.1 Source code 6
3. Installation 6
   3.1 Windows 6
   3.2 Linux 7
   3.3 Directories installed 7
4. Structure and Location of Source Data 8
5. Structure and Location of Output Data 8
6. Setup for Robit Watershed, Lake Tana Basin 9
7. Watershed Delineation 10
   7.1 Loading the DEM 10
      7.1.1 Creating channels and streams 12
      7.1.2 Inlets and outlets 13
      7.1.3 Creating landscape units 15
      7.1.4 Merging subbasins 18
   7.2 Create HRUs 21
      7.2.1 Landuse and soil maps 21
      7.2.2 Defining soil properties 23
      7.2.3 Merging short channels 29
      7.2.4 Splitting and exempting landuses 30
      7.2.5 Ponds and reservoirs 32
      7.2.6 Creating HRUs 33
   7.3 Setting up the Robit example in the SWAT Editor 37
      7.3.1 Weather data 37
      7.3.2 Simulation period 37
      7.3.3 Print options 37
      7.3.4 Run SWAT+ 38
      7.3.5 Save the outputs 38
8. Output Visualisation with QSWAT+ 38
   8.1 Running visualise 39

8.2 Visualisation period 39
8.3 Static visualisation 39
8.4 Combining results 43
8.5 Combining tables 44
8.6 Printing results 45
8.7 Animation 46
8.8 Plotting 49
8.8.1 Collecting plot data 49
8.8.2 How compatible do different scenarios need to be? 53
8.8.3 Including observed data 54
9. Post processing 55
9.1 Environmental flows 55
9.1.1 Qq 55
9.1.2 dQp 55
9.1.3 Qb 58
10. Using grid models 60
10.1 Creating a grid 60
10.2 Using an existing grid 64
11. Adding lakes 68
11.1 Polygon shapefiles 68
11.2 Dissolving shapes 69
11.3 Delineation with lakes 71
11.3.1 Channel splitting 71
11.3.2 Problems 76
11.3.3 Fixing problems 79
11.3.4 Short outlet channels 81
11.3.5 Multiple outlets 81
11.3.6 Water routing 82
11.4 Files involved 82
11.5 Lakes in grid models 83
11.6 Lakes in predefined watersheds 85
11.6.1 Predefined grid models 85
11.6.2 Predefined non-grid models 86
12. Parameter settings 87
12.1 SWATPlus directory 88
12.2 MPI bin directory 88
12.3 Channel widths and depths 89

12.4 Slope and length multipliers 89
12.5 Font point size 90
12.6 Stream burn-in depth 90
12.7 Upslope HRU drain % 90
13. Exporting SQLite database tables 91
14. Appendix I - Adding your own data 91
14.1 Adding weather generator data 91
14.2 Preparing daily climatic data for QSWAT+ 93
14.3 Preparing landuse and soil tables 94
15. Appendix II – preparing global DEM data for QSWAT+ 96
15.1 Introduction 96
15.2 Obtaining SRTM DEM data 96
15.2.1 90m data 97
15.2.2 30m data 97
15.3 Clipping the grids 97
15.4 Merging the grids 98
15.5 Reprojecting the grid 99
15.6 Masking a DEM 103
15.7 Large DEMs 105
16. Appendix III: Using a Predefined Watershed and Channel Network 106
16.1 Non-grid models 107
16.2 Grid models 110
17. Appendix IV: Installing and Using MPI 111
17.1 Installing MS-MPI 112
17.2 Configuring QSWAT+ to use MPI 113
17.3 Blocking network access 113
18. Appendix V: Conversion from ArcSWAT to QSWAT+ 114
18.1 Purpose 114
18.2 Instructions 115
18.3 Landuse and soil data 117
18.4 Other data 118
18.4.1 Weather generator data 118
18.4.2 Observed weather gauges 119
18.4.3 Reference data 119
18.4.4 Inlet and point source data 119
18.4.5 Dates 119
19. Appendix VI: Conversion from QSWAT to QSWAT+ 120

19.1 Purpose 120
19.2 Instructions 120
19.2.1 Prerequisites 120
19.2.2 Conversion 120
19.3 Data 121
19.3.1 Landuse data 122
19.3.2 Soil data 122
19.3.3 Weather generator data 122
19.3.4 Observed weather gauges 122
19.3.5 Reference data 122
19.3.6 Inlet and point source data 122
19.3.7 Dates 123

1. Notes for QSWAT users

For experienced QSWAT users we first highlight the things that are new or different in QSWAT+.

- **Environment**: You can use 64-bit tools (Installation, Environment and Tools Required) and QSWAT+ uses SQLite instead of Microsoft Access databases. Linux is not yet supported, but will be soon. You build and run SWAT+ models using only free, open-source software.
- **QGIS is version 3, not 2:** this means we are using Python 3 instead of Python 2. The differences for the user are not very great, and you should soon get used to them. QGIS 3 can be safely installed alongside an existing QGIS 2 installation.
- **Streams and channels**: Subbasins are formed as before by stream drainage areas, but subbasins can contain multiple channels (Creating channels and streams). HRUs drain to channels, and so are more precisely positioned than in QSWAT..
- **Landscape units**: Areas draining to channels can be divided into *floodplain* and *upslope* areas, each forming a *landscape unit* (Creating landscape units). HRUs belong to landscape units.
- **Short channel merge:** Subbasins can be merged as in QSWAT, but in addition channels can be merged within subbasins (Merging short channels) with the aim of avoiding small HRUs and landscape units.
- **Lakes:** Lakes may be added, defined by shapefile. (Adding lakes)
- **Ponds and reservoirs:** Areas with landuse WATR can be used to form ponds or reservoirs, if the user marks a pond or reservoir, or sets a threshold for when WATR becomes a reservoir (Ponds and reservoirs). WATR areas that do not become ponds

or reservoirs, or included in lakes, are changed to landuse WETN; there are no longer HRUs with landuse WATR.

- **Grid models**: As well as models based on delineated subbasins, grid models can be created and used (Using grid models).
- **Parameters:** A wide range of parameters can be set using the parameters form (Parameter settings).
- **Environmental flows**: A range of environmental flows can be generated (Environmental flows).

2. Environment and Tools Required

- Microsoft Windows (any version, as far as we are aware) or Linux (probably any recent version, but only tested on Ubuntu).
- Text editor (e.g. NotePad in Windows, gedit in Linux, Emacs in either) that enables you to read and edit ASCII text files.
- QGIS. The instruction in this document assume you have a 64 bit machine, and if so you should use the 64 bit versions of QGIS, QSWAT+ and SWAT+. But if you have a 32 bit machine you can use 32 bit versions of them all. You will not be able to run such large projects as are possible with 64 bits.
- A tool that allows for viewing and editing SQLite databases. A good choice in Windows is the free, open source SQLiteStudio which can be downloaded from https://sqlitestudio.pl. In Linux you can also use SQLiteStudio, or you may find SQLiteBrowser available in your distribution.

2.1 Source code

The source code for QSWAT+ is available from
https://bitbucket.org/ChrisWGeorge/qswatplus3

3. Installation

3.1 Windows

- Install QGIS by running **QGIS-OSGeo4W-3.4.11-1-Setup-x86_64.exe**. It can be downloaded from the QGIS download page, where you should select the **Long term release repository, and** the **64 bit standalone installer.** Use the default folder *C:/Program Files/QGIS 3.4* as the installation folder.
  Note that you can use a later version than this provided it is QGIS 3, not QGIS 2 which is incompatible with QSWAT+.
- Install QSWAT+ by running **swatplustools-installer**, version 1.2.2 or later. It can be downloaded from SWAT+ page of the SWAT website. You may be informed that *Windows protected your PC*, and you need to click *More info* and then *Run anyway* to proceed with the installation. You then need to accept the license. This is a combination installer that provides QSWAT+, the SWAT+ Editor, and some other

materials. You can choose which components you need (for the first install QSWAT+, the SWAT+ Editor, probably the global weather generator, and the SSURGO/STATSGO soil data if you will be modelling anywhere in the USA), and you also need to make a couple of choices:

- The SWAT+ editor, together with the SWAT+ program and other materials are by default installed in *C:/SWAT/SWATPlus*, but the user doing the installation has the option of changing this directory. If you choose to change from this default, you will need to use the QSWAT+ Parameters form the first time you run QSWAT+ to set the *SWATPlus* directory to whatever path you chose. The setting will then be retained by QSWAT+ and the same one used in future, unless you change it again.
- QSWAT+ is a plugin for QGIS. It can be installed for *all users*, or just the user running the installer. In the second case the plugin is placed within the user's personal files, and will only appear as a plugin when the same user starts QGIS. This does not require admin privileges and is the natural choice if you are installing on your personal laptop, or on a machine where you are the primary user. In the *all users* case, the plugin is placed within the QGIS installation's files. This will require admin privileges. It also has the consequence that if you upgrade QGIS, QSWAT+ will need to be reinstalled. The installer may also have to ask you to browse to where QGIS is installed, if it can't find it in the usual places.
  Note that it is possible to install QSWAT+ individually for more than one user, if this is more convenient than using the *all users* option, simply by each user running the QSWAT+ component of the swatplustools-installer.

3.2 Linux

TODO

3.3 Directories installed

The directories created in the *SWATPlus* directory are:

- *SWATPlusEditor* containing the editor and the SWAT+ program
- *Databases* containing a template project database, a reference database, a template project file, the global weather generator database (if selected) and the SSURGO/STATSGO soil database (if selected)
- *Documents* containing this manual (if selected in the QSWAT+ installer)
- *ExampleDatasets* containing the *Robit* example used in this manual
- *TauDEM5Bin* (if selected in the QSWAT+ installer) containing the TauDEM programs used in watershed delineation. You don't need this if you have QSWAT installed.
- *Tools* containing
  - Windows only*: ConvertFromArc* containing a converter from ArcSWAT projects to QSWAT+ projects.
  - *SWATGraph* (if selected in the QSWAT+ installer) containing *runSWATGraph.bat* (Windows) or *runSWATGrap.sh* (Linux) This gives stand-

alone access to the SWATGraph program. You don't need this on Windows if you have QSWAT installed.

Users can join the QWSWAT+ user group at https://groups.google.com/d/forum/qswatplus. There are also user groups for the SWAT+ Editor and SWAT+ model.

4. Structure and Location of Source Data

Before starting QGIS, users should prepare their tabular and spatial data according to SWAT+ requirements. Users can refer to **Appendix I** on preparing tabular data (e.g. preparing soil and climatic data). All the spatial data (DEM, land use and soil) should be projected into the same projected coordinate system. Users can refer to **Appendix II** for preparing global data for QSWAT+. The example files used in this manual are available in the *SWATPlus* directory in a directory *ExampleDatasets/Robit.* This directory contains:

- A DEM in directory *DEM*
- A stream reaches file to burn in to the DEM, in directory *RobitStreams*
- An outlet file in directory *MainOutlet*
- Landuse and soil maps in directories *Landuse* and *Soil*
- Climate data in directory *ClimateRobit*
- Some csv files for making database tables, and
- An observed data file *observedFlow.csv* in directory *Observed.*

Users are recommended to save the location *C:/SWAT/SWATPlus/ExampleDatasets/Robit* to your *Favorites* or *Quick access* list in Windows Explorer. There are frequent references to this folder as files and maps are loaded, and it is tedious to access if its path is long. We shall refer to this folder as *Robit*

5. Structure and Location of Output Data

We will be establishing a project called *Demo1* in a folder *C:/QSWATPlus_Projects/Robit*. This will create:

1. A folder *C:/QSWATPlus_Projects/Robit/Demo1* called the *project folder*.
2. A file *C:/QSWATPlus_Projects/Robit/Demo1/Demo1.qgs* called the *project file*. This contains details of the current state of QGIS, and the project settings we have chosen. If we later want to reopen the project *Demo1.qgs* will be the file we look for..
3. A folder *C:/QSWATPlus_Projects/Robit/Demo1/Scenarios/Default/TxtInOut* that will contain all the SWAT+ input and output files.
4. A folder *C:/ QSWATPlus_Projects/Robit/Demo1/Scenarios/Default/Results* that may be used by the SWAT+ Editor and is used to hold results files from visualisation.
5. If we choose to save a SWAT run as *Sim1*, say, then a folder *C:/ QSWATPlus_Projects/Robit/Demo1/Scenarios/Sim1* will be created (a copy of *C:/ QSWATPlus_Projects/Robit/Demo1/Scenarios/Default*, including its sub-folders).
6. A folder *C:/ QSWATPlus_Projects/Robit/Demo1/Watershed* with several sub-folders:
    6.1. *Rasters* to contain the rasters used, divided into 3 further subfolders

6.1.1. *DEM* which will contain the local copy of your DEM, plus all the files generated from it like flow direction, streams and subbasins as rasters

6.1.2. *Landscape* which will contain, if you create floodplains, floodplain rasters and the rasters created to generate them.

6.1.3. *Landuse* to contain the local copy of your landuse raster

6.1.4. *Soil* to contain the local copy of your soil raster

6.2. *Shapes* to contain all the shapefiles used or generated

6.3. *Text* to contain reports.

6. Setup for Robit Watershed, Lake Tana Basin

Start **QGIS 3** and select **Plugins** menu -> **Manage and Install Plugins** and find **QSWATPlus3_64**. Click its checkbox to install, which takes a few seconds. The QSWAT+ icon will appear in the toolbar.

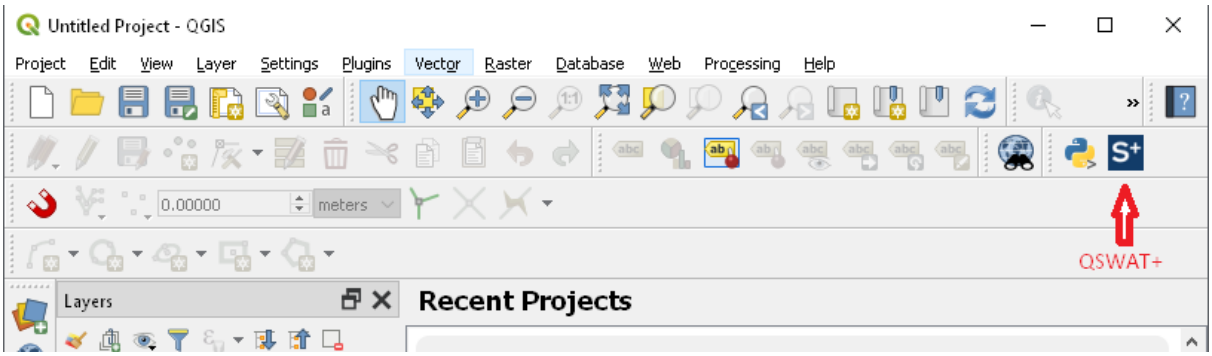


Figure 1: QGIS 3 with the QSWAT+ plugin loaded

Start **QSWAT+** by clicking the **QSWAT+** icon. The main QSWAT+ interface will be displayed..

Click the box **New Project**. A browser will be displayed requesting a parent directory for the new project. Browse to a folder such as *C:/QSWATPlus_Projects*. You will probably need to create the folder *Robit*, so in the left panel right click on *QSWAT_Projects* to open a menu, select *New -> Folder* and type in *Robit* to overwrite the default name of *New Folder*, and hit *Enter.* *Robit* will appear in the right panel. Select it so it appears in the text box below and click **Select Folder.**

A new form opens asking for the project name, which must start with a letter. Type in *Demo1* and click **OK.**

The interface now presents a step-by-step configuration to be followed in order to prepare the SWAT+ simulation, starting with Step 1 (Figure 2). Note that it displays the path of the project folder at the foot. The **QSWAT+ Parameters** and **Export Table** buttons are described later in sections 12 and 13.

Figure 2: QSWAT+ interface at step 1

At this stage the project database is created as *Demo1.sqlite* in the project folder, and a copy of the SWAT reference database *swatplus_datasets.sqlite* is also created there.

At this stage all your maps should be prepared in an equal area projection (probably, but not necessarily, UTM[1]). All the maps should be in the same projection coordinate system. Refer to Appendix **II** – preparing global DEM data for QSWAT+ for advice on preparing DEMs.

7. Watershed Delineation

7.1 Loading the DEM

To start automatic watershed delineation click the **Delineate Watershed** button. Then click the button in the **Select DEM** group.

[1] UTM is not an equal area projection, but is close to being so, and sufficiently close for SWAT+ to use on most watersheds.

Browse to the *Robit/DEM/srtm_30m* folder and open the file *hdr.adf*. This DEM is an ESRI grid (or raster) and consists of a number of files. The file called *hdr.adf* is the one to choose for such a grid. QSWAT+ accepts a very wide range of raster file formats. The most common are probably GeoTiff files, which normally have a *.tif* suffix and come as single files.

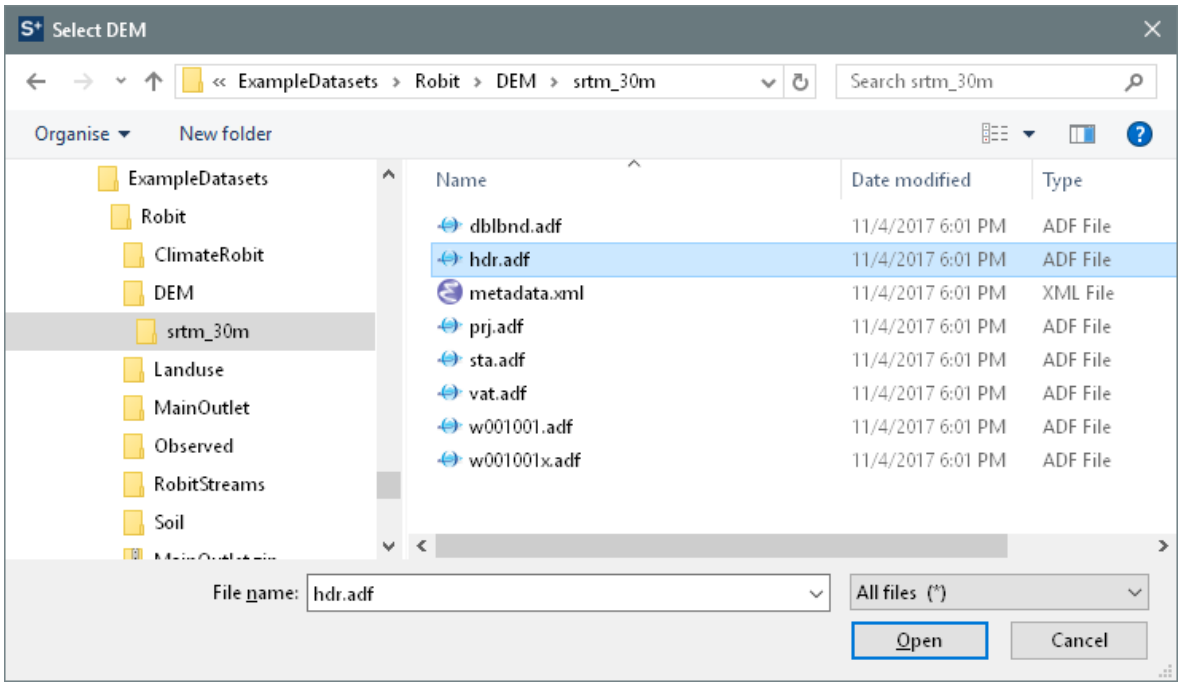


Figure 3: Selecting the DEM

The name of the elevation map grid will be displayed in the *DEM* text box below the *Watershed* dialogue box. The selected DEM will be copied to the projects *Source* folder and, if necessary, converted to GeoTiff format (.tif). If your DEM's units are not metres you can select the **DEM properties** tab and change them. This tab also shows the DEM cell size, cell area, projection system and extent in degrees.
Users can burn in using an existing stream network using the option **Burn in existing stream network**: use a file with the extension **.shp* (Figure 4). Check the *Burn in existing stream network* option and select the file *Robit/RobitStreams/robReach.shp*.

The delineation form allows you to opt for a **predefined watershed and stream network** by selecting the **Use existing watershed** tab. If you are interested in using a predefined watershed and stream network please refer to Appendix III: Using a Predefined Watershed and Channel Network).

We will not use MPI, which enables multiprocessing during watershed delineation (see Appendix IV: Installing and Using MPI) and so the *Number of processes* spinbox should be set to 0.

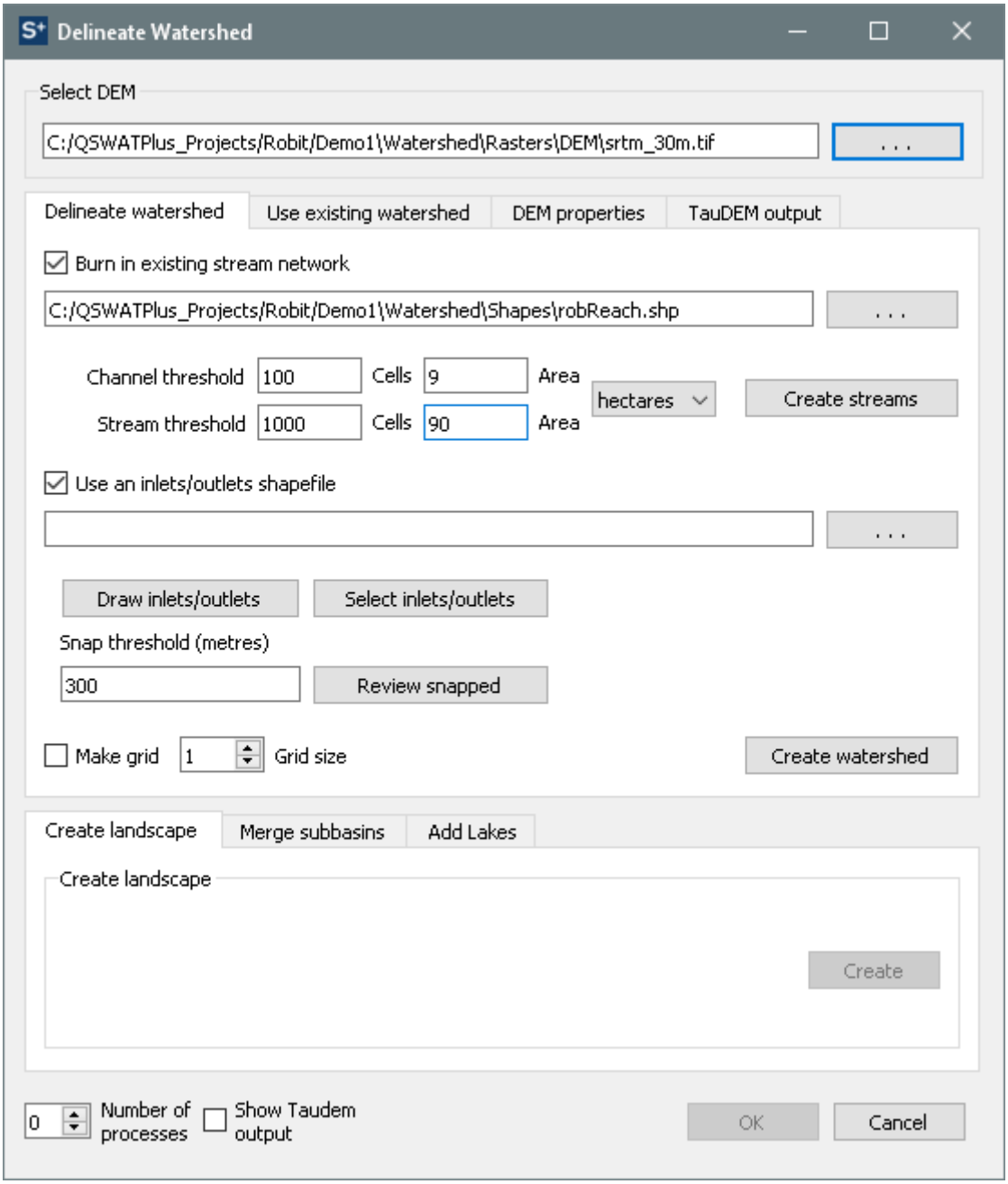


Figure 4: Creating the stream network

7.1.1 Creating channels and streams

The threshold sizes for creating channels and streams should be set next. They can be set by area, in various units such as sq km or hectares, or by number of cells. If necessary, change the threshold method to use hectares, change the area for channels to 9 and for streams to 90, and press **Create Streams**: the number of cells will be adjusted to the corresponding value (100 and 1000 respectively). The threshold is the number of cells (or area) required to form a channel or a stream: a cell will be made part of a channel or stream if it has at least the threshold number of cells draining into it. Stream reaches are sections of the stream network between significant points, where significant points are stream sources, monitoring points like water gauges, stream junctions, inlets, and watershed outlets. A subbasin is an area

draining into a stream reach. Subbasins are the main divisions of the watershed, and typically the entities whose SWAT outputs we will want to generate and study. Channels are finer divisions and extensions of stream reaches, and allow us to place more precisely the components of the watershed which will form the SWAT+ model: landscape units, reservoirs, ponds, point sources, and hydrological response units. This step creates streams (Figure 5). It also loads the stream burn-in shapefile if one was selected.

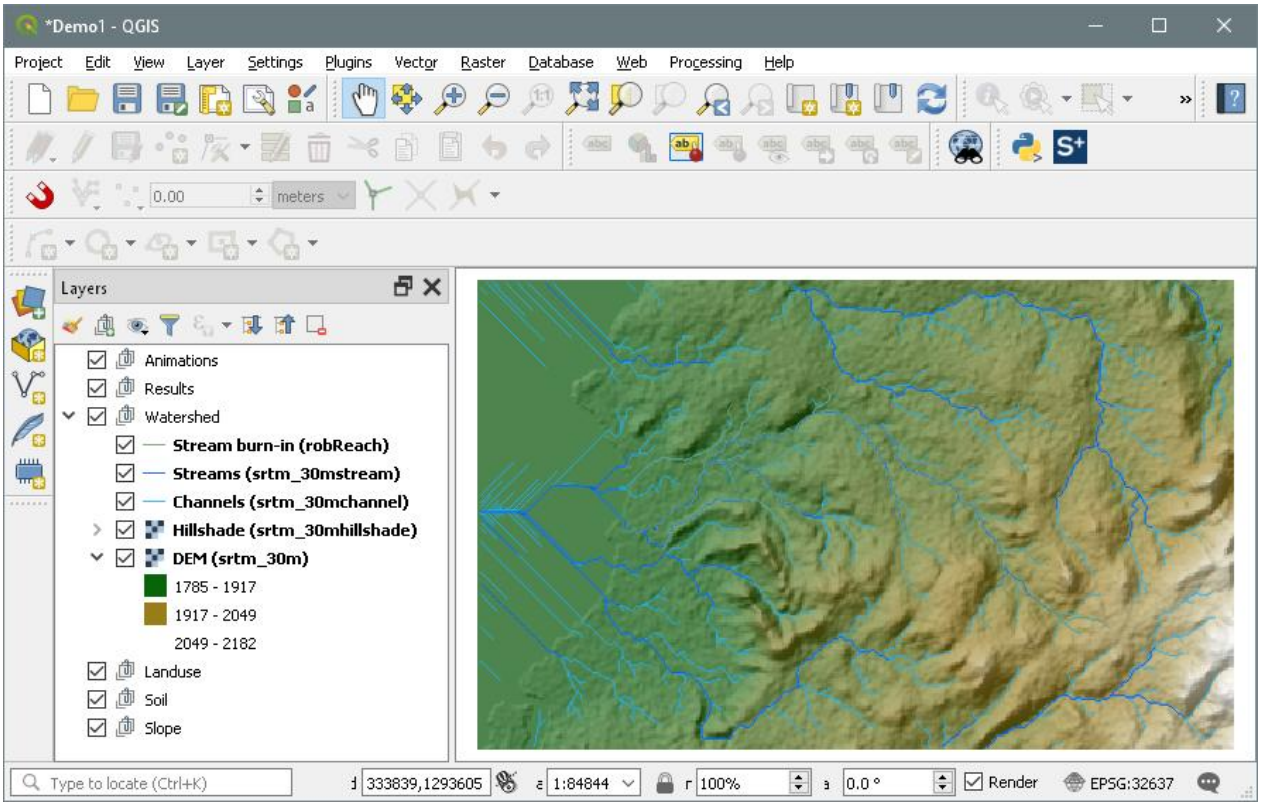


Figure 5: Stream and channel networks displayed

7.1.2 Inlets and outlets

An inlets/outlets file containing just a main outlet is provided for this example. Check the **Use an inlets/outlets shapefile** option, and browse to *Robit/MainOutlet/MainOutlet.shp*. The outlet is added (Figure 6).

Users can add outlets, reservoirs, ponds, inlets, and point sources by selecting the **Draw Inlets/Outlets** option. This first asks if you wish to add to the existing inlets/outlets file, or create a new one. Having made this choice, select the type of point to add, and click on the map to place it at the appropriate place. Inlets and outlets need to be placed on the stream network, so you will need to make streams but not channels visible first. You may need to zoom in to place them precisely. Point sources, ponds and reservoirs need to be placed on the channel network. Only points within the snap threshold (default 300 metres) of a stream (or channel) reach will be counted as points. Click **OK** to confirm and exit, and **Cancel** to remove points and exit. If you choose to draw a new file, *drawoutlets.shp* is created in the *Watershed/Shapes* folder, which may be used again. See Appendix III: Using a Predefined

Watershed and Channel Network for details of the fields needed in an inlets/outlets file. We do not add any additional outlets, ponds, reservoirs, inlets, or point sources in this example.

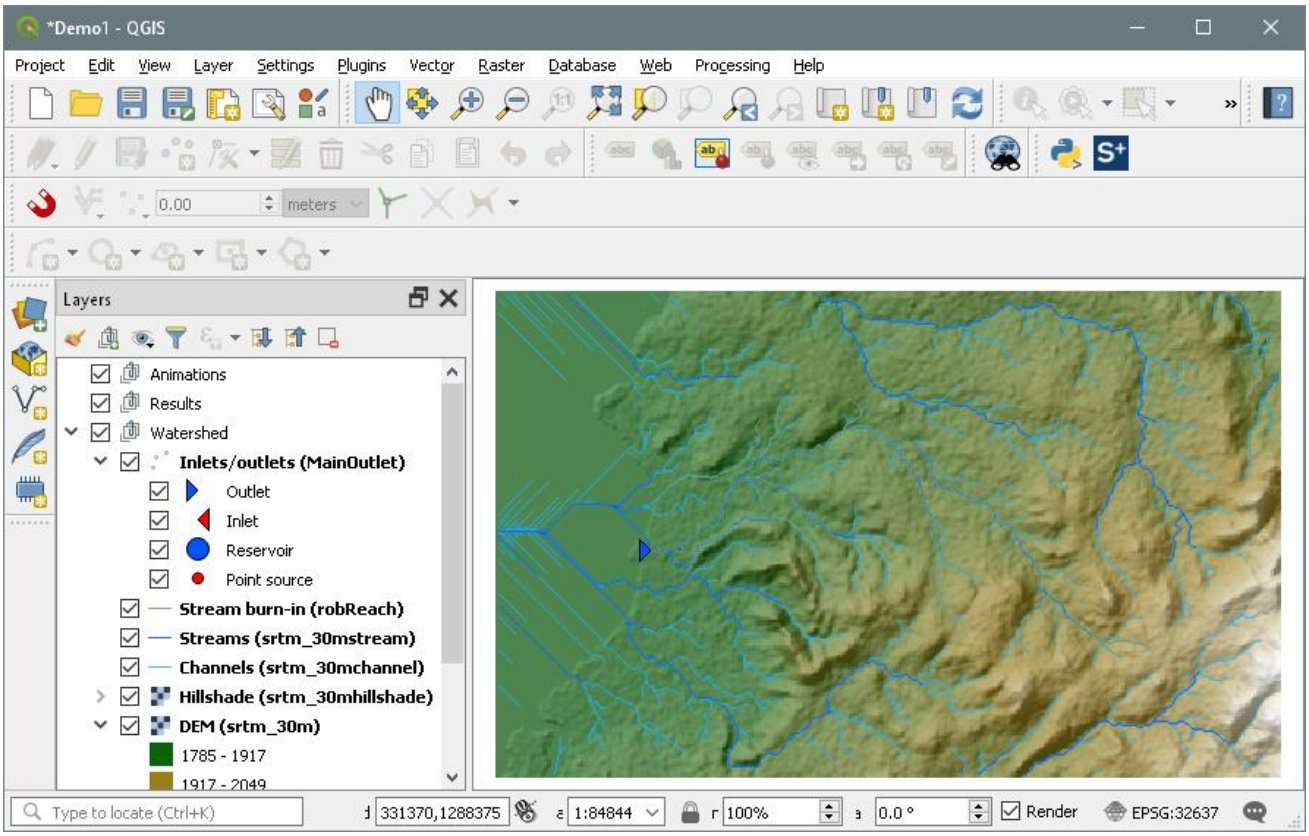


Figure 6: Outlet added

If you have several points in your inlets/outlets file you can choose to use just a selection of them. To do this choose the **Select Inlets/Outlets** option. Hold Ctrl and select the points by dragging the mouse to make a small rectangle around them. Selected points will turn yellow, and a count will be shown at the bottom left of the main window. You only need to use this option if you want to use only a subset of outlets. The default is to use all the points in the outlets file or those drawn.

You will be told how many points were snapped successfully. If some were not snapped (or to check in advance before creating the watershed), you can click **Review snapped** which shows the snapped inlets/outlets, i.e. those within the defined threshold distance.

Clicking **Create Watershed** will create the watershed (Figure 7). Check that the delineation form shows the right number of points have been snapped (in our case 1) and none have failed. Note that in Figure 7 we have zoomed in a little to focus better on the watershed. This is done by selecting the subbasins layer in the **Layers** panel and then clicking the **Zoom to layer** button (in the top tool bar, just under **Processing**).

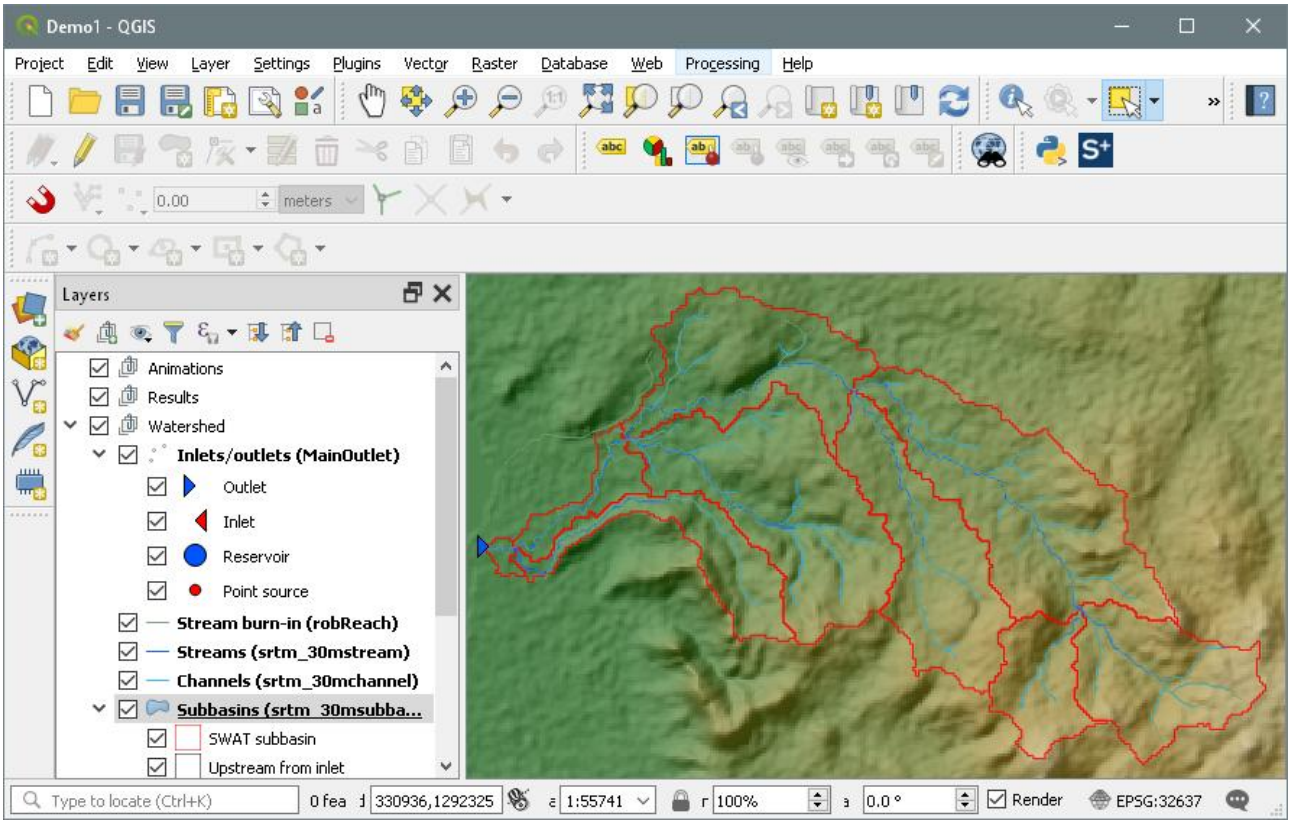


Figure 7: Delineated watershed

In QSWAT+ it is possible to divide subbasins into *landscape units*. Two landscape units are possible, *floodplain* and *upslope*. If we define landscape units, when we later come to define hydrological response units (HRUs) these will be defined within each landscape unit, and if we also choose to select only some HRUs, then each landscape unit will be treated separately, so that each landscape unit contains at least one HRU. The choice of defining landscape units applies to the whole watershed. It is possible that very small subbasins might fall wholly within the floodplain, but in general each subbasin will have two landscape units.

7.1.3 Creating landscape units

To define landscape units, we click the **Create landscape** tab and then the **Create** button. This brings up the **Landscape analysis** form (Figure 8).

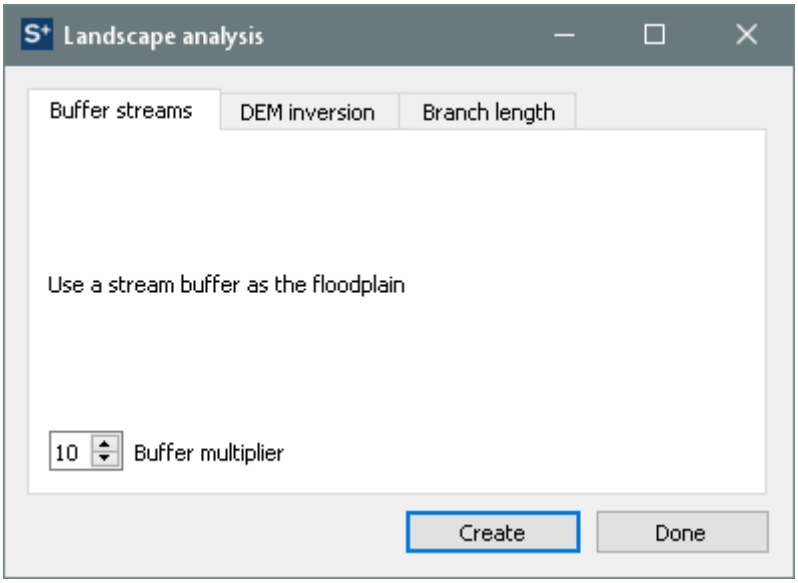


Figure 8: Creating landscape units by buffering streams

There are three possible methods for creating landscape units. The first is to make the floodplain simply as a buffer drawn around the stream reaches. There is no terrain analysis involved. The width of the floodplain on each side of the stream is set as a multiple, default 10, of the stream width. Main stream width varies with the area draining to it, so the floodplain width will also vary, increasing downstream. This method is most likely to be used when the terrain is mostly flat, when other methods can give poor results.

The second method is called DEM inversion. It is calculated by negating all the DEM elevations and recalculating flow directions, and calculating how much water would then flow into each point, the *flow accumulation*. Points where the flow accumulation exceeds a threshold, the *ridge threshold*, are designated as ridges. This exactly mirrors the method of calculating where streams will occur, and the default ridge threshold is set the same as the stream threshold. A slope position for each point is then calculated. This is the ratio of the drop from each point to where its water flow meets a stream, to the total drop from its ridge point (where its inverse flow meets a ridge) to the same stream point. If this ratio is less than the slope position threshold its landscape position is floodplain, else it is upslope. 0.1 is a typical value to use for slope position, and is the default (Figure 9)

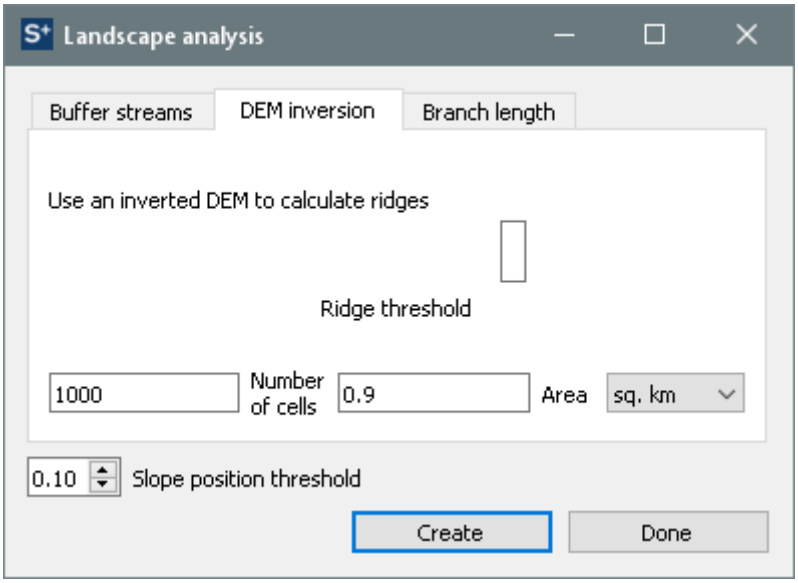


Figure 9: Creating landscape units by inverting the DEM

The third method of creating landscape units is the branch length method. This uses the same slope position method, but a different method for defining ridges. We can calculate the *branch length* of any pair of adjacent points as the lesser of the distances from each point to where their flow paths meet (which may be infinite if these paths go to different watershed outlets). A point is said to be on a ridge if it has a branch length with an adjacent point which exceeds a threshold. Then for each point in the watershed we use for its stream point where its flow path meets a stream, and for its ridge point the nearest point classified as on a ridge, and calculate its slope position as with DEM inversion. Users should note that the branch length method takes perhaps an order of magnitude longer than DEM inversion to calculate.

Since it is not clear what method will be best for any particular watershed, the choice needs to be made by the user, and is probably best made visually. The interface therefore allows for several versions of possible landscape units to be made, and then one to be chosen later when HRUs are created. To this end, landscape unit rasters show just the floodplain, and are given the name *bufferfloodn* if created by buffering, where *n* is the buffer multiplier, *invfloodn_m* if created by DEM inversion, where *n.m* is the slope position threshold, and *branchfloodn_m* if created by the branch length method. Users can therefore if they wish create several candidates, using different methods and/or different parameters. Figure 10 shows the result of creating three using each of the methods with the default parameter settings, and making only the third currently visible in the map canvas. For this exercise you should create the same three floodplain rasters. Click **Done** to close the **Landscape analysis** form.

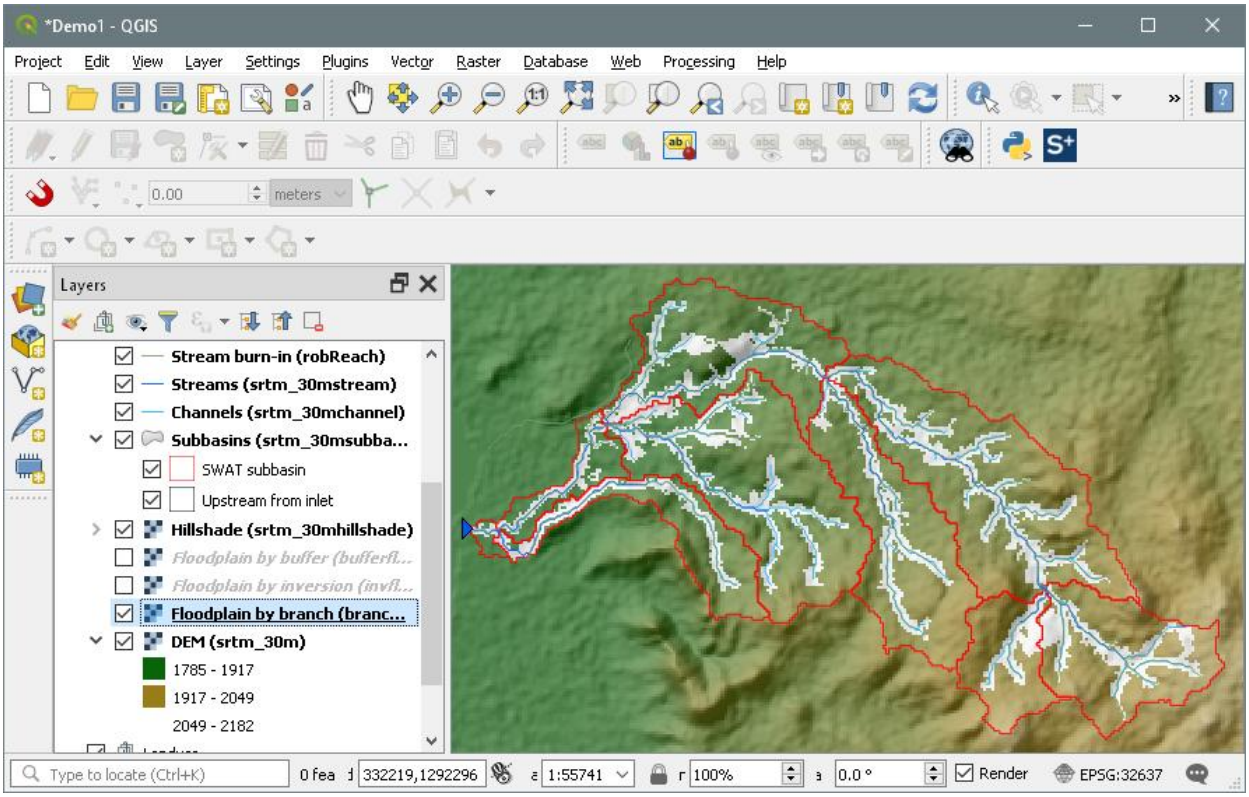


Figure 10: Potential floodplain rasters created

7.1.4 Merging subbasins

In QSWAT+ users can select and merge subbasins. This is especially important in avoiding small subbasins. To merge subbasins, select the option **Select subbasins** in the **Merge subbasins** tab. Hold the Ctrl key and click in subbasins you want to select. Selected subbasins will turn yellow, and a count is shown at the bottom left of the dialogue window (Figure 11).

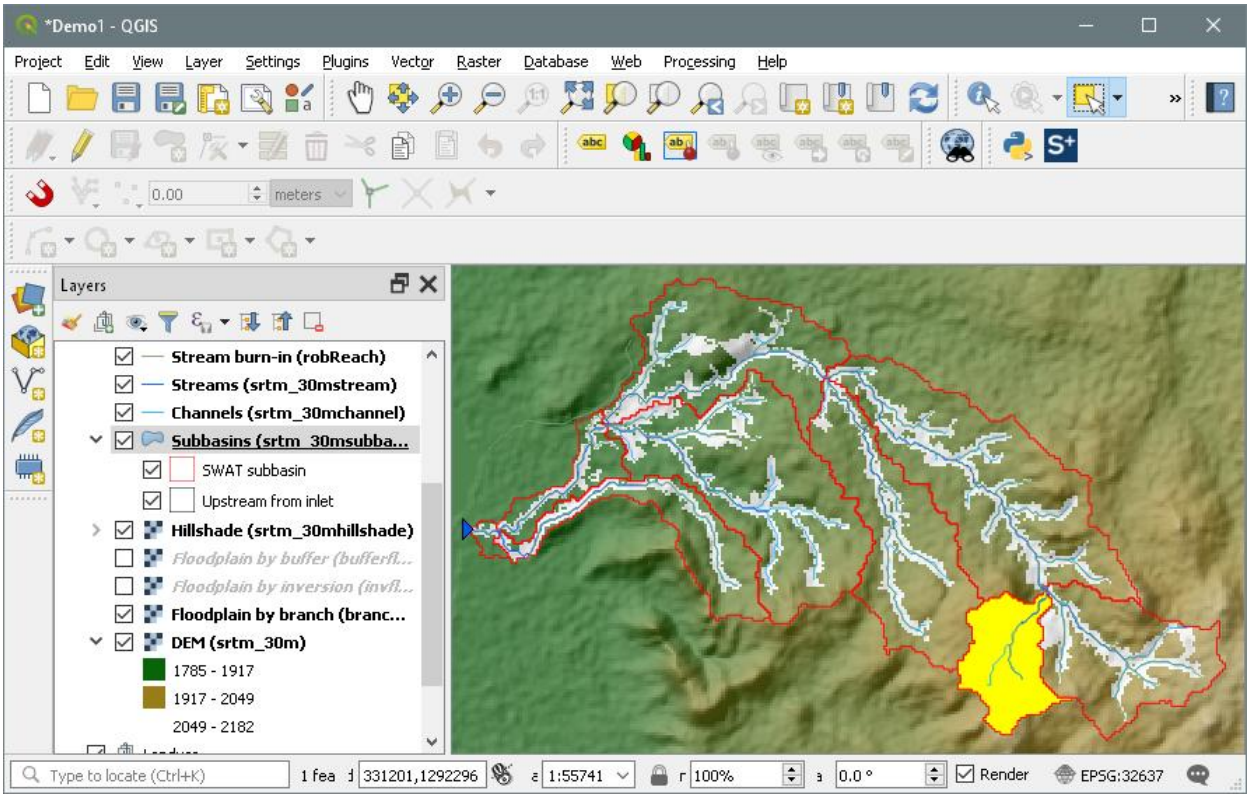


Figure 11: Subbasin identified for merging

For the sake of this exercise, we will not merge the selected subbasin. Release the Ctrl key and click on the map outside the watershed and the selected subbasin will be unselected. Checking the option of **Select small subbasins** provides the option to merge subbasins below a certain threshold (i.e. either the area in hectares or a percentage of the mean subbasin area) (Figure 12). For this exercise click the radio button **percentage of mean area**, and leave the default figure of 5 in the box next to it. Click **Select**. It will show you subbasins that have an area less than 5% of the mean area. When finished click "**Save**" to save your selection, or "**Cancel**" to abandon your selection. You click **Merge** to perform the merge. Note: You cannot merge a subbasin that has an outlet or inlet at its exit. So in this example, the only small subbasin is the one at the watershed outlet, which cannot be merged, and QSWAT+ will report this.

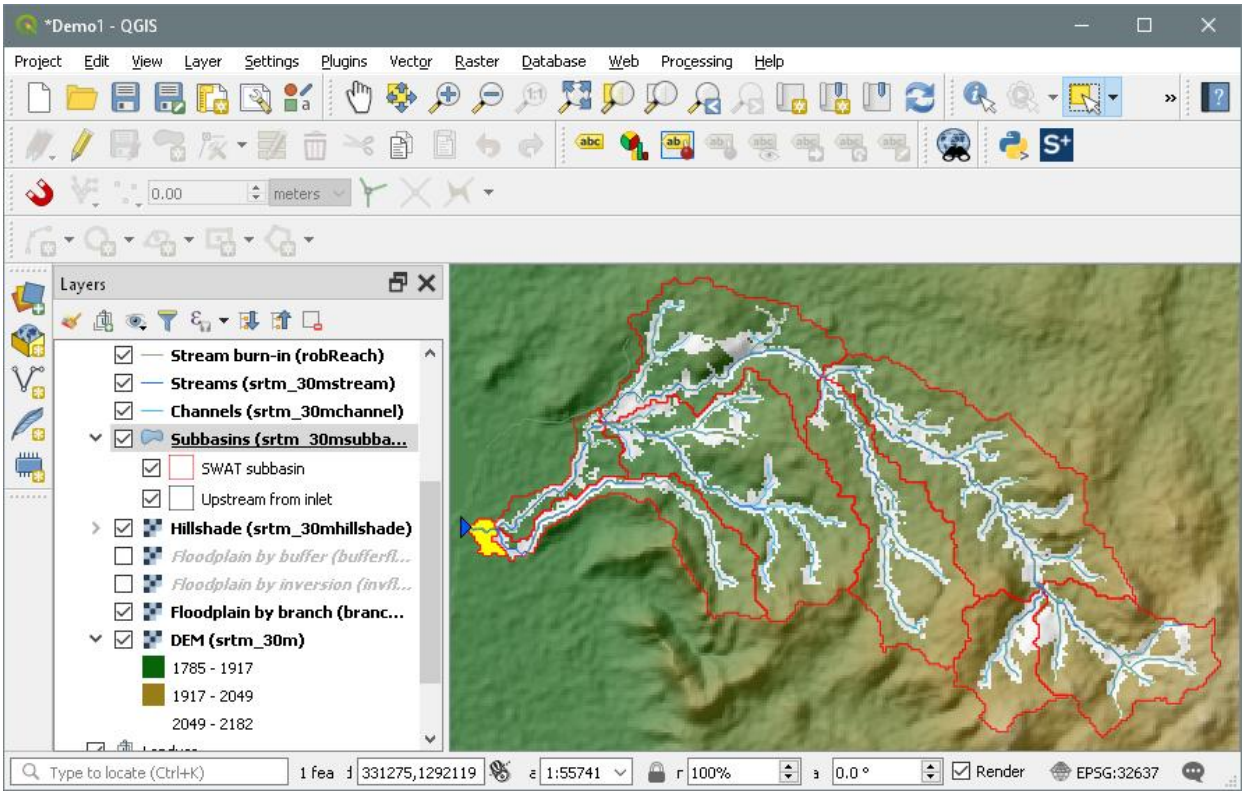


Figure 12. Subbasins that have an area less than 5% of the mean subbasin area intended for merging with downstream subbasins.

The third tab in the delineation form is to **Add Lakes**. We have no lakes to add in our Robit example, and it is a substantial topic, so we will return to it later in section 11.

To complete watershed delineation you need to click **OK**. This calculates and saves information about the watershed, with the message *Constructing topology*, and causes the subbasins to be numbered. This step ends watershed delineation, and enables the **Create HRUs** option (Figure 13).

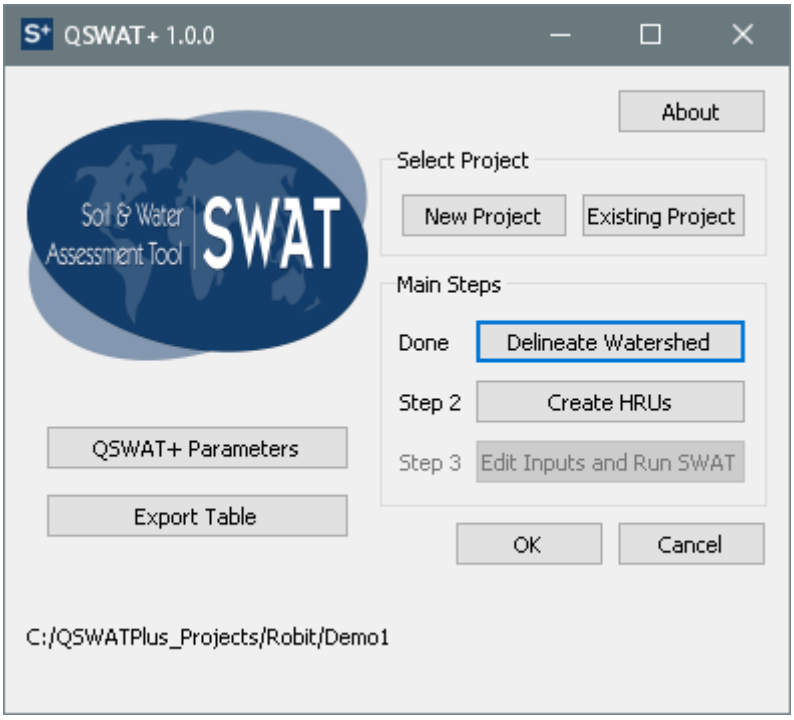


Figure 13. Watershed delineation is completed, and Create HRUs is enabled

7.2 Create HRUs

7.2.1 Landuse and soil maps

Having calculated the basins you needed to calculate the details of the Hydrological Response Units (HRUs) that are used by SWAT+. These are divisions of basins into smaller units each of which has a particular soil/landuse (crop)/slope range combination.

To do this we click **Create HRUs**, select *Robit/Landuse/roblandusenew/hdr.adf* as the **Landuse map**, and select *Robit/Soil/mowr_soil90/hdr.adf* as the **Soil map**. You may notice that the landuse map is loaded as selected, as an ESRI raster with a name *hdr.adf*. But the soil map, instead of being *mowr_soil90/hdr.adf* is called *mowr_soil90_clip.tif*. This is because QSWAT+ checks to see if landuse and soil maps are substantially bigger than the DEM (at least 10 DEM pixels in any direction) and if so clips them to avoid loading a larger map than necessary. The clipping also converts them to GeoTiff format. So when preparing such maps for QSWAT+ you need to make sure their projection matches the DEM, and that they easily cover the watershed area, but do not worry about clipping them to size

We need lookup tables to convert from the numeric values found in the landuse and soil maps to SWAT+ landuse codes and soil names respectively. These lookup tables must be in the project database. You can prepare these in advance, or use predefined ones like *global_landuses* and *global_soils* intended for use with the corresponding global maps, or, as we will do here, use the option to import a comma separated values (.csv) file. Lookup tables for landuse must have the string *landuse* in their names; lookup tables for soils must have the string *soil* in their names, or they will not appear in the pull-down menus **Landuse**

table and **Soil table** respectively.

For landuse, select the option *Use csv file* in the **Landuse table** pull-down menu. A form appears asking for the landuse lookup csv file. Browse to *Robit*/*Robit_landuses.csv* and click **Open**. This reads the csv file and makes a table *Robit_landuses* in the project database. The csv file has the appropriate structure for a landuse lookup table. Its contents are shown in Figure 14. The first line, commonly included, is ignored if its first field is not numeric. On each other line, the first field, delimited by a comma, is a value from the landuse map. The second field is a SWAT+ landuse code, as found in one of the tables *plant* or *urban* in the project database, or as defined by the user in a new table. Optionally, a third column with heading *DESCRIPTION* can be included, and a third text field (possibly empty, but preceded by a comma) included on each following line. More than one map value may map to the same landuse code, but map values may not be repeated. The table name is determined as follows. First, if the file name contains *landuse* (as ours does) it is used as the basis. Otherwise *landuse_lookup* is used as the basis. Second, if the basis does not exist as a table name it is used, otherwise successively 0, 1, etc is appended until an unused table name is found and used. In our case *Robit_landuses* will be the table name. You will not need to read the csv file again: if you rerun the project, *Robit_landuses* will be available in the pull-down menu (and will be the default selection as it will be recorded in the project file as the landuse lookup table).

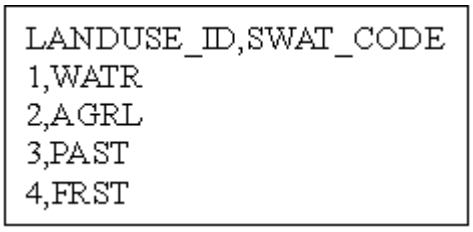


Figure 14: Robit_landuses.csv

The SWAT+ landuse codes are usually to be found in the tables *plant* or *urban* in the project database, but users are free to set up their own tables. First, they can change the landuse and soil database from its default of the project database to another database. Second, the *plant* or *urban* table need not use that name: the pull down menus labelled **Plant table** and **Urban table** will include any tables with *plant* or *urban* respectively in their names. Users can also use the *Use csv* options in these menus to load a csv file with the appropriate data and create a suitable table.

For grid models, where grid cells may contain only a few landuse cells, there is a danger that nodata landuse cells could cause the whole grid cell to have no landuse value. To avoid this, nodata landuse values are replaced by a default value. The default value is the value mapped by zero in the landuse lookup table, else the first one read. Since the order of entries in database tables is arbitrary, it is a good idea to include 0 in lookup tables if you are creating grid models.

For soil, similarly select *Use csv file*, and then open *Robit*/*Robit_soils.csv*. It maps values from our soil map to soil names found in the *SNAM* field of the *usersoil* table we are about to create.

7.2.2 Defining soil properties

Unless our project is in the USA and we are using SSURGO or STATSGO, we also need a table describing soil properties. This traditionally has a name *usersoil,* or one including that name, and is typically to be found in the project database. These are the defaults currently in the form for the database and the table name. You can change the database if you have your soil data in some other SQLite database, (but it must include your plant and urban tables as well) and you can use any table with *usersoil* in its name. The default *global_usersoil* in the project database is appropriate for the global soils maps from the FAO, but not in our case. Leave the default landuse and soil database selection as it is (the project database), make sure the **Soil data** option is set to *usersoil,* and select *Use csv file* in the **Usersoil table** pull-down menu. Open . *Robit*/*Robit_usersoil.csv* to create and select the table *Robit_usersoil.*

The other **Soil data** options are for watersheds in the USA for which you are using STATSGO, STATSGO2, or SSURGO soil maps. If you select the **STATSGO** or the **SSURGO/STATSGO2** option then soil data will automatically be extracted from the *swatplus_soils.sqlite* database stored in *C:/SWAT/SWATPlus/Databases.* No soil lookup table is needed with SSURGO/STATSGO2. For STATSGO, a lookup table is required, and it will map soil map values to one of four options, all based on the table *statsgo* in *swatplus_soils.sqlite*:

1. *muid* values such as AL001
2. *muid+seqn* values such as AL001+1
3. *muid+name* values such as AL001+DICKSON
4. *s5id* values such as TN0042

The lookup table must use one of these options consistently. Where lookup target matches more than one row in the *statsgo* table the first (as ordered by increasing *id* values) is taken. So the four examples above would all give the first row (*id* 1) in the table.

We will use one of the floodplain maps we created earlier, to divide our subbasins into landscape units. Use the pull-down menu labelled **Select floodplain map (optional)** to select *invflood0_1.tif*. This menu will include all files found in the project's *Watershed/Landscape/Flood* folder: you can add such files yourself if you wish. If you do not select from this menu. no landscape units will be created. If you place your own flood maps in this folder you should **not** include the string *flood* in the their names, since such files will be removed by watershed delineation. Floodplain maps created by QSWAT+ need to be recreated after rerunning delineation because they are based on the channel network, which is likely to change when delineation is rerun with different parameters or with changed inlets or outlets.

We will form HRUs based on slope as well as landuse and soil. We add an intermediate point for slopes (e.g. 10) to divide HRUs into those with average slopes in the range 0-10% and

those with average slopes above 10%. Type 10 in the box and click **Insert**. The **Slope bands** box shows the intermediate limit is inserted.

We will also generate a **FullHRUs** shapefile: click in the check box. Generating this file is optional, as it can take some time if there are many HRUs. If you intend to visualise HRU outputs after running SWAT+ then you need to generate this file.

To read in the data from the DEM, landuse, soil and slope maps and prepare to calculate HRUs, make sure **Read from maps** is checked and click **Read** (Figure 15). There is also an option **Read from previous run**. This can be used when rerunning the project, provided you have not rerun the delineation step, and not changed the soil or landuse inputs, or the slope bands, to recover information from the project database instead of rereading the grids. Reading from the database is substantially faster.

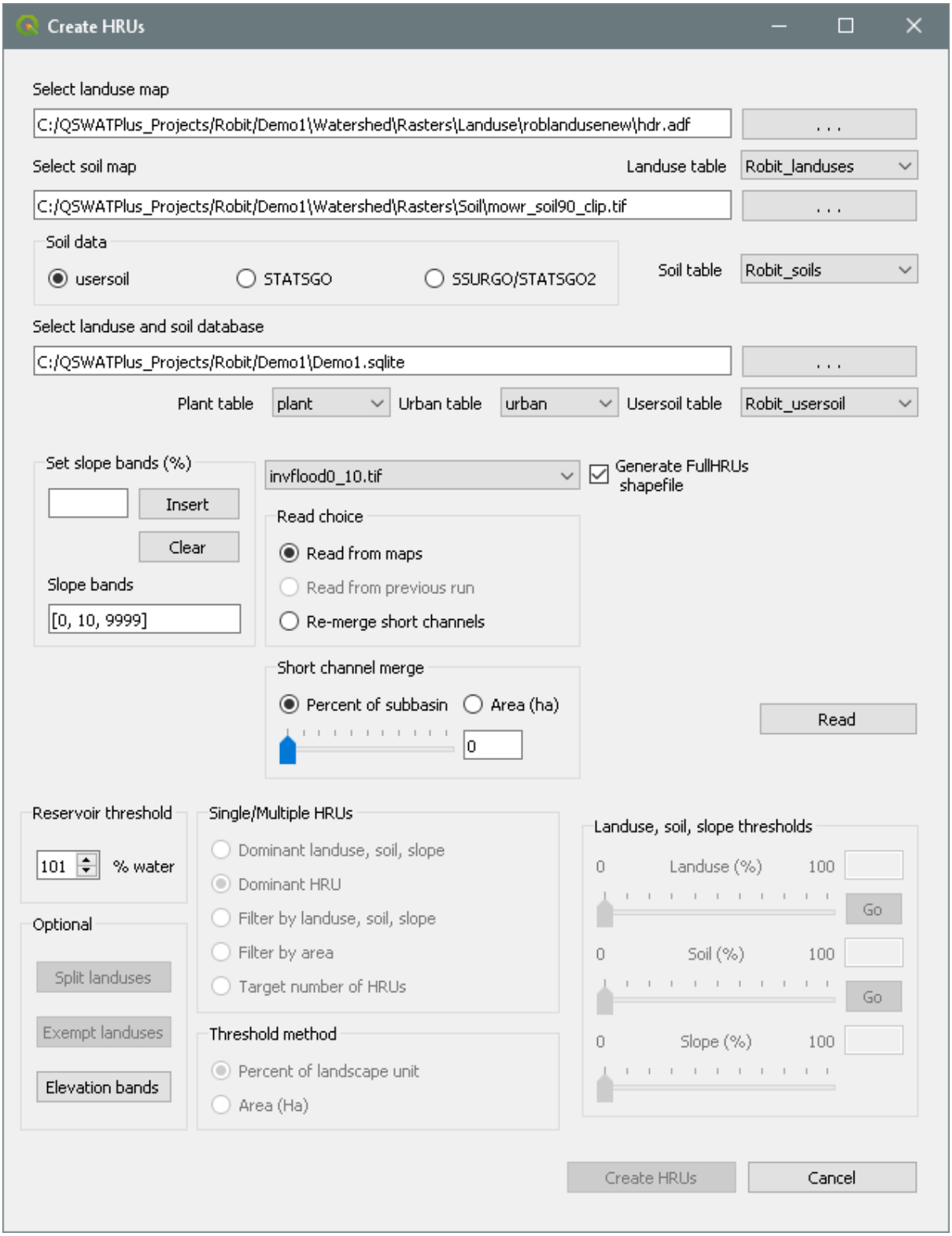


Figure 15: Ready to read files

After reading the grids you will notice a number of changes to the QGIS display (Figure 16):

- A **Slope bands** map has been created and added. This allows you to see where the areas of the two slope bands selected for this project are located. If no intermediate slope limits are chosen this map is not created.
- The legend for the landuse map *robelandusenew* includes the landuse categories from

its lookup table. (This should also happen for the soil map, but in this case there is apparently a very wide range of values, from 106 to 198, but QGIS does not recognize that only 4 values actually occur.).

- Shapefiles *Full HRUs* and *Full LSUs* have been created and added. This allows users to see where in each subbasin the landscape units (LSUs) and potential Hydrological Response Units (HRUs) are physically located. A landscape unit is the region draining into a channel reach, further divided into the floodplain and upslope regions if a floodplain raster is used. LSUs form single regions: they each form a single polygon. An HRU is the collection of pixels within an LSU that share the same landuse, soil and slope range. HRUs typically form a collection of polygons. If, for example, we zoom in on subbasin 3, in the Legend panel select (left button) Full HRUs, open its attribute table (right button), set the mouse to *Select Feature(s)* (QGIS toolbar, or use *View -> Select -> Select feature(s)*) and click on the south-east of subbasin 3, then scroll through the attribute table to find the line selected, then we get a view like Figure 17. We see that this potential HRU is composed of several parts, is in subbasin 3 as we expected, drains to channel 57, has the landscape unit Upslope, landuse AGRL, the soil LVx, and the slope band 10 upwards. Its area of 18.45 ha is 5.4% of the subbasin and 59.9% of the channel's upslope landscape unit. **Close** the Attribute Table Editor. Generating the FullHRUs shapefile is optional, as it can take some time, but is necessary if you intend later to visualise HRU results. Counts of the subbasins (9), channels (87) and full (potential) HRUs (516) are displayed in the *Create HRUs* form whether the FullHRUs file is created or not.

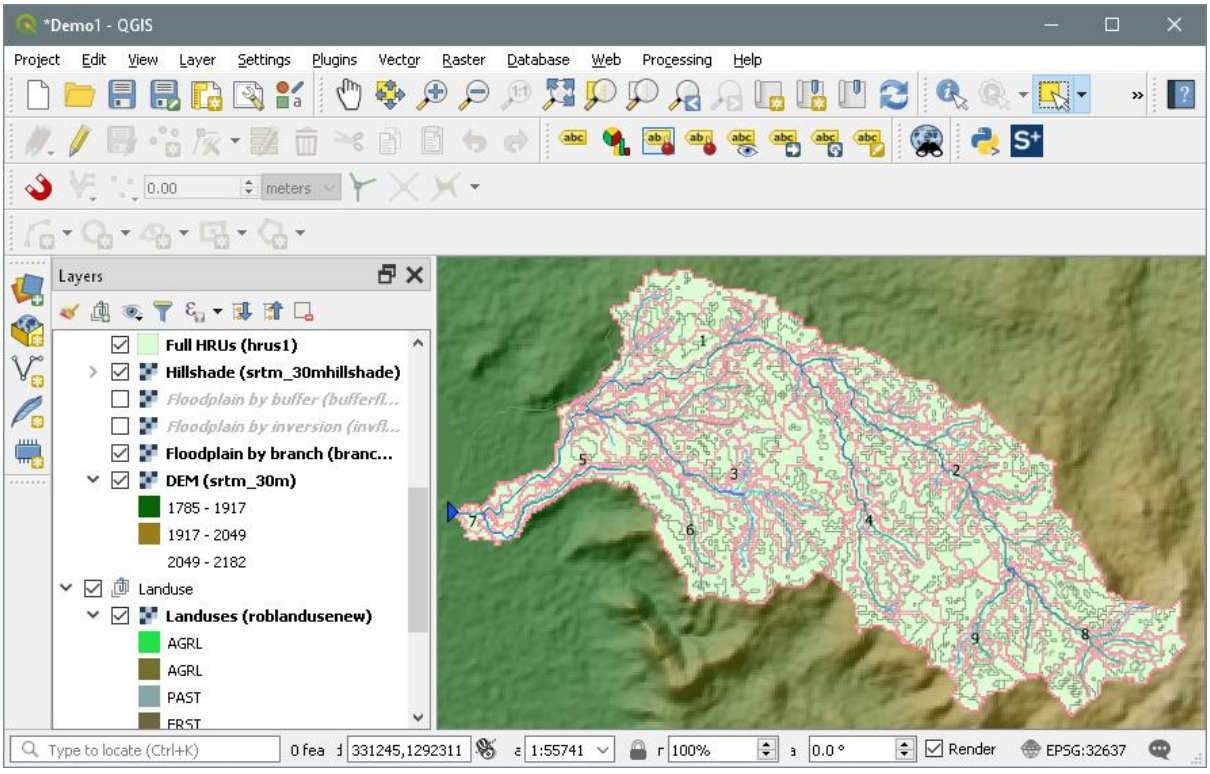


Figure 16: After reading grids

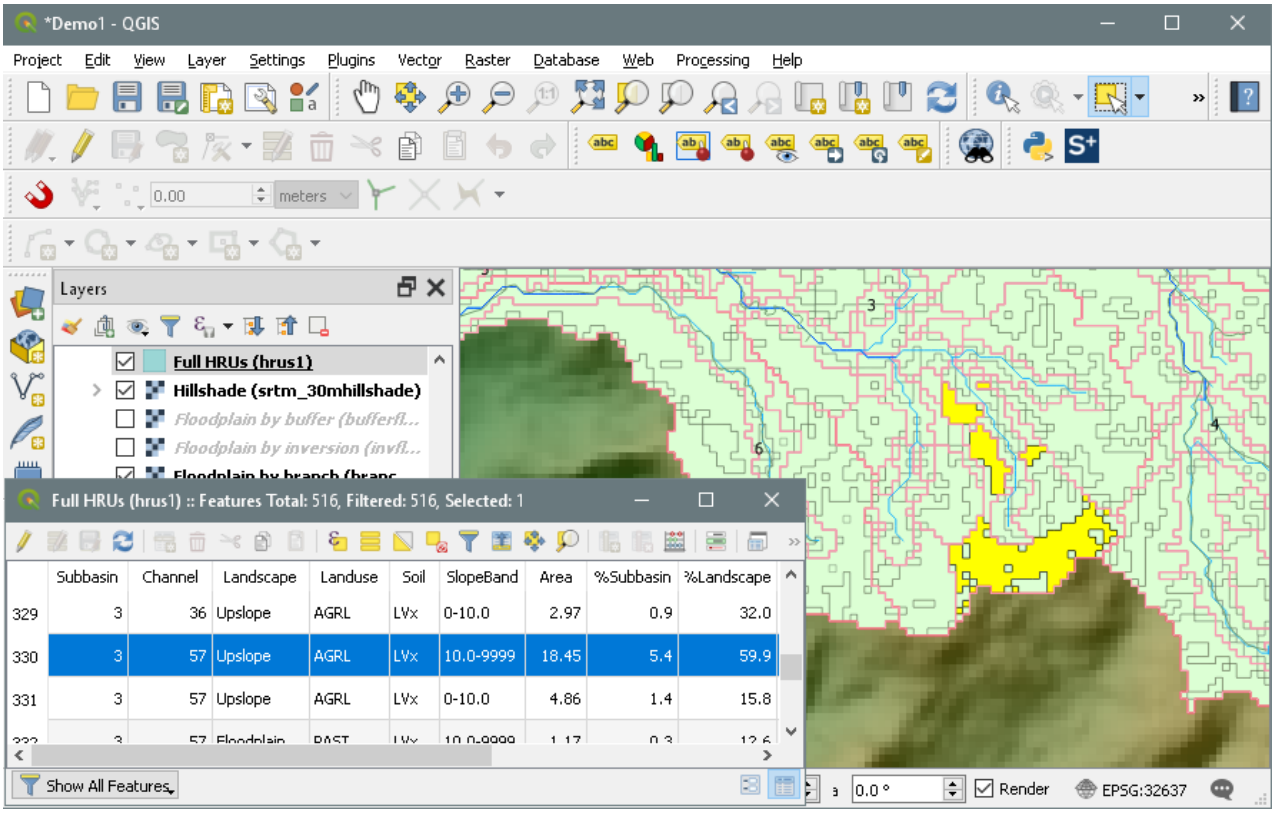
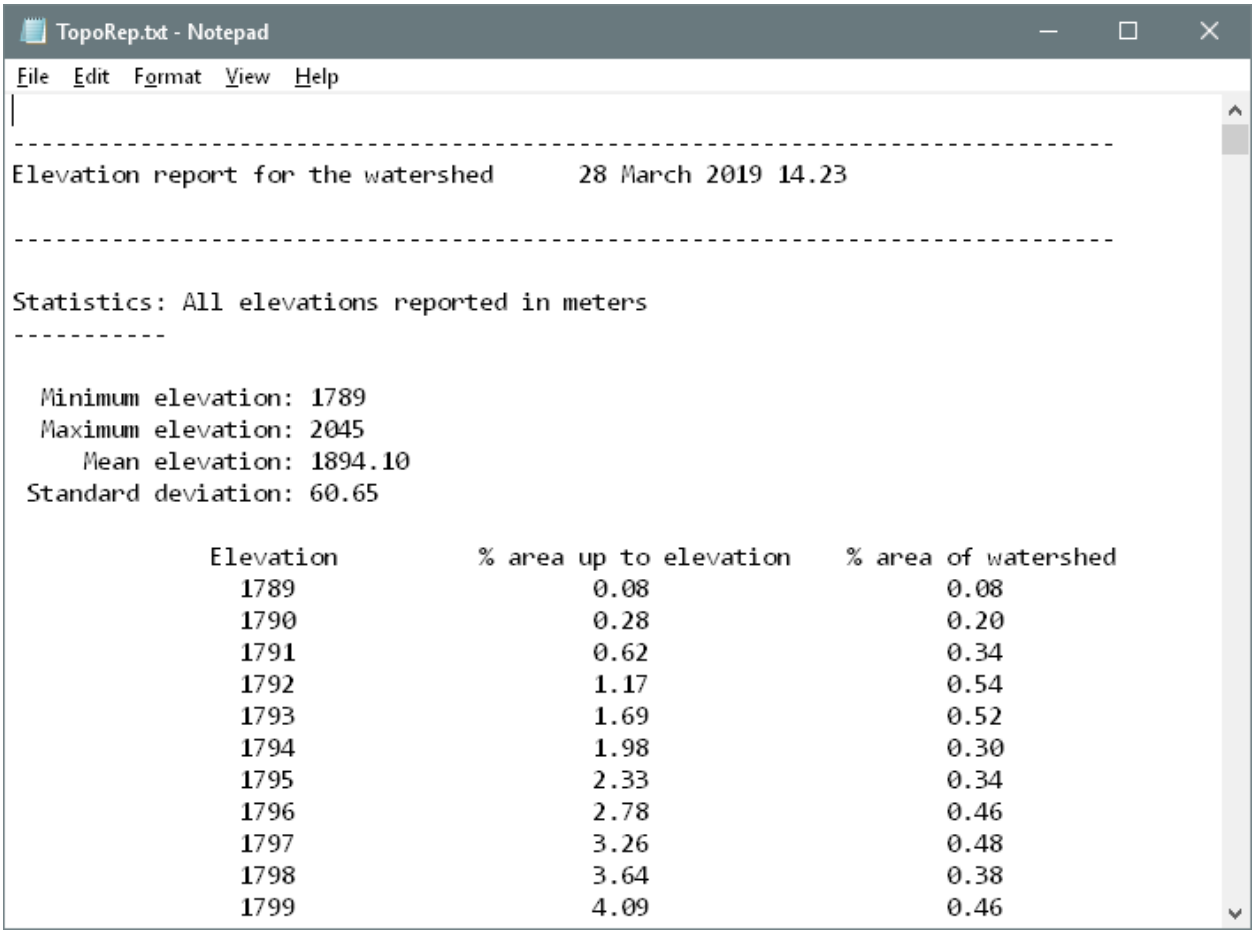


Figure 17: Viewing a particular HRU

Before you continue with HRU definition, if you look at the main QSWAT+ window you see that a new item **Select report to view** is available and you can choose to view just two reports at this point, which are the **Elevation** and **Landuse and Soil** reports. The elevation report gives information about how much land is at each elevation from the lowest to the highest, both for the watershed as a whole and for each subbasin (Figure 18). The landuse and soil report lists the landuse, soil and slope-band areas for each subbasin (Figure 19). It also lists the areas, divided into landscape units if these are in use, for each channel reach.

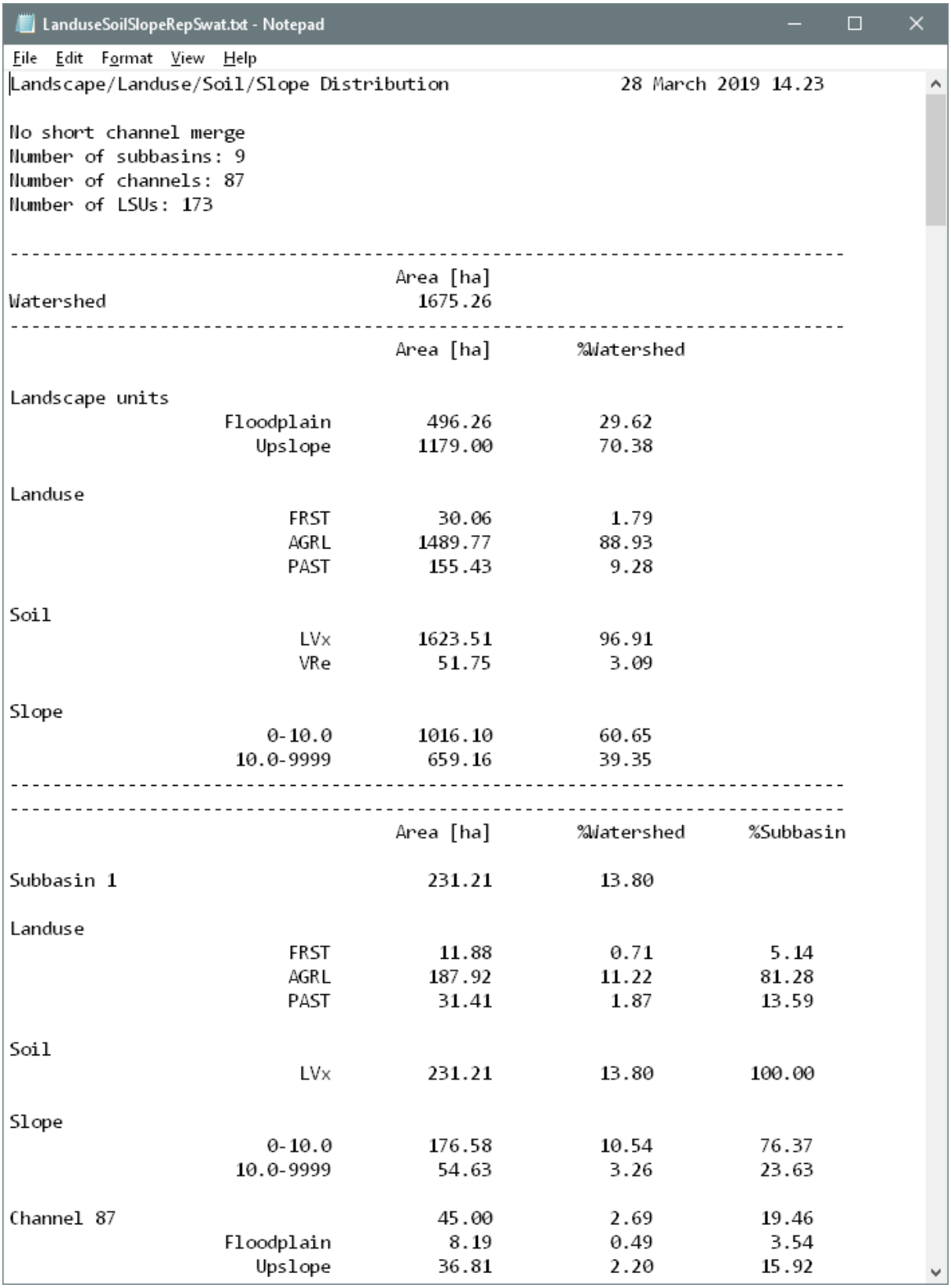


TopoRep.txt - Notepad

File Edit Format View Help

```
--------------------------------------------------------------------------------
Elevation report for the watershed      28 March 2019 14.23

--------------------------------------------------------------------------------

Statistics: All elevations reported in meters
-----------

  Minimum elevation: 1789
  Maximum elevation: 2045
     Mean elevation: 1894.10
 Standard deviation: 60.65

             Elevation         % area up to elevation    % area of watershed
               1789                    0.08                    0.08
               1790                    0.28                    0.20
               1791                    0.62                    0.34
               1792                    1.17                    0.54
               1793                    1.69                    0.52
               1794                    1.98                    0.30
               1795                    2.33                    0.34
               1796                    2.78                    0.46
               1797                    3.26                    0.48
               1798                    3.64                    0.38
               1799                    4.09                    0.46
```

Figure 18: Elevation report



LanduseSoilSlopeRepSwat.txt - Notepad

File Edit Format View Help

```
Landscape/Landuse/Soil/Slope Distribution                    28 March 2019 14.23

No short channel merge
Number of subbasins: 9
Number of channels: 87
Number of LSUs: 173

-------------------------------------------------------------------------------
                                  Area [ha]
Watershed                          1675.26
-------------------------------------------------------------------------------
                                  Area [ha]       %Watershed

Landscape units
                    Floodplain        496.26          29.62
                       Upslope       1179.00          70.38

Landuse
                          FRST          30.06           1.79
                          AGRL        1489.77          88.93
                          PAST         155.43           9.28

Soil
                           LVx        1623.51          96.91
                           VRe          51.75           3.09

Slope
                        0-10.0        1016.10          60.65
                     10.0-9999         659.16          39.35
-------------------------------------------------------------------------------
-------------------------------------------------------------------------------
                                  Area [ha]       %Watershed      %Subbasin

Subbasin 1                         231.21          13.80

Landuse
                          FRST          11.88           0.71           5.14
                          AGRL         187.92          11.22          81.28
                          PAST          31.41           1.87          13.59

Soil
                           LVx         231.21          13.80         100.00

Slope
                        0-10.0         176.58          10.54          76.37
                     10.0-9999          54.63           3.26          23.63

Channel 87                          45.00           2.69          19.46
                    Floodplain          8.19           0.49           3.54
                       Upslope         36.81           2.20          15.92
```

Figure 19: Landuse and soil report

7.2.3 Merging short channels

It often happens that although we set a certain threshold for the area needed to form a channel

(9 hectares in our case), channel reaches within the network can be very short, because junctions happen to be near each other, and their drainage areas correspondingly small. Figure 20 shows a channel reach in subbasin 3 only 72.4 metres long with a drainage area, landscape unit (LSU) number 121 (highlighted in yellow) of only 0.54 ha. This is in fact divided into 2 potential HRUs, one of only 0.09 ha! Such HRUs are best removed because SWAT+ will spend time processing them but their effect will be insignificant. We will later be removing small HRUs, but there is a rule that each channel reach, and indeed each landscape unit if we use them, will retain at least one. To counter this there is an option to merge short channels into neighbouring ones, by selecting a threshold either as a percent of subbasin or as an area in hectares, and recreating the channel network. Use the slider in the *Short channel merge* group (or type in the box) to set the threshold at 2%, select the *Re-merge short channels* option and click *Read* again. We see that the channel count drops to 72, and the full HRUs count to 455[2]. If we inspect the landuse and soil report for subbasin 3, we see that the smallest channel drainage area now is 2.62% of the subbasin, with a drainage area of 9 hectares. If we still think this too small we can change the short channel threshold and click **Read** again. But we will continue with the setting we have. The short channel merge threshold can be set before the first **Read** if you wish.

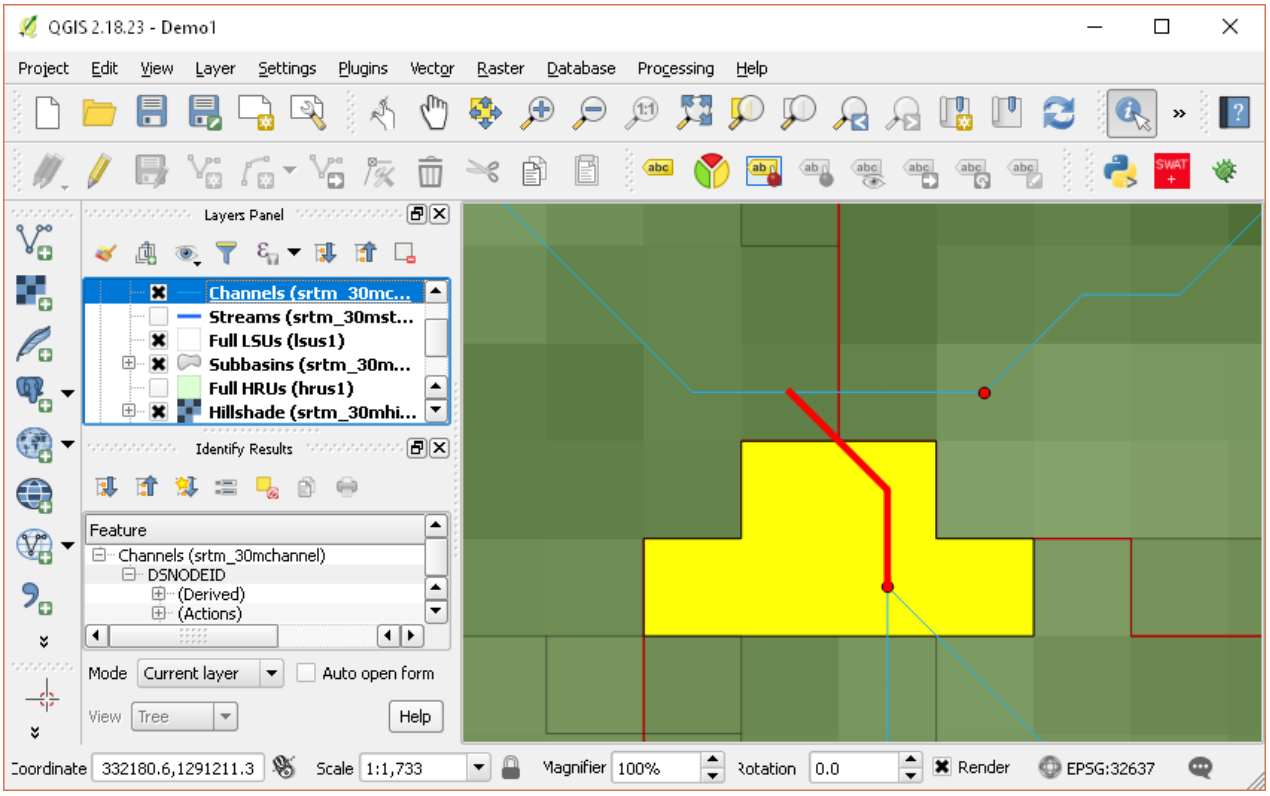


Figure 20: Small channel reach and its drainage area

7.2.4 Splitting and exempting landuses

At this point you have the options to split landuses, and to exempt landuses, both of which

[2] HRU counts may differ slightly on different machines

will affect how HRUs are defined.

- Splitting landuses allows you to define more precise landuses than your landuse map provides. If, say, you know that in this basin 50% of the AGRL (Agricultural land generic) is used for corn, and the other 50% is used for teff, you could split AGRL into 50% CORN and 50% TEFF (Figure 21). It is possible to use the original landuse (here AGRL) as one of the sub-landuses. The percentages are integers and need to sum to 100 for each landuse you split. Split AGRL into 50% CORN and 50% TEFF as shown. To do this, first select AGRL in the **Select landuse to split** pull down menu. This immediately raises a **Select sub-landuse** form. Select **CORN** and click **OK**. This completes the first line of the table. Click the **Add sub-landuse** button, select **TEFF** and click **OK**. This adds the second line. Then change the two percentages to 50, Click **Save edits** to store the contents of the table as a split landuse, and finally click **Save splits** to complete the form and exit from it.
- Exempting landuses allows us to ensure that a landuse is retained in the HRU calculation even if it falls below the thresholds we will define later. For example, we might decide to exempt the forest landuse FRST (Figure 22). Make this exemption.

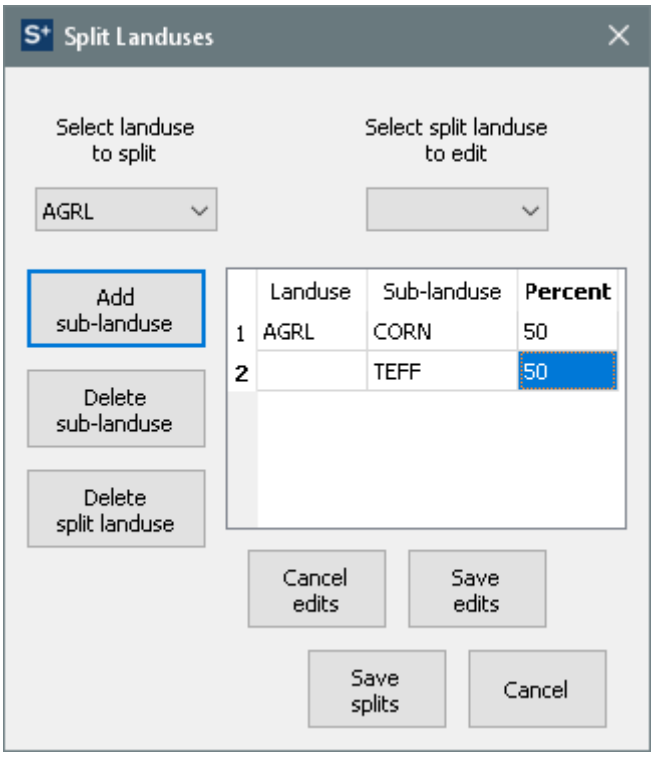


Figure 21: Splitting a landuse

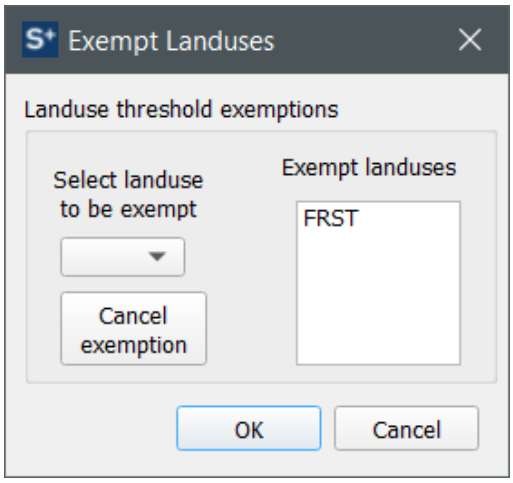


Figure 22: Exempting a landuse

7.2.5 Ponds and reservoirs

There are 3 ways in which ponds and reservoirs may be formed in QSWAT+:

1. Lakes can be added, and these must be designated as either ponds or reservoirs. See section 11.
2. Points can be identified as pond or reservoir points in the inlets/outlets shapefile. In this case each such point must be within snapping point of a channel. The channel is divided at the snap point into two, the downstream portion remaining as a channel and the upstream portion becoming a pond or reservoir. Any pixels with landuse WATR in the adjacent LSU become part of the pond or reservoir, instead of potentially becoming HRUs. Here the channel's *adjacent LSU* is the floodplain LSU if a floodplain map is selected, else the whole LSU.
3. A reservoir threshold percentage can be set. This is the minimum percentage of WATR in the adjacent LSU for the WATR area to become a reservoir. When this threshold is met, the channel is replaced by a reservoir, with area the WATR area of the adjacent LSU. The default threshold is 101%, effectively turning off the automatic creation of reservoirs. If we change the threshold at this point we will need to re-read the data from the landuse, soil and slope maps: **Read from previous run** is selected and the HRU creation options are disabled until we click **Read** again. The short channel merge setting is retained (or can be changed) and will be reapplied. Connected channels which meet the threshold for forming automatic reservoirs will form a single reservoir. The exception to this is when a pond or reservoir has been marked on a channel during delineation. The pond or reservoir's position prevents it being combined with a reservoir downstream. This gives users a means to control the otherwise automatic merging of reservoirs.

Channels which become ponds or reservoirs effectively disappear from the routing (water flow) network. Channels, landscape units and HRUs that would normally be routed to them are routed to the pond or reservoir instead.

This watershed has no WATR areas and so no reservoirs or ponds could be formed, even if we reduced the reservoir threshold.

7.2.6 Creating HRUs

Once all the data has been read in and stored, the **Single/Multiple HRUs** choice is enabled. **Dominant landuse, soil, slope** and **Dominant HRU** options provide just one HRU for each landscape unit (i.e. each is a single HRU option). **Dominant landuse, soil, slope** chooses the landuse with the biggest area in the landscape unit, the soil with the biggest area in the landscape unit, and the slope range with the biggest area in the landscape unit and uses them for the whole landscape unit. **Dominant HRU** selects the largest of the potential HRUs in each landscape unit and makes its landuse, soil and slope range the ones chosen for the whole landscape unit.

The options with **Filter by landuse, soil, slope; Filter by area;** and **Target number of HRUs** create possible multiple HRUs in each landscape unit. For the **Multiple HRUs** option, users can exclude HRUs that are insignificant by considering **percentage thresholds** or **area thresholds**. The option of **Filter b*y* land use, soil and slope,** will ignore any potential HRUs for which the landuse, soil or slope is less than the selected threshold. The areas of HRUs that are ignored will be redistributed proportionately amongst those that are retained within each landscape unit. The option of **Filter by area** will ignore HRUs that are below a specified threshold amount (either percent of the landscape unit, or area). Finally the option of **Target number of HRUs** limits the number of HRUs in the whole watershed to a user preferred amount (i.e. between the lower limit of one HRU per landscape unit and the upper one of retaining all potential HRUs).

For this exercise, use the method of **Filter b*y* land use, soil and slope** and choose the **Threshold method** of **Percent of landscape unit**. For landuse, the value of 51 as the maximum we can choose indicates that there is a landscape unit where the maximum value for a landuse is 51%. If we chose a higher value than 51% we would be trying to ignore all the landuse categories in that landscape unit. Hence 51% is the minimum across the landscape units of the maximum landuse percentage in each landscape unit. For this exercise, select 10% for landuse, by using the slider or by typing in the box, and click **Go**. The interface then computes the min-max percentage for a soil as 51%. Select 10 for Soil, click **Go**. Similarly select 10 for slope. At this stage the user can create HRUs by clicking **Create HRUs** (Figure 23). The number of HRUs created (651) is reported at the top of the map canvas. (This figure is higher than the FullHRUs count (455) reported earlier because of the splitting of AGRL HRUs.)

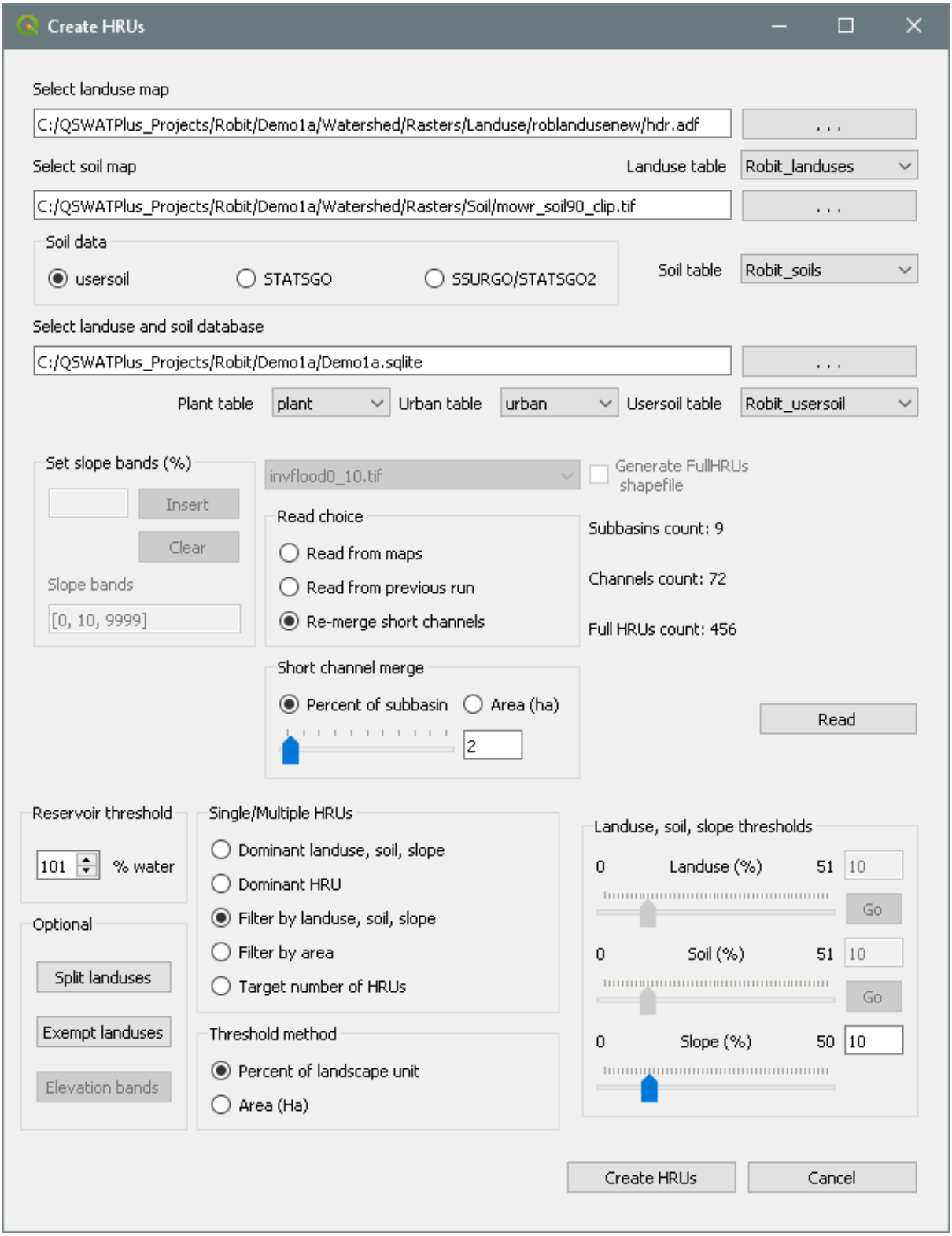


Figure 23: Creating multiple HRUs by Filter by land use, soil and slope

Create HRUs is now reported as done and the third step is enabled (Figure 24). A file *Actual LSUs* has been added and, if you requested a *Full HRUs* file, an *Actual HRUs* file has also been added. These contain the actual LSUs after any channel merges, and the potential HRUs that were retained as actual HRUs (and so typically has some holes representing those that were not retained). The actual LSUs shapefile attributes table shows the LSU identifiers, the

channel and the subbasin. The actual HRUs shapefile attribute table shows the subbasin and channel, the landscape category, (original) landuse, soil, and slope band, and the HRU number(s). In the case of split HRUs there can be more than one HRU number for a shape, as it is impossible to show in a shapefile the splitting by arbitrary amounts of the original potential HRUs. If you look at the *Reports* now available you will find that there is an *HRUs* report that only includes the landuses, soils and slope bands left after HRU selection, and also gives details of the HRUs that have been formed (Figure 25). If you wish to change the HRU thresholds then you can click the **Create HRUs** button again, select **Read from previous run,** click Read, change the thresholds and/or the landuses to be split or exempted, and rerun the **Create HRUs** step.

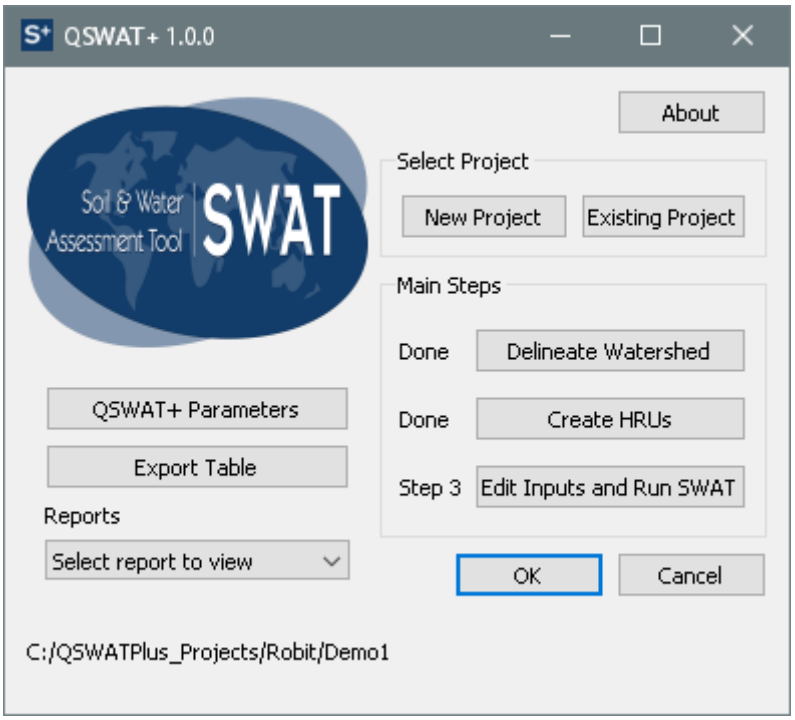
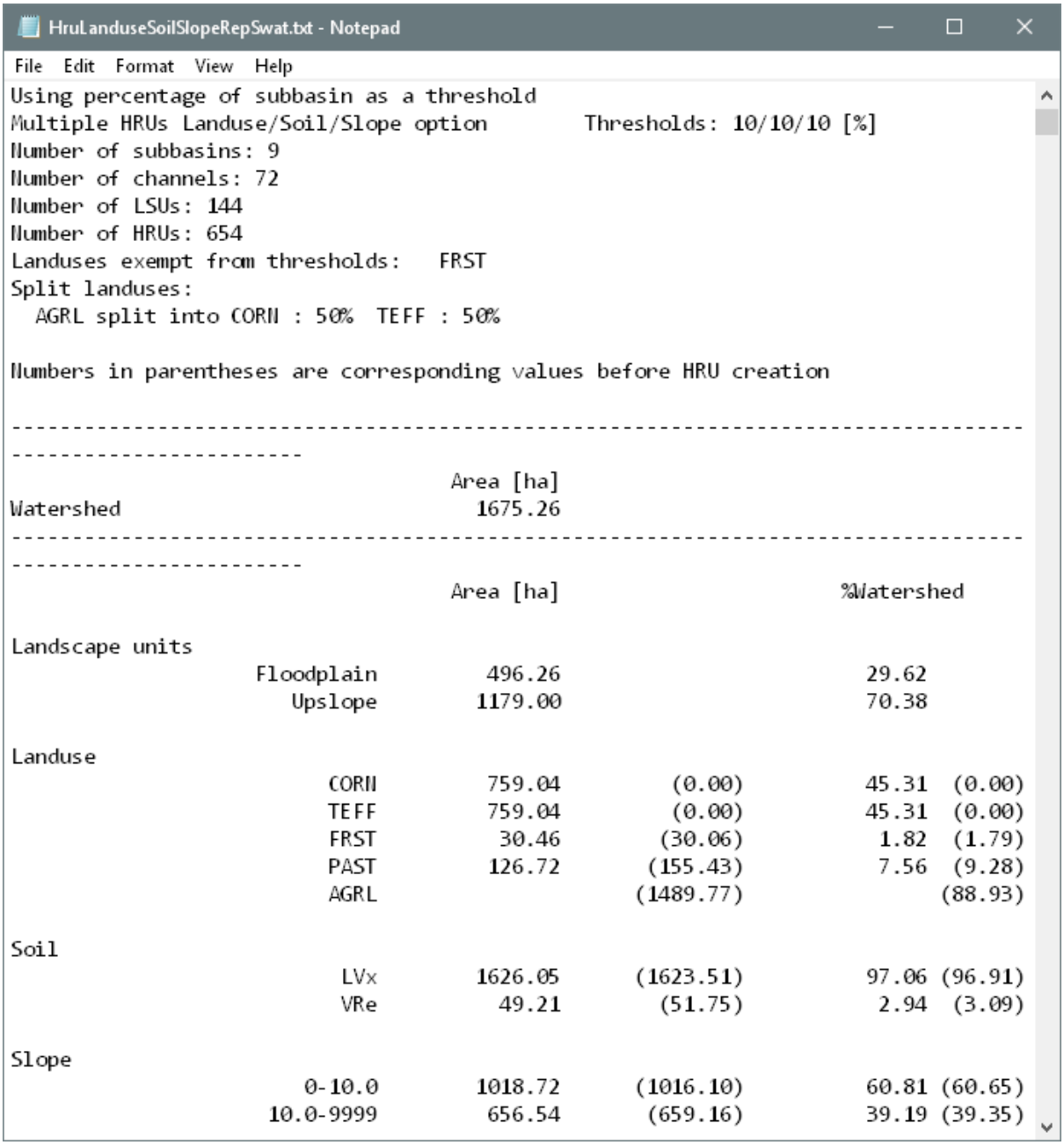


Figure 24: About to do step 3



```
HruLanduseSoilSlopeRepSwat.txt - Notepad
File  Edit  Format  View  Help
Using percentage of subbasin as a threshold
Multiple HRUs Landuse/Soil/Slope option        Thresholds: 10/10/10 [%]
Number of subbasins: 9
Number of channels: 72
Number of LSUs: 144
Number of HRUs: 654
Landuses exempt from thresholds:   FRST
Split landuses:
  AGRL split into CORN : 50%  TEFF : 50%

Numbers in parentheses are corresponding values before HRU creation

-------------------------------------------------------------------------------
-----------------------
                                   Area [ha]
Watershed                            1675.26
-------------------------------------------------------------------------------
-----------------------
                                   Area [ha]                      %Watershed

Landscape units
                    Floodplain        496.26                          29.62
                       Upslope       1179.00                          70.38

Landuse
                          CORN        759.04          (0.00)          45.31  (0.00)
                          TEFF        759.04          (0.00)          45.31  (0.00)
                          FRST         30.46         (30.06)           1.82  (1.79)
                          PAST        126.72        (155.43)           7.56  (9.28)
                          AGRL                     (1489.77)                (88.93)

Soil
                           LVx       1626.05       (1623.51)          97.06 (96.91)
                           VRe         49.21         (51.75)           2.94  (3.09)

Slope
                        0-10.0       1018.72       (1016.10)          60.81 (60.65)
                     10.0-9999        656.54        (659.16)          39.19 (39.35)
```

Figure 25: HRUs report

A new shapefile has been added, title **Channel reaches.** This is formed from the channels shapefile created during delineation, but has channels merged. Another shapefile, showing the point sources that are automatically added to each channel, and any reservoirs that were added, has also been added. A point source is added to every channel as a convenience. Users can add data (in the SWAT+ Editor) for any point sources they want to include in their model: point sources without data are ignored.

It is strongly recommended to save the project (via the file menu of QGIS) at this stage.

You are now able to click the **Edit Inputs and Run SWAT** button to start the SWAT+ Editor, set the rest of the parameters and input data you need, and run the SWAT+ model. This document does not cover the editor - you need to consult the SWAT+ Editor documentation

for details, but there are a few things you need to know about how to run the Robit example. Check the next subsection before starting the editor.

7.3 Setting up the Robit example in the SWAT Editor

Before starting, you can prepare weather generator information in the project database. See section 14.1 on how to do this.

Second you will need (if you have it) to supply observed weather data. The Robit example data includes such data, in *Robit/ClimateRobit,* and no more preparation is required.

After starting the editor, the initial setup involves confirming the project name, project database and swatplus_datasets (reference) database. Then click **Start Import**. Then in the next screen click **Start editing SWAT+ inputs,** and then **Add weather generators now.**

7.3.1 Weather data

The Weather Generator form should find the single station we imported into table *weather_sta_cli*. If you forgot to set up the weather generator tables just close the editor, follow the instructions in section 14.1 and restart the editor.

With the weather generator set up, select **Weather stations** at the top left. In the Weather Stations form click **Import data.** The Robit observed data is in the SWAT 2012 format, so leave this as the choice in the first box. For the weather files directory, browse to *Robit/ClimateRobit.* For a directory to save the files you can choose anywhere you like, preferably within the project tree so that these files will be available if you move or copy the project in future. You could use the *Scenarios/Default/TxtInOut* folder, where other input files will be stored, or create a new one. Leave the two boxes unchecked, and click **Start Import.** The Weather Stations form will reappear, showing a single station with data for precipitation, temperature, solar radiation, relative humidity and wind speed.

7.3.2 Simulation period

Select **Time** under **SIMULATION** on the left. We have daily data for the beginning of 1990 to the end of 2013, but to keep down the time needed to run and to import the output data we will restrict the period to 1990 to 2000. Enter these as the start and end years. Leave the start and end days as 0 (which means beginning and end of the year respectively) and the time steps as daily. Click **Save Changes.**

7.3.3 Print options

Select **Print** under **SIMULATION**. We will use 3 as the **nyskip** value, the warm up period. SWAT+ will run for those three years, but the output will be ignored. You can leave the other values as they are. Click **Save Changes.**

For Print Objects yearly output is preselected. We will visualise a variety of outputs, so select just three of the top boxes under **Monthly, Yearly** and **Avg. Annual**. Then find and add *channel_sd* under **Daily**, which we will need for calculating environmental flows. We won't include other daily output as it will take some time to write nearly 3000 records for each channel, lsu or hru in requested output. Scroll down and click **Save Changes.**

7.3.4 Run SWAT+

For this demonstration we will not make any editing changes to the inputs. Select the third symbol on the far left, with the tool tip **Write input files.** Leave the default *TxtInOut* folder as the location for the files to be written and click **Save & Write Files.** This warns about overwriting existing files, and we put our weather data there, but this is safe from being overwritten so click **Start Writing Files.** The form reports the files written, and we select the fourth symbol on the far left menu, to run the model. We don't need the debug version, which you would normally only use if the model has errors, to get more information. Click **Run SWAT+.** A new window shows the model running. When it concludes with *Execution successfully completed* click **Close.**

7.3.5 Save the outputs

In order to run visualisation we need to save the outputs in an output database, which will be *swatplus_outputs.sqlite* in the *Scenarios/Default/Results* folder. Select the fifth far left symbol **Analyze model output**, and then **Import Output.** We are warned about overwriting previous output. Click **Start Importing Output.** We selected a lot of outputs, so this takes a few minutes. When it is finished we can **Exit** (under **File)** from the SWAT+ editor.

8. Output Visualisation with QSWAT+

Once SWAT+ has been run and the output imported, so that there is an output database *swatplus_output.sqlite* in the *Scenarios/Default/Results* directory of the project, then a button *Visualise* becomes available in the main QSWAT+ form (Figure 26). Visualisation allows outputs from SWAT+ to be visualised graphically.

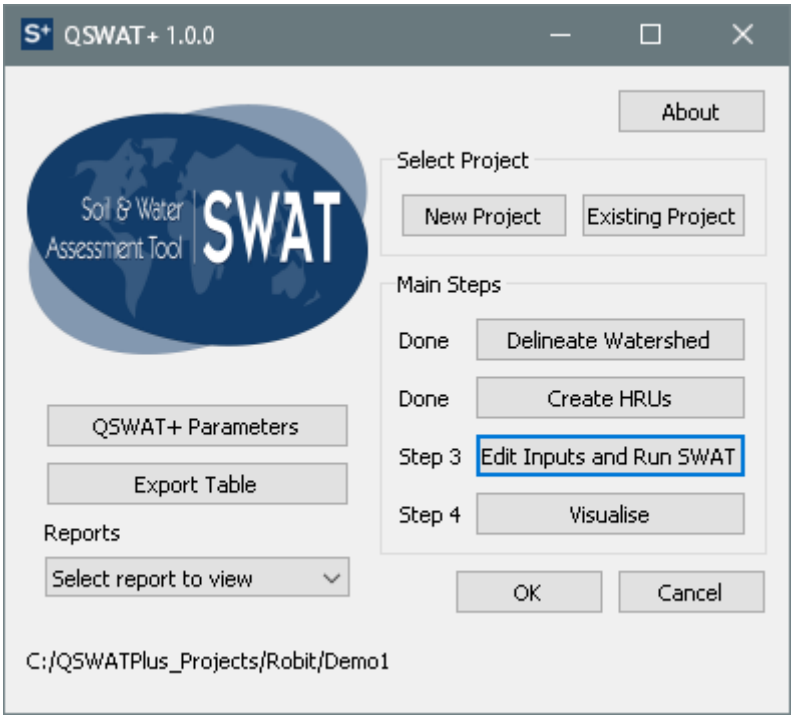


Figure 26: Main QSWAT+ form showing Visualisation button

8.1 Running visualise

Click **Visualise** on the main QSWAT+ form to bring up the **Visualise Results** form (Figure 27). Here the **Choose scenario** pull-down menu shows the simulations, or scenarios, that are available, currently only the **Default** one. Select the one that you wish to display. Then use the **Choose SWAT+ output table** to see what tables are available. This depends on what you chose to print in the SWAT+ Editor, and for HRU results on whether a FullHRUs shapefile was generated[3].

8.2 Visualisation period

The start and finish dates shown are initially the start and finish dates of output from the *Default* scenario. Note that the initial year is 1993 since we simulated from 1990 but set *nyskip* to 3. You may choose dates within these if you wish, to shorten the period being visualised. If other scenarios are selected, the dates will be changed if necessary to ensure there is data available for the whole period.

8.3 Static visualisation

There are three possibilities shown by the tabs for **Static data, Animation** and **Plot**. For static data a single summary value is calculated for each aquifer, channel, HRU or LSU and displayed as a map. For animation there is an animated display of the value in each aquifer, channel, HRU or LSU at each time step in the animation. Plot supports the creation and display of plots of selected variables. This section deals with static data: animation and

[3] HRU tables (those with names starting *'hru_')* are available for plotting, but not for static visualisation or animation unless a FullHRUs file was generated. Rather than repeating this warning many times, we assume in the rest of this section that the FullHRUs shapefile was generated.

plotting are described in the following sections. Make sure the **Static data** tab is selected.

Static results are created by generating a shapefile, and by default this will be called **<table>results.shp** (e.g. *lsunit_pw_monresults.shp*) and stored in the **Results** directory of the current scenario. You can leave this default, or change the file name and location as you wish. You may notice files called hrus.shp, lsus.shp, rivs.shp, subs.shp, aquifers.shp and deep_aquifers.shp in this directory. Do not choose any of these as your file name: the first four are copies of the actual HRUs, LSUs, channels and subbasin shapefiles generated by QSWAT+, copied into this folder and used as a basis for the results shapefile (and for animation). The aquifer shapefiles were created for the same purpose. Copying them to or creating them in the Results directory is done because these shapefiles may be changed by subsequent runs of QSWAT+ and not have the same HRUs, LSUs, channels or subbasins as this run had.

Select *lsunit_pw_mon* as the SWAT+ output table.

The pull-down menu in **Choose variables** contains all the output variables from your chosen table. You can select these one at a time and use the **Add** button to add them to the box below, or simply click **All** to get all of them. You can adjust your selection by selecting and using **Del** to delete ones, or using **Clear** to delete all of them. Data from all the variables you place in the box will be placed in the results file. You then need to select one of them to be displayed. It is a good idea in general to select **All** as this allows you to display different variables without having to change the results file.

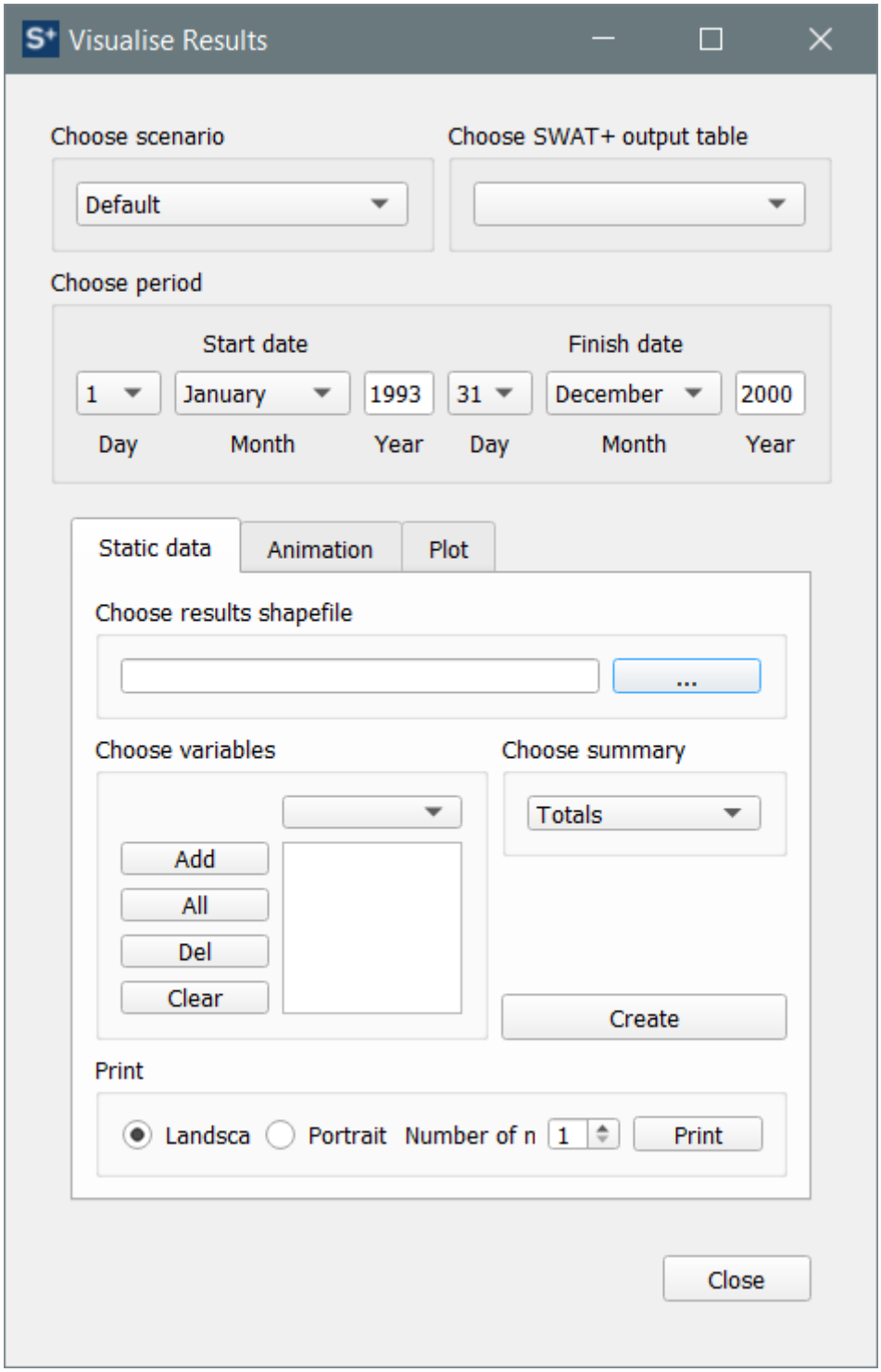


Figure 27: Visualise results form

For each variable, for each aquifer, channel, LSU or HRU the output data can have a value for each day, month, or year of your simulation, or an average year for the whole simulation (according to what outputs you chose to print). To display the results graphically we need to produce a single value for each variable for each subbasin, and for this we choose a **Summary**. Summaries available are:

- **Totals**: the summary is the total of the values for the variable for the aquifer, channel, LSU or HRU..

- **Daily means**: the summary is the daily mean of the total, the total divided by the number of days in the simulation.
- **Monthly means**: the summary is the monthly mean of the total, the total divided by the number of months in the simulation. The number of months may be fractional
- **Annual means**: the summary is the annual mean of the total, the total divided by the number of years in the simulation. The number of years may be fractional.
- **Maxima**: the summary is the maximum value for the variable for the aquifer, channel, LSU or HRU..
- **Minima**: the summary is the minimum value for the variable for the aquifer, channel, LSU or HRU.

If the output table is an annual average (its name ends in '_aa') the summary method is already chosen as annual mean, and the **Choose summary** menu is not enabled..

For variables which are effectively means, such as rates of flow, only some of the summaries make physical sense, namely the mean for the chosen printout frequency, maximum, and minimum. Users should therefore exercise caution in choosing summaries for such variables.

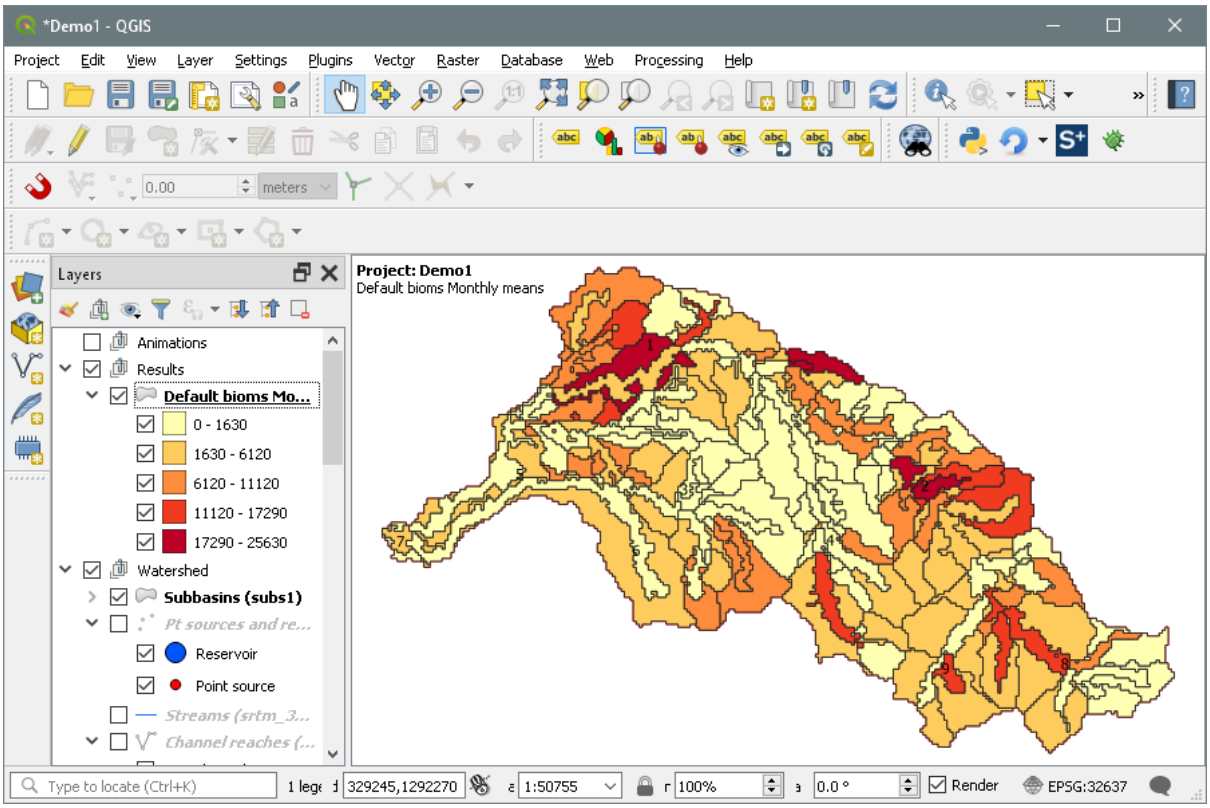


Figure 28: Displaying static results

Figure 28 shows the display after selecting table *lsunit_pw_mon*, variable *bioms*, and monthly means as the summary method, and clicking **Create.** You will be asked to confirm if you are overwriting an existing shapefile The text in the map canvas shows that we are displaying **bioms** monthly mean values from the Default simulation.

The results layer is given a default colouring using 5 intervals chosen by the Jenks natural breaks algorithm to cover the range of summary values for the variable. You can change the

colouring scheme if you wish (right click on the layer entry in the **Layers** panel, select **Properties** and then select the **Symbology** tab).

To display a different variable from the list, select it with the mouse and click **Create**. To change the summary, select the new one from the pull-down menu and click **Create**.

We chose an LSU results table, so it was displayed by colouring LSU shapes. HRU output colours HRU shapes. Results from channels are displayed using channels.. The thickness of the channel shapes is proportional to the channel width.

Static results files are supplied with map tool tips showing the channel, LSU, HRU or aquifer number, so you can check the number of any feature. Make sure that the **Show Map Tips** button is pressed (Figure 32), and that the results layer is selected in the layers panel, and when the mouse hovers over a feature its channel, LSU, HRU or aquifer number is displayed.

8.4 Combining results

You might want to display an expression that is a combination of result values, or some function of an existing result. This can be achieved as follows.

First, generate a results shapefile as described above, making sure that you include in its variables at least all that you want to use in your expression. Then right click on its entry in the **Layers** panel and select **Open Attribute Table.** Click on the yellow pencil symbol at top left to start editing, and then select **Open field calculator** (abacus symbol).

In the **Field Calculator** make sure **Create a new field** is checked, give it a name (at most 10 characters and not already a field name), choose **Decimal number (real)** as the field type, change the length and precision if you wish, and then create the expression. The easiest way to do this is to double click on entries in the centre panel: **Fields and Values** contains all the existing field names, **Math** contains mathematical functions, **Operators** a variety of numerical and logical operators (some of which are also available as buttons above the left panel), and so on.

As you double click on items they are added to the left panel. You can also edit this panel directly (so you can easily delete mistakes). When the expression is valid the resulting value from the first row of the table is exhibited below.

When you are happy with your expression, click **OK** to close the calculator and the yellow pencil to finish editing. Confirm that you want to save the changes to the layer.

You will find the new field name(s) added to the variables that can be selected for display. Select one, change the summary method if you wish, and click **Create** to see your new variable displayed.

Note that this method adds data to the attribute table of the current results file: it does not add to the current table from the output database. This means it cannot, for example, be animated or plotted. To add such data to the original table it is often possible to use an SQL statement: SQL has a powerful expression language. For example, we could use SQLiteStudio to add a column *flo_diff*, the difference between *flo_out* and *flo_in* to *channel_sd_mon*. We proceed as follows:

- Open *channel_sd_mon* and select the *Structure* tab.

- Use the **Add column** button to add a column called *flo_diff*, type *REAL*
- Click the green tick to confirm the addition
- In the **Tools** menu select **Open SQL Editor,** and type in the command

  ```
  UPDATE channel_sd_mon SET flo_diff = flo_out - flo_in;
  ```
  Click the blue **Query execute** button.

- In the visualise form select a different output table and then select *channel_sd_mon* again to force the table to be reread and the variable made available in the static, animation, and plot pull-down menus.

Another method to exploit the power of spreadsheets would be to export the table as a csv file (easily done in SQLiteStudio), import the csv file into a spreadsheet tool like Excel or Open Office, add the column(s) and generate their data using the functions available in the spreadsheet tool, edit the table structure in SQLiteStudio to add the necessary column(s), and import the csv file back into the table

8.5 Combining tables

The previous section 8.4 presents a method to produce results values based on functions of original values, but requires the original values to be in the same table. But SWAT outputs are grouped in a range of tables: what if a function of values from different tables are required? We explain in this section how this can be done.

The tables that can be visualised generally have names in the form *X_Y_Z*[4] where *X* is *lsunit*, *hru*, or *channel*, and *Z* indicates the frequency: *day* for daily, *mon* for monthly, *yr* for yearly, and *aa* for average annual. If two tables in the same output database have common *X* and *Y* values then they can easily be combined in a *natural join*. They will automatically share the same *id*, *jday*, *mon*, *day*, *yr*, *unit*, *gis_id*, and *name* columns, so these will appear in the join, and the remaining columns will simply be the union of all the other columns in the two tables. For example, if you use SQLiteStudio, then you can open the output database, in the **Tools** menu select **Open SQL Editor,** and type in a command like

```
CREATE TABLE lsunit_lswb_mon AS SELECT * FROM lsunit_ls_mon NATURAL JOIN
lsunit_wb_mon;
```

and click the **Execute query** button.

Note the new table name *lsunit_lswb_mon* preserves the *X* and *Z* values from the two input tables, so that the visualise tool knows how to present the data from the new table. Now reselect the scenario (even if there is only *Default*), the new table will appear in the output tables menu, and you can proceed as in section 8.4 above.

This only illustrates a simple case, and you are free to create new tables from old by whatever means you like. You might, for example, want to join tables from different scenarios, if the LSUs, HRUs and channels are the same, and the periods of the runs are the same (or you can prune them to produce the same common period).

The table below shows how the data from tables is collected for different frequencies, so that you can see what is needed in any table you generate:

[4] *aquifer_* is currently the only exception to the *X_Y_* pattern.

Frequency	Key	Order by
Daily	gis_id	yr, jday
Monthly	gis_id	yr, mon
Yearly	gis_id	yr
Average annual	gis_id	yr[5]

The table shows, for example, that to extract data for variable *x* from a monthly table *X_Y_mon* for feature *f* the SQL statement used by visualise will have the form

```
SELECT x from X_Y_mon WHERE gis_id=f ORDER BY yr, mon;
```

And hence that the columns needed in a new monthly table are at least *x, gis_id, yr,* and *mon*. In addition, of course, the new table's name must follow the *X_Y_Z*[6] pattern described above.

8.6 Printing results

QSWAT+ uses the QGIS Print Composer to support the printing of results. Figure 29 shows the result of adding two more static results files to the display (*sedorgn* from table *lsunit_ls_aa* and maxima *peakr* from *channel_sdmorph_mon*), selecting **Portrait** as the orientation and 3 as the number of maps, and clicking **Print**. The print composer is started and shows the first 3 (in this case) maps in the **Results** group in the layers panel, regardless of their current visibility. (It is normal to have only one such map visible at a time in the map canvas as they are slightly transparent to allow the subbasin, LSUs and channels to remain visible.) Visible layers in the **Watershed** group are included as background.

The items in the print composer can be edited. You may want to change the title from its default of **SWAT+ results: <project>**, to edit the **Company/Author** box appropriately, change the map scales, adjust their positions, add other items, etc. You can then export it as an image, pdf or svg file. To scale several maps, select the **Item Properties** tab, click on one map item, which forms a box round it. Drag a corner of the box to change the scale. Note the scale number. Then change the others to the same scale, this time by editing the number and pressing **Enter** for the change to take effect. The scale bar is attached to the top left map, so it changes with it. You may also need to use the **Position and size** settings further down under **Item properties** to keep the maps aligned.

When you have finished with a layout, maybe by exporting it as a pdf, closed it and don't want to revisit it, it is a good idea to delete it from the QGIS Layout Manager, since QGIS seems to be sensitive to having more than a very few layouts. Open the Layout Manager (using **Project -> Layout Manager**), select the layout in the manager panel, and click **Remove.** Or, if you have closed but not removed a layout, you can open it again using the Layout Manager's **Show** button.

[5] not strictly necessary but it simplifies the code
[6] or *aquifer_Z*

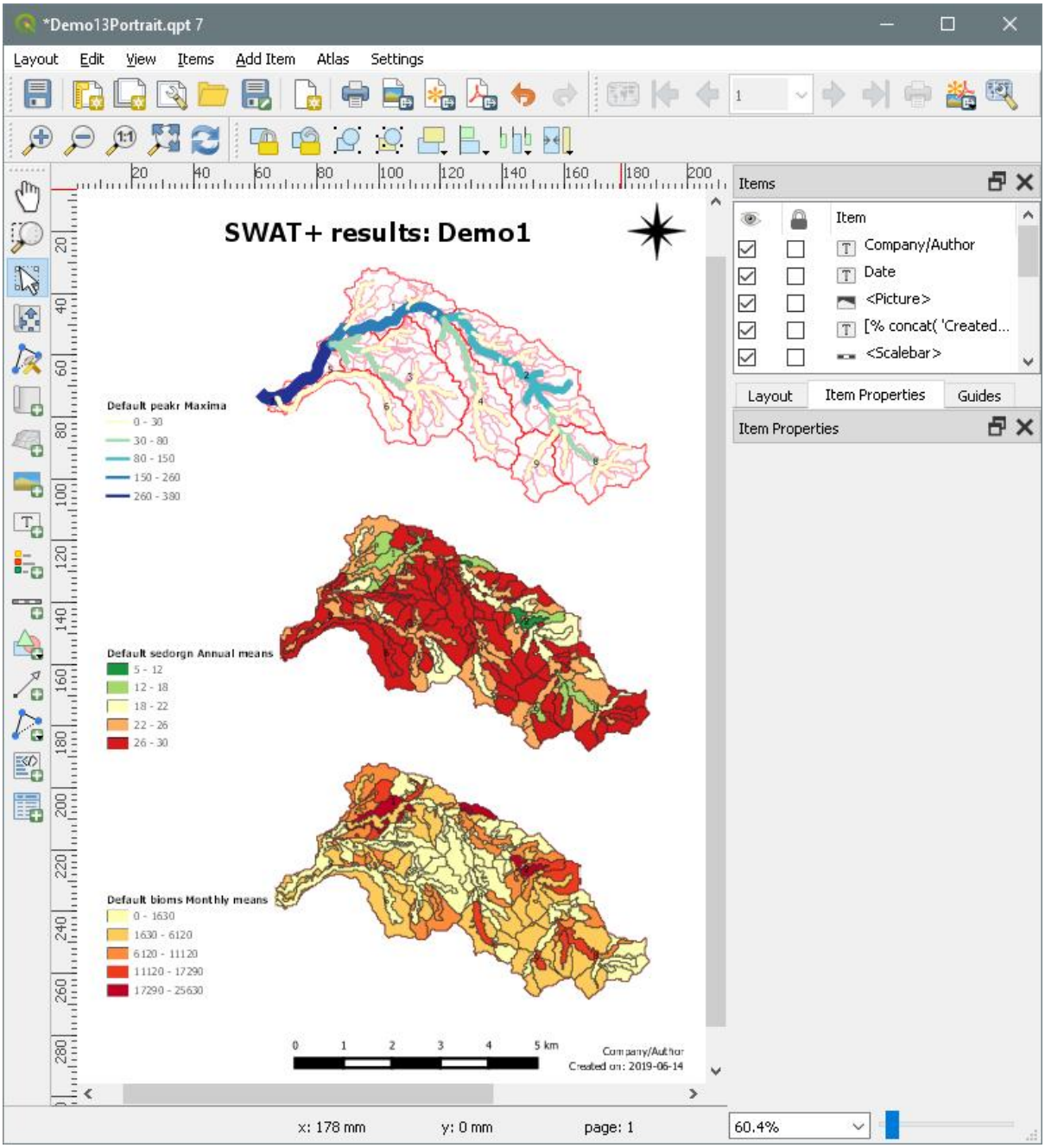


Figure 29: Printing results

8.7 Animation

To animate, after choosing the scenario and the SWAT output table (e.g. *channel_sdmorph_mon*), select the **Animation** tab. The **Variable** pull-down menu contains all the variables from the chosen output table (Figure 30): select one of these (e.g. *flo_out*). A new animation layer is created in the map canvas, showing the initial values of this variable in the simulation (Figure 32). Note that the range of possible values runs from 0 to 62 (the minimum and maximum values for *flo_out* for the whole simulation period). You can view the actual initial values for each channel by examining the attribute table (right click on the **Default flo_out** entry in the **Layers** panel and **Open Attribute Table**).

There are controls to run, pause and rewind the simulation: it steps through daily, monthly or annual time steps according to the chosen table. The speed of the animation can be adjusted

using the **Speed** spinbox. You can also investigate the changing values manually, by dragging the slider, or (with the **Visualise Results** form the currently selected window**)** by pressing the left and right arrows on your keyboard. The date (day, month, or year) of each step appears above the slider.

The animation is achieved by creating a shapefile, in this case *rivs0.shp* in the *Animation* directory within the *Results* directory of the scenario. At each time step the values of the variable are updated and the file redisplayed. Hence this file is being continuously changed. It is regarded as a temporary file and so is not preserved. If you wish to save a view of the animation at a particular moment you can pause the animation and make a copy of this file (remembering to copy the .dbf, .prj, and .shx files as well as the .shp file).

It is also possible to make an animated gif of the animation. Click the green **Start recording** button before running the animation, and then the red **Stop recording** at any point. You will be informed (after a pause while the file is created) that an animated gif has been created (called *<variable>Video.gif* in the *Results* directory). Click **OK** to accept the message, and then click **Play** to see it run in whatever is the default player for animated gifs on your machine. Animated gifs can also be played in web browsers. The speed of the animated gif is determined by the value in the speed spinbox when the **Stop recording** button is pressed, so you may run it at one speed, wait for it to finish or press pause, and then change the speed before you click **Stop recording.**

The animated gif is made by saving a .png file at each time step in the *Results/Animation/Png* directory of the Default scenario. These files are deleted when **Stop recording** is clicked, so if you want to use them, perhaps to make your own animation, you need to copy them before you stop recording.

We chose to start with the defaults of a **New** animation in the **Map canvas.** If you keep these defaults and select a new animation, e.g. table *lsunit_wb_mon* and variable *surq_gen* then a second animation will be added to the map canvas. Choosing **Current** would instead cause the second animation to replace the first.

Another possibility if you want to watch several animations in parallel is to use the print composer as the **Animation mode.** This may be recorded if desired. Select **Print composer**, choose **Portrait** or **Landscape**, and change the number of maps if desired, and click **Start recording**. An information form is shown, explaining that the print composer will open with the initial layout, which you may edit if you wish (to change the title, author, etc.). Then, if you have changed anything, you should **Save as Template,** using *AnimationTemplate.qpt*, in the *Results* directory as the template file, else you can simply close the composer**.** After doing this, and saving or closing the composer, start the animation. It runs slowly as it creates a layout and saves it as a .png file at each time step.

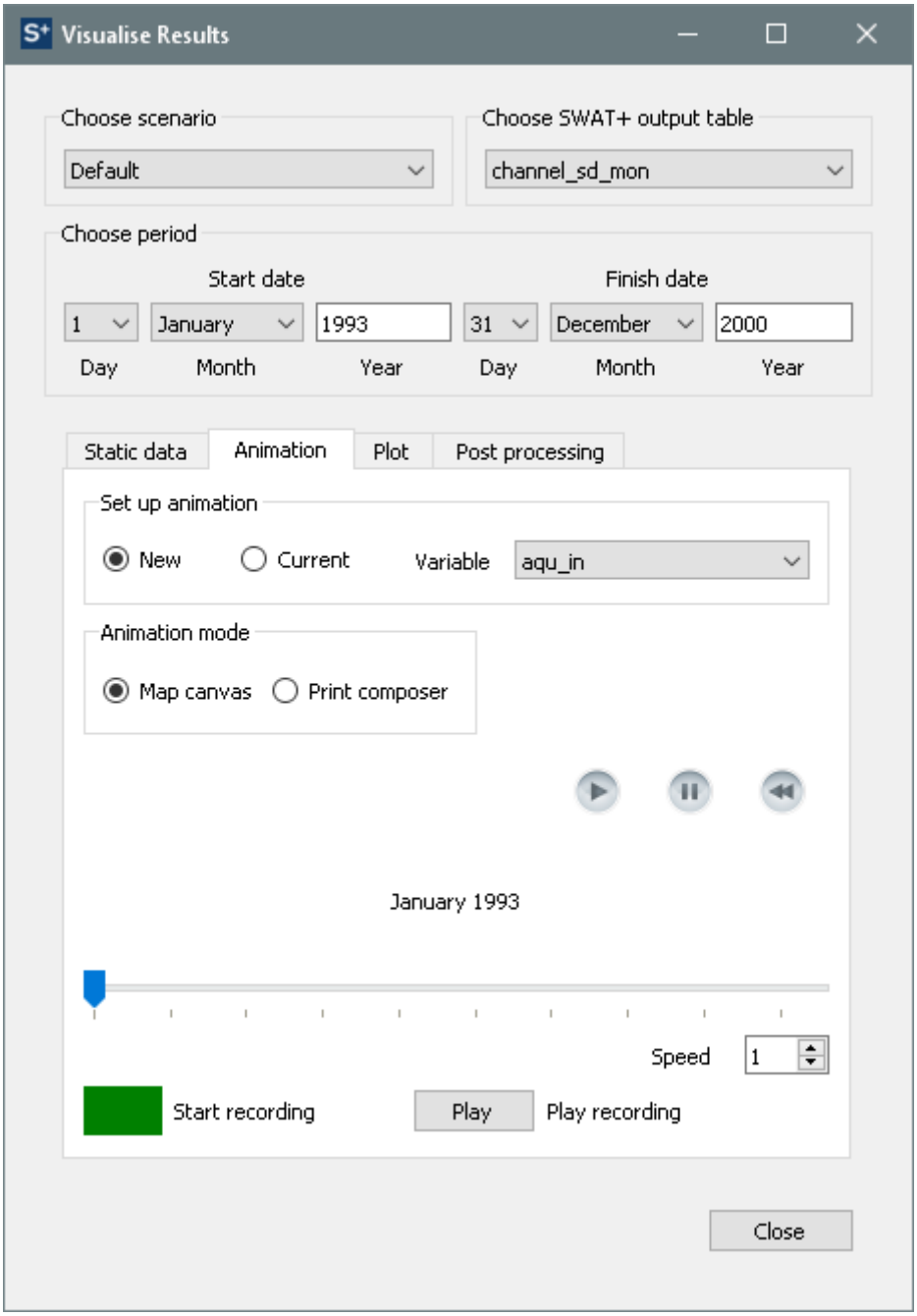


Figure 30: Choosing an animation variable

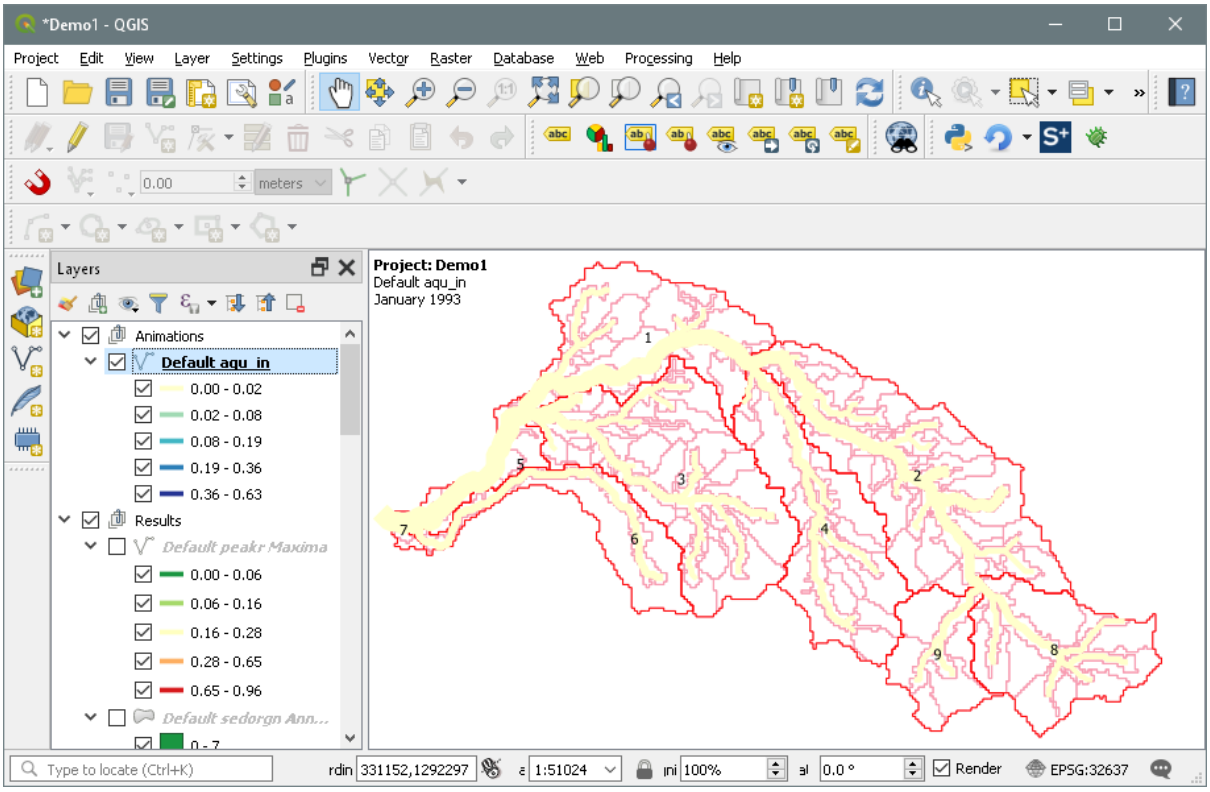


Figure 31: Start animation

As with static visualisation, it is possible to change the colouring scheme used (the colour ramp and/or the number of classes): right click on the legend entry, select **Properties** and then select **Style**.

8.8 Plotting

Plotting allows the rapid display of output data as line graphs or bar charts, plotting values against time. It is intended to facilitate comparisons of outputs from different scenarios, or different aquifers, channels, LSUs or HRUs, or between simulated and observed values.

Plotting is done in two stages, data collection and display. After collection, just before display, data is saved in a comma-separated value (.csv) file. This file can be loaded into tools like Excel to support more extensive analysis than provided by the plotting tool SWATGraph that is supplied with QSWAT+.

8.8.1 Collecting plot data

As with other visualisation options, start by selecting a scenario and SWAT+ output table. Then reduce the period if you wish. Next you can select an observed data file if you have one, and we have *observedFlow.csv* in the *Robit/Observed* directory (Figure 33). The structure of observed data files is described later.

As many plots as you like may be added to the table. You can start by making sure an output

table is selected, e.g. *channel_sdmorph_mon*, and clicking **Add plot**. This adds a row to the table, and already shows the chosen scenario and table. The **Unit** field is empty and so is the **Variable**. This first row is also the **currently selected** row, and this is important. Whenever we change the scenario, output table, unit or variable settings through their pull-down menus we will immediately change values in the currently selected row. When you have more than one row you can select a row by clicking in it.

Suppose we want to compare the main monthly outlet flows from two scenarios. Then we could choose for each scenario the table *channel_sdmorph_mon*. We need to find the channel number of the outlet channel. There are several ways of doing this.

1. When we use the **Unit** pull down menu, as each potential choice is highlighted when we move the mouse over it, the corresponding channel, LSU, or HRU is highlighted in the map canvas. This is a good way to check we have the right item if we think we know the unit number, but a rather slow method to find a unit by location, like the watershed outlet channel.
2. We make sure that the **Show Map Tips** QGIS button (Figure 32) is pressed, select the appropriate map in the layer panel (rivs1, gridstreams or drainstreams for channels; Inlets/outlets for points marked in the inlets/outlets file; Pt sources and reservoirs for point sources and reservoirs added by QSWAT+; lsus2 or lsus for LSUs; hrus2 or hrus for HRUs; aquifers for aquifers) and hover the mouse over the feature. A *map tool tip* will be displayed showing the unit type (e,g, **Channel**) and number.
3. We select the appropriate map in the layers panel, as in the previous method 2. We click the **Identify Features** QGIS button (Figure 32), and click on the intended feature, such as the outlet channel (Figure 33). We see from the **Identify Results** panel that the channel number is 1[7], so we select 1 as the **Unit**.

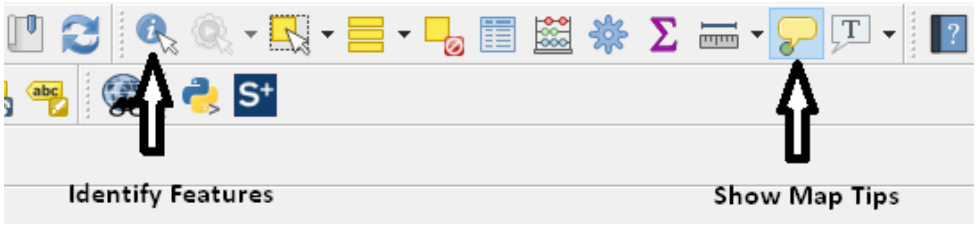


Figure 32: Identify Features and Show Map Tips buttons

We select *flo_out* as the **Variable**. Now we want to add a second row to the table for the other scenario. A convenient way to do this is the **Copy plot** button. This copies the current row to a new row, and makes it current. Then all we have to do is choose a new scenario and the second row will be what we want.

[7] This number may be different on different machines, or with change in the number of MPI processes.

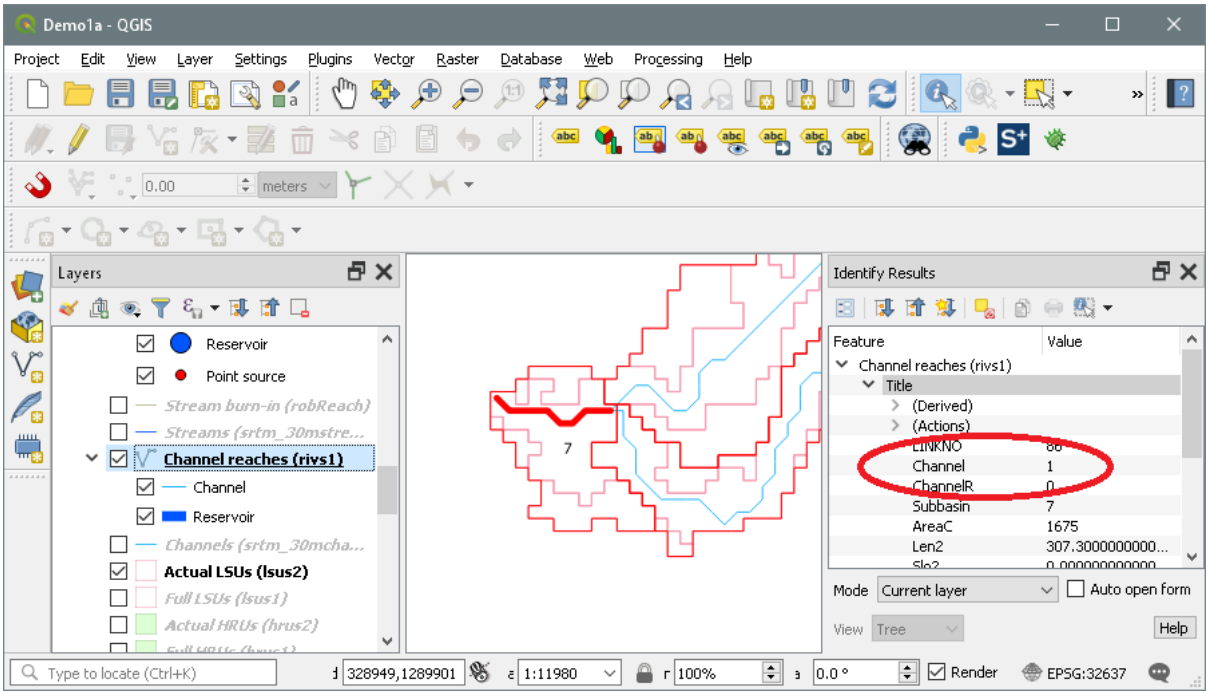


Figure 33: Identifying the outlet channel

It is also possible to move the current row up or down: this just affects the ordering of the plots in the output.

To add observed data we select an observed data file, and use the button **Add observed**. *observed* is used as the scenario name, '-' is used for *Table*, and *Unit*, and we can select a variable name from the file.

We use the observed data file *observedFlow.csv* from *Robit/Observed.* There is only one variable, in this file, *Flow*, so it is already selected in the *observed* row.

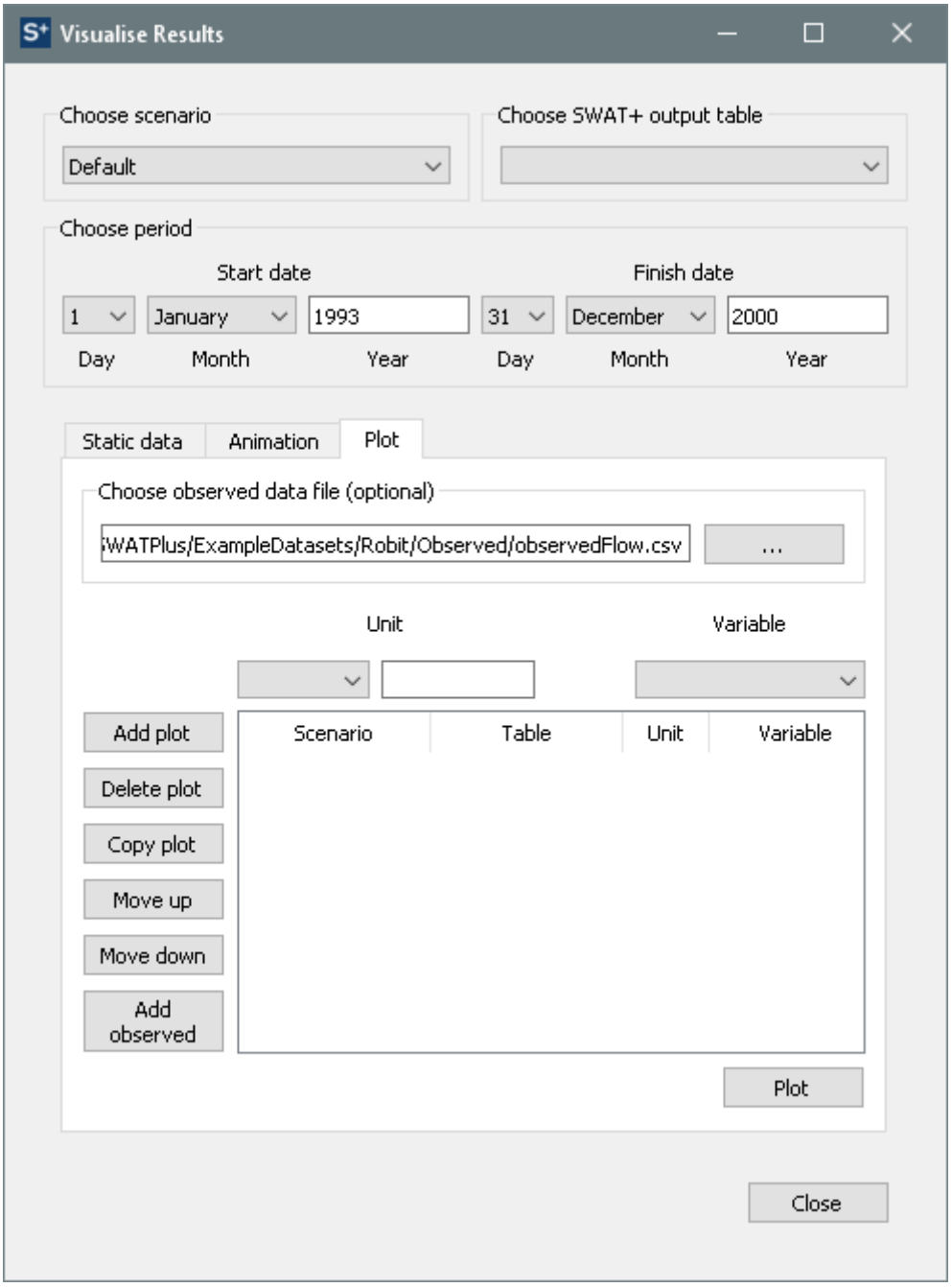


Figure 34: Plot screen

When we have completed the plot selections we click **Plot**. This invites us first to choose a *.csv* file to save the data. We can if we wish later import this into, for example, Excel if we wish to do further manipulations or use Excel's graphing tools.

If we do this for this example, we immediately get a message that there is missing data for 1998/1 (i.e. January, as we are using monthly data) for plot 2, which is the observed plot. The observed data only runs from the beginning of 1993 to the end of 1997. We click **OK** to close the message, change the finish year to 1997, and click **Plot** again. We can choose the same *.csv* file, and tool checks we are happy to overwrite the file. We click **Yes.**

Once the data is written to the *.csv* file, SWATGraph is automatically started to display the data

graphically: see Figure 35. The data has been displayed as line graphs. We can change the display to bar chart using the **Chart Type** pull-down box and clicking **Update Graph**. Figure 34 shows a plot of the flow out of channel 1 against the observed data file *observedFlow.csv* supplied with the Robit example. **.**

Other SWATGraph options include importing another (similarly created) *.csv* file, using **New File to Plot**, and using the buttons immediately below the graph. These include zooming and panning the graph, saving the graph in a variety of image formats (the rightmost button) and a number of editing options (rightmost button but one). You can add a title, add a label to the vertical axis, change the colours of the plots, and (for line graphs only) change the vertical axis from linear to log scale, which is useful when there is a wide variation in values.

The data from the *.csv* file is also displayed. The column headings, also shown in the graph legend, take the form **Scenario-Table-Unit-Variable**, where **Unit** is the aquifer, channel, LSU or HRU number as appropriate for the table. For an observed plot the form is **observed-Variable**.

At the bottom of the screen some statistics are displayed. These are correlation coefficients for each pair of plots, and Nash coefficients for each pair of plots where one is observed. Some of these may be meaningless, of course, if for example you choose to plot four things that are really two pairs, but it is quicker as well as simpler to calculate them all than it is to ask the users what they want.

SWATGraph may also be run as a stand-alone tool. Double click on *runSWATGraph.bat* to start it, or use this as the command in a command shell. In the latter case you can optionally add the path to a .csv file to plot the csv file automatically at the start. *runSWATGraph.bat* is in *C:/SWAT/SWATPlus/Tools/SWATGraph* if you installed it with QSWAT+, else, if you had it from QSWAT, in *C:/SWAT/SWATEditor*.

8.8.2 How compatible do different scenarios need to be?

The period is automatically adjusted to include all the scenarios involved (but see below for observed data).

You also need to be reasonably confident that the things you want to compare are indeed comparable, so if you are comparing HRUs, for example, they should be the same HRU in each scenario. But if you are comparing the main outlet channel in each scenario, which gives figures for the whole watershed, it doesn't matter how the subbasins, LSUs and HRUs in each were selected, or how the units are numbered, as long as you select the main outlet channel number for each.

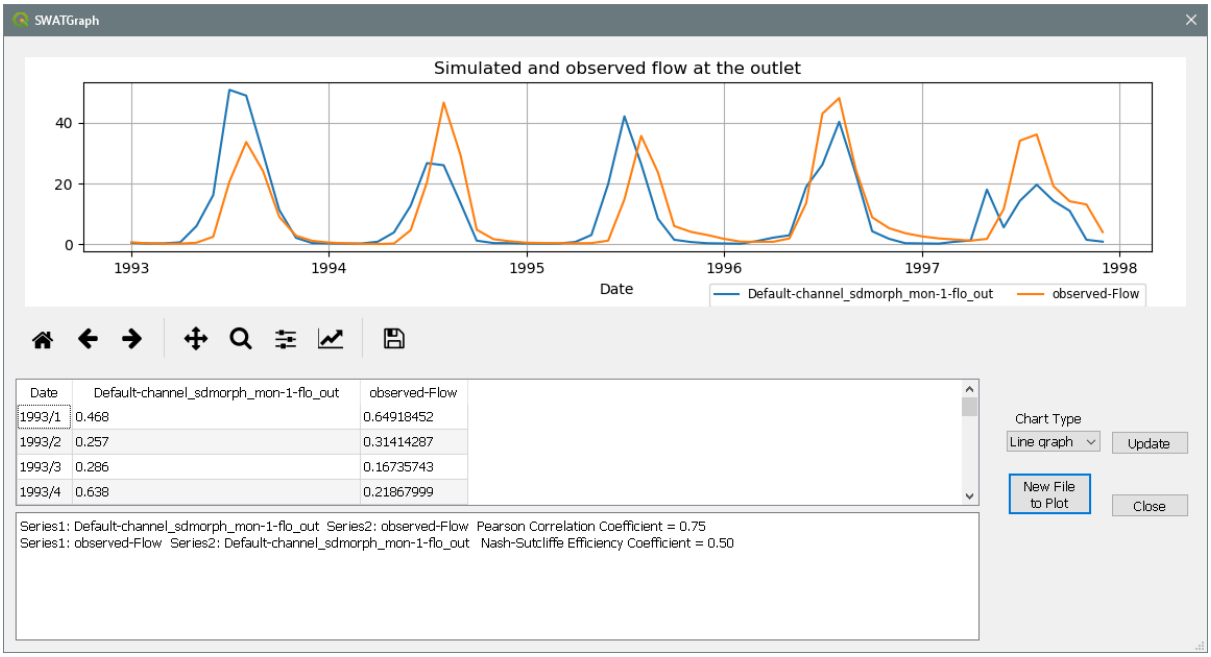


Figure 35: SWATGraph

8.8.3 Including observed data

To include observed data you need to prepare a *.csv* file, i.e. a text file where each line consists of the same number of items (one or more), separated by commas.

The first line must be comma-separated text strings, which will be used as the variables. If the first string is DATE, or Date, or indeed these four letters in any case, it and the data in the first column are ignored. The dates are assumed to be the same as in the SWAT output plots; they are not checked.

The data on lines two to the end (apart from ignored dates) should normally be numeric. If you do not have complete observed data, you should include the relevant dates in the file, since dates are assumed to be sequential, but leave the data fields empty. You still need to insert the same number of commas as in other lines. In fact you can put anything in the fields, like "missing" if you wish: any field that SWATGraph cannot parse as a number will be taken to be missing data.

Dates on which some data is missing are excluded from the Pearson and Nash coefficient calculations (and the number of values used in the calculation is reported). Missing data will appear as breaks in the relevant line graph, and as missing bars (indistinguishable from zero values, of course) in bar charts. You can check where data is missing from the data table, where you will see blank fields, or the text like "missing" that was used in the .csv file.

Table and Unit boxes in an observed plot row will be unavailable, but you need to choose a Variable. The choices in the Variable pull-down box will be the variables (apart from *date*) in the first line of the *.csv* file.

9. Post processing

Post processing of SWAT+ results is available via the **Visualise** form, under the tab **Post processing**.

9.1 Environmental flows

Three environmental flow statistics are available, Qq, dQp and Qb.

9.1.1 Qq

The Qq statistic provides, for each calendar month, a measure of the flows for that month (including for each calendar month the flows that occur in that calendar month regardless of the year) that satisfies the condition that *q*% of the flows exceed the measure. The user chooses the subbasin and the value of *q*. The flow is the flow in m^3/s at the outlet of the subbasin, so if the subbasin immediately above the watershed outlet is chosen then the statistics are for the whole watershed.

Calculation of Qq depends on data in the *channel_sd_day* table in the outputs database, so this must have been be selected amongst the outputs to be printed by the SWAT+ Editor.

The Qq form initially selects 95 as the percentage *q* and the outlet subbasin for the watershed (7 for the Robit example), as illustrated in Figure 36. (If there is more than one watershed outlet in the model, the one chosen is arbitrary.) The user can change the percentage and subbasin, and also change the period if desired, before clicking **Calculate**. The result for each month is then written to the table. The results can be saved to a text file using the **Save** button. If the **Append** box is checked the results will be appended to the selected file, allowing for results for different parameter settings to be saved together.

9.1.2 dQp

The dQp statistic provides a single flow statistic for a given subbasin, where *d* is the number of days for moving average, *p* is a threshold percentage. The initial form is shown in Figure 37. 21 is the default for *d*, 95 for *q*, and the watershed outlet subbasin for the subbasin. Users also need to choose a start month and can choose between *Percentile* and *Mean* as the final statistic selection. The statistic is a flow in m^3/s.

Calculation of dQp depends on data in the *channel_sd_day* table in the outputs database, so this must have been be selected amongst the outputs to be printed by the SWAT+ Editor.

The calculation of dQp is as follows:

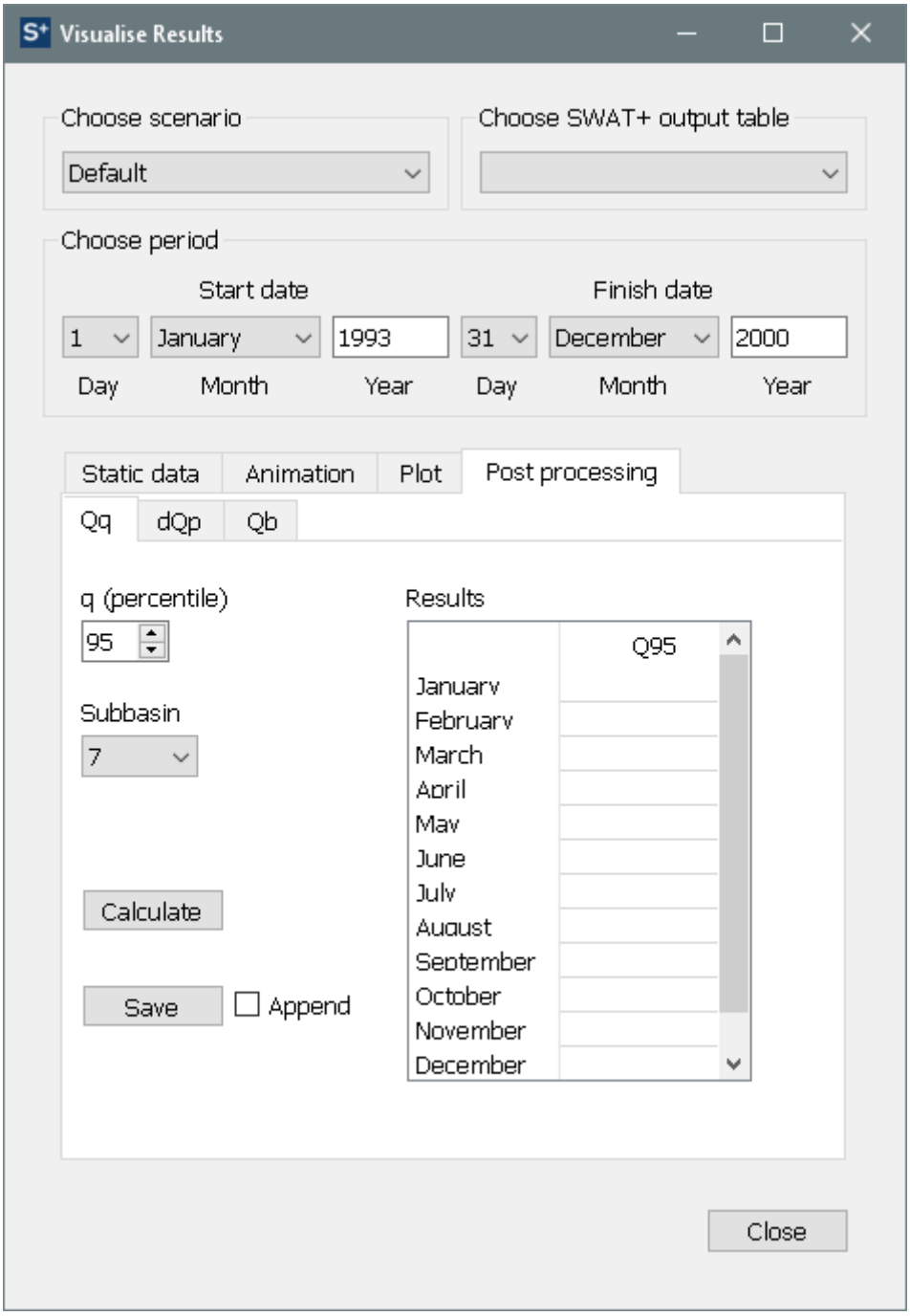


Figure 36: Initial Qq form

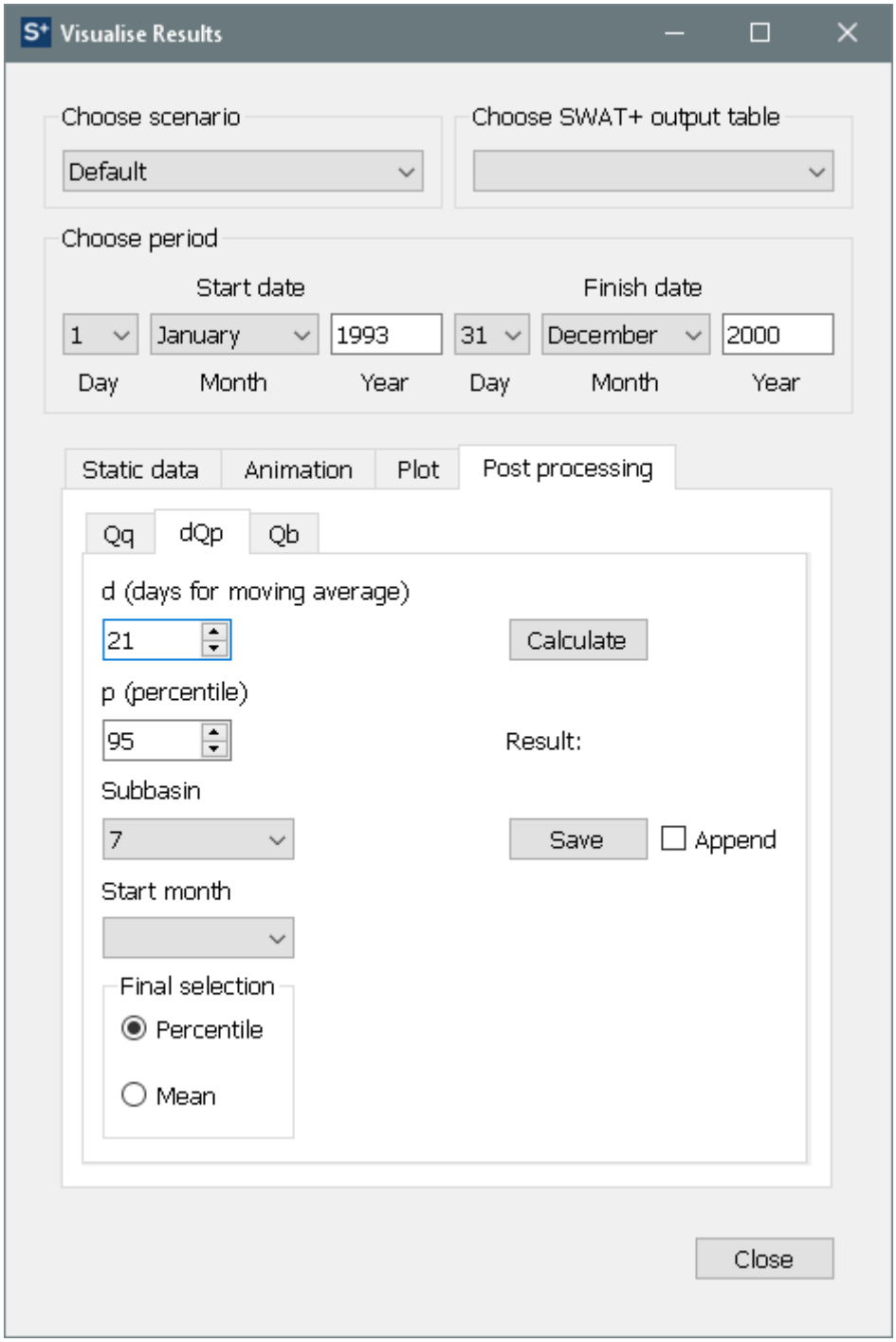


Figure 37: Initial dQp form

- The chosen period is divided into successive year-long periods, starting with the first day of the start month and finishing with the last day of the preceding month one year later. Days before the first start month are ignored, as are days at the end that do not make a complete year. For example, if the chosen period is from the start of 1993 to the end of 2000 (so is 8 years long) and the selected start month is April, then the division will give 7 year-long periods, the first running from 1 April 1993 to 31 March 1991, and the last running from 1 April 1999 to 31 March 2000. Common start month choices are January (natural), October (hydrogeological) and April (hydrobiological). The month should be chosen so that it is not in a low flow period.

- Moving averages of length *d* days are calculated for the daily flows for each year period, and for each year period the minimum moving average is selected as the flow for that year.
- There are then two possible methods for selecting the final statistic from the flows for the year periods:
    - **Percentile**: The result is the flow value which is exceeded by *p%* of the flows for the year periods. For the default settings, the result would be called 21Q95.
    - **Mean**: The result is the mean of the flows for the year periods. For the default settings the result would be called 21Qm. Note that the value of *p* is ignored.

  The choice between these methods is made by selecting the appropriate radio button in the *Final selection* group box.

After setting the parameters, the result is calculated and displayed in the form by clicking **Calculate**. The result can be saved to a text file using the **Save** button. If the **Append** box is checked the result will be appended to the selected file, allowing for results for different parameter settings to be saved together.

9.1.3 Qb

The Qb (*Basic Flow)* statistic provides for a given subbasin an annual statistic and also monthly statistics. The initial form is shown in. Figure 38. The watershed outlet subbasin is the default subbasin. Users also need to choose a start month. The annual and monthly statistics are flows in m^3/s.

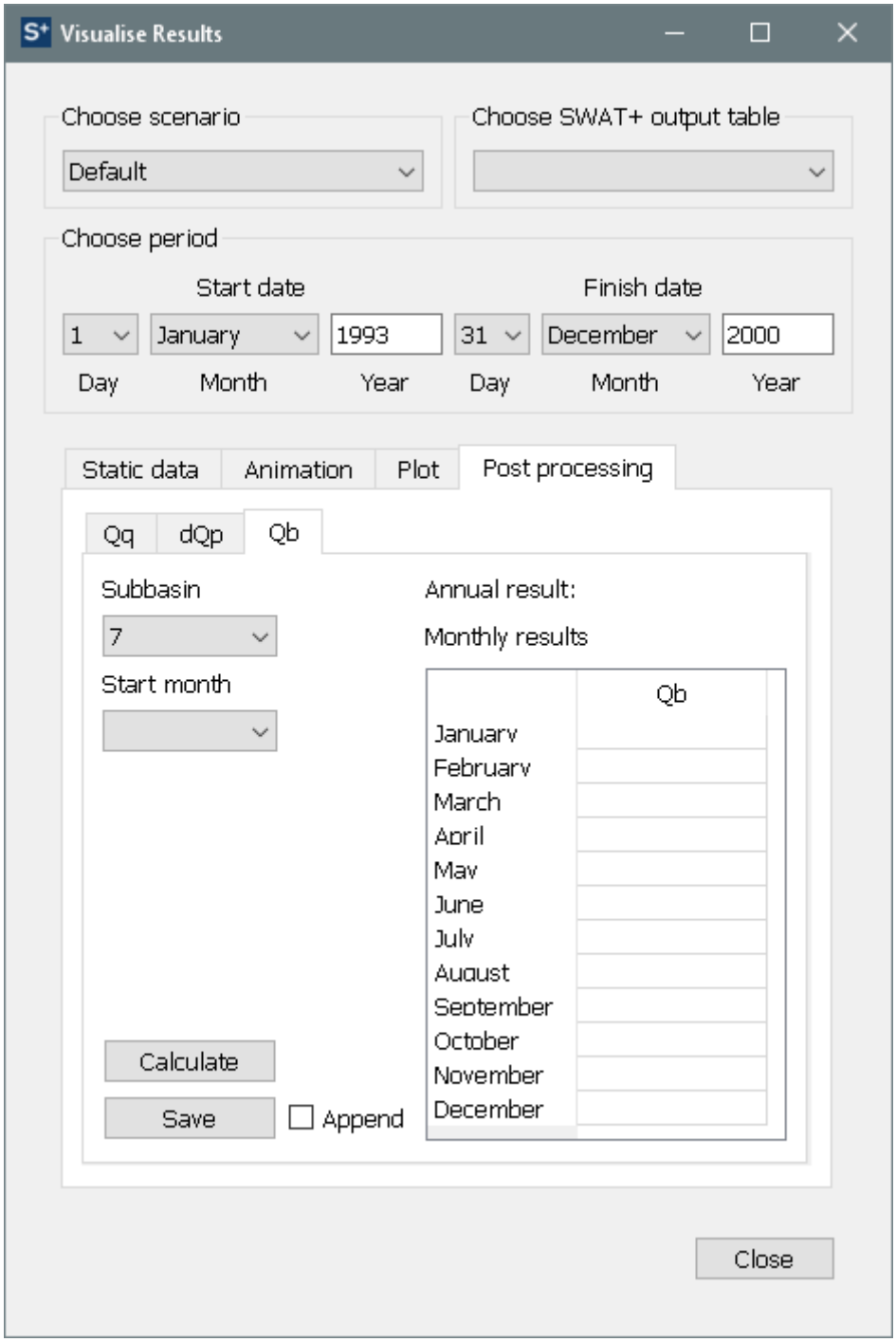


Figure 38: Initial Qb form

Calculation of Qb depends on data in the *channel_sd_day* table in the outputs database, so this must have been be selected amongst the outputs to be printed by the SWAT+ Editor.

The calculation of Qb is as follows:

- The chosen period is divided into year-long periods according to the start month in the same way as for dQb: see subsection 9.1.2 for details.
- For each year period, moving averages of outflows over periods of length 1 to 100 days are calculated, followed by selecting the minimum moving average for each period length. Then the rate of change of the minimum moving average is calculated, comparing the minimum moving average for period length *i* to that for period length

i-1, 1<i≤100. If period length *j* gives the highest rate of change then the corresponding minimum moving average for period length *j* is selected as the flow for that year period.

- The annual result is the mean of the results for the year periods.
  For each month, the monthly statistic is calculated by multiplying the annual result by the variation factor for the month. To calculate the variation factors we calculate the mean daily flow for days falling in each calendar month, and note the minimum of these means. The variation factor for a month is then the square root of the ratio of mean for that month to the minimum mean.

After setting the parameters, the result is calculated and displayed in the form by clicking **Calculate**. The result can be saved to a text file using the **Save** button. If the **Append** box is checked the result will be appended to the selected file, allowing for results for different parameter settings to be saved together.

10. Using grid models

As well as using watershed divided into subbasins and (optionally) landscape units, QSWAT+ supports division of the watershed into square grid cells. The grid cells are treated by SWAT+ as small subbasins, and they may be divided into landscape units and HRUs in the same way. Grids allow in particular for finer control over drainage patterns.

10.1 Creating a grid

You can create your own grid using watershed delineation as described earlier for the Robit watershed, but before clicking the **Create watershed** button you need to check the box **Make grid** and set the **Grid size** that you want in the spin box. Grid sizes are defined in terms of DEM pixels. So, for example, for the Robit example we have a DEM whose pixels are 30 x 30 metres: you can see the pixel size after loading the DEM by selecting the **DEM properties** tab in the delineation form. We will choose a grid size of 3, so that our grid cells are 90 x 90 metres (Figure 39).

A couple of things have changed in the form as a result of choosing to create a grid. First, there is no channel threshold to set. Grids cells are in many ways treated like subbasins, but will be small, so there is no reason to allow multiple channels within them. Effectively channels are the same as streams. Second, the option to merge subbasins is not available. (The tab is still there, but its contents are not enabled.) Grid models are designed to be homogeneous in grid cell size, so it makes no sense to merge them. Otherwise, things are much the same as the non-grid case: inlets and outlets can be defined, lakes can be added, and floodplain maps can be created and used to form upslope and floodplain areas. If we do this, we will find it sometimes the case that grid cells are entirely floodplain or entirely upslope.

The result of clicking **Create watershed** is shown in Figure 40. The watershed delineation has been done as before, and then grid cells created to cover the watershed. There are 2,185 grid cells in the result: you can see this by right clicking on the **Watershed grid** entry in the

layers panel and selection **Show Feature Count**. A new streams layer **Grid streams** has also been created, containing stream reaches to connect grid cells. These reaches are created by connecting the points of maximum flow accumulation in each grid cell, and you should find if you make visible the original stream network calculated by TauDEM (the layer named **Streams)** that they show good consistency (for small grid sizes) The grid streams shapefile has an attribute *Drainage* whose values are the areas in km^2 draining to the outlet of each grid cell, and this is used to set the widths of the stream lines in the map canvas.

After clicking the final **OK** button on the delineation form, creating HRUs proceeds exactly as in the non-grid case. Processing times will typically be longer, because we are dealing with much larger numbers of landscape units or subbasins, and of HRUs. Figure 41 shows the Create HRUs form after we have selected the landuse and soil maps, imported the csv files for landuse and soil lookup and for the usersoil table, and clicked **Read**. There are 2185 subbasins (since each grid cell is a subbasin). 2185 channels (or streams - remember they are the same) and 2502 potential HRUs. When we click **Create HRUs** at this point we will get one HRU per grid cell, since the **Dominant HRU** choice is currently selected. We can have more HRUs by selecting a different filter, but it is clear that most grid cells already have only one HRU in any case.

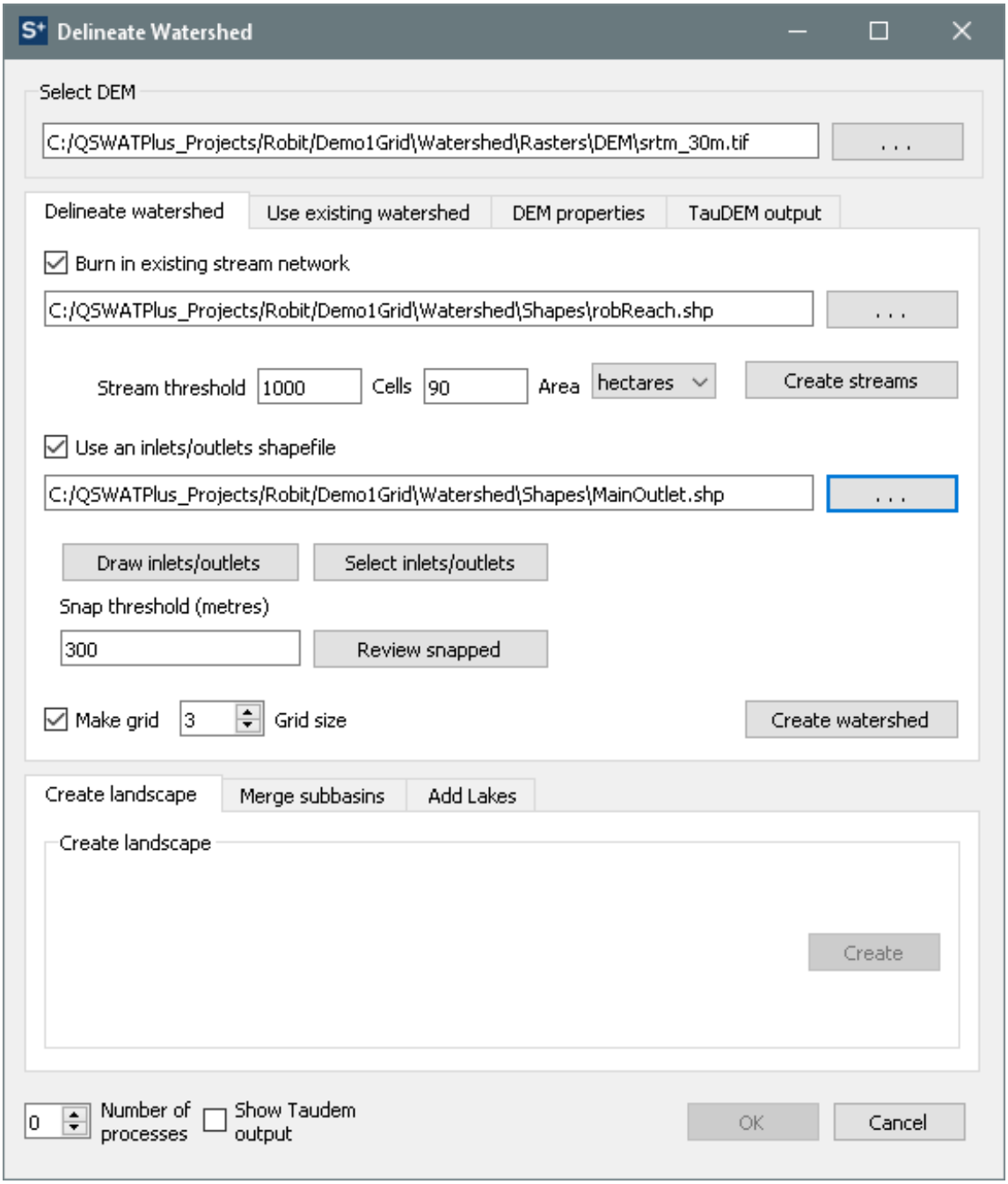


Figure 39: Making a grid

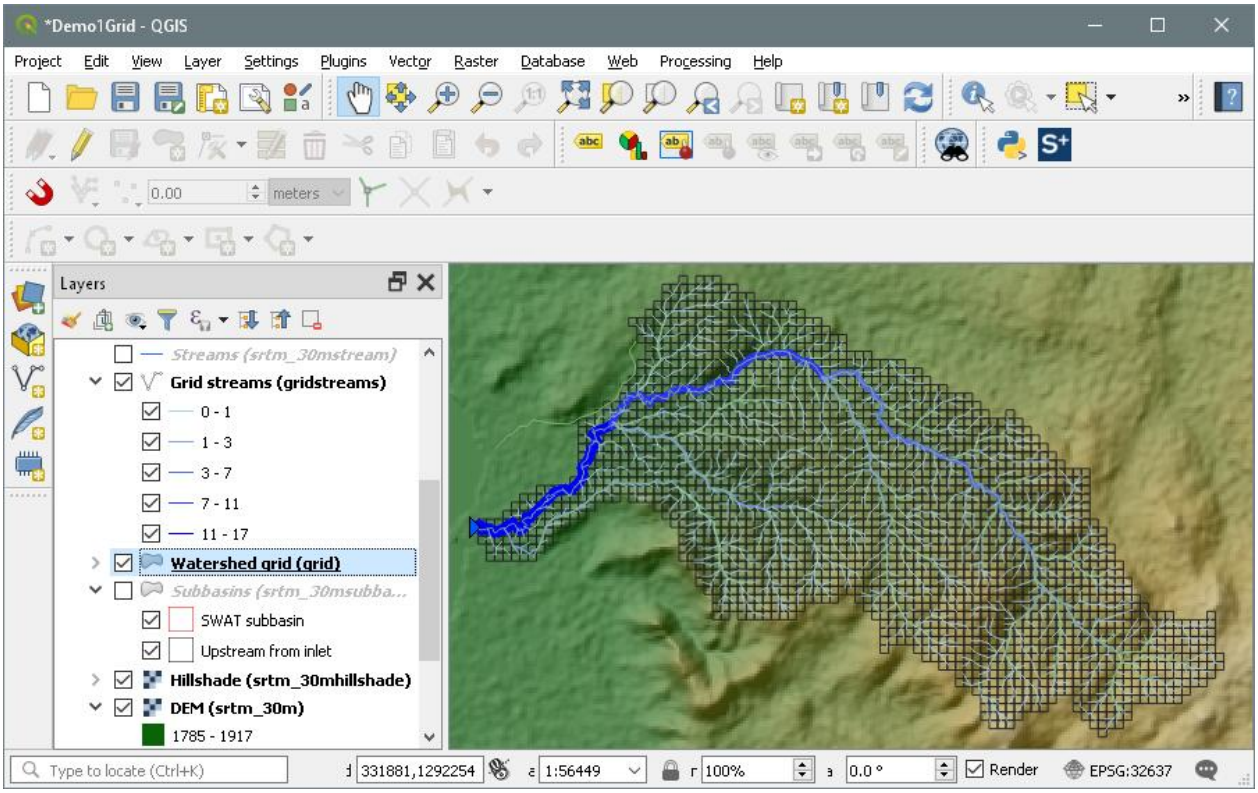


Figure 40: Grid created

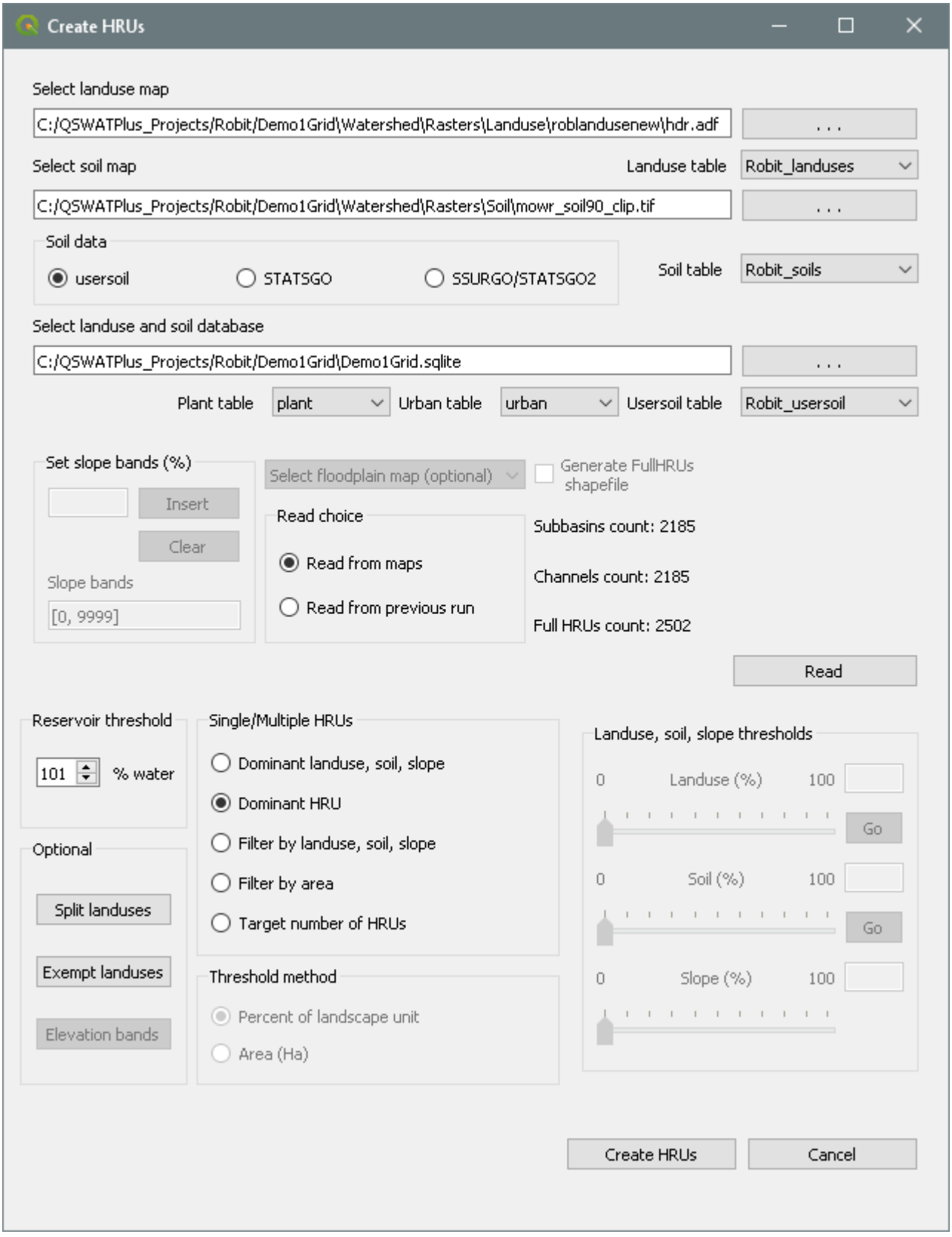


Figure 41: About to create HRUs

10.2 Using an existing grid

You can use a grid created as in the previous section, or one obtained from another source, as

an existing grid. We start, as before, with running the watershed delineation form and loading the DEM: we will need this because we need to calculate slopes. Then we choose the **Use existing watershed** tab, and click **Use grid model**. Next we load a grid shapefile. We can use the grid we created in the *Demo1Grid* project in section 10.1. We will find it under the project folder as *Watershed/Shapes/grid.shp*. The delineation form is shown in Figure 42. The inlets/outlets shapefile is optional, and certainly unnecessary if it only contains outlet(s), as in our example. The important thing to notice is the **Drainage definition** box, which contains three options. These options concern how the grid cells will drain into each other. The first option, **Grid shapefile**, is based on using data stored in the grid shapefile. Grids created by QSWAT+ have attributes *PolygonId* and *DownId*. PolygonId (the standard identifier for subbasins used by TauDEM) is unique to each grid cell. The DownId value is another PolygonId, indicating which other grid cell this one drains to, or -1, indicating a watershed outlet. The second option is to use **Streams shapefile**, and its LINKNO and DSLINKNO attributes will be used to define the drainage routing. As in the non-grid case, each stream reach also has a WSNO attribute, which is the same as the PolygonId attribute in the watershed shapefile. (For streams created by QSWAT+ the LINKNO and WSNO values will often be the same, but this is not required.) The watershed grid shapefile also has an attribute Subbasin, and it is these Subbasin values which are displayed in the map canvas (seen for grids only when you zoom in sufficiently) and will be used as subbasin numbers by SWAT+. To use the streams shapefile option we select it and then we can load the file *gridstreams.shp* created also in the *Demo1Grid* project in section 10.1., and found in its *Watershed/Shapes* folder. In Figure 43 we can see that subbasin 1 drains to 6, which drains to 15 and so on. These drainage routings are defined by the LINKNO and DSLINKNO attributes in the stream shapefile, but also by the PolygonId and DownId attributes in the watershed grid shapefile. In fact the streams shapefile is strictly unnecessary.

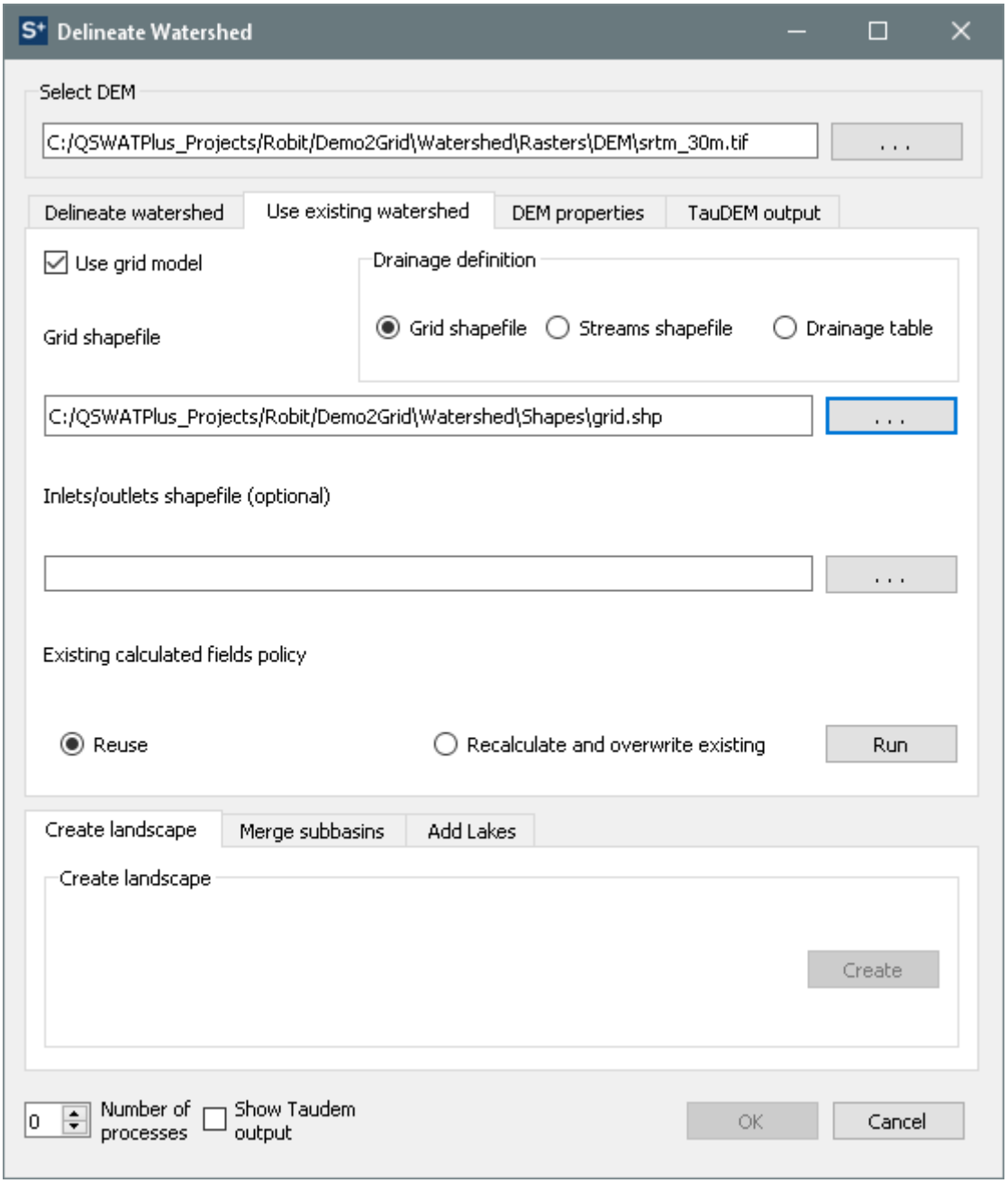


Figure 42: Using an existing grid

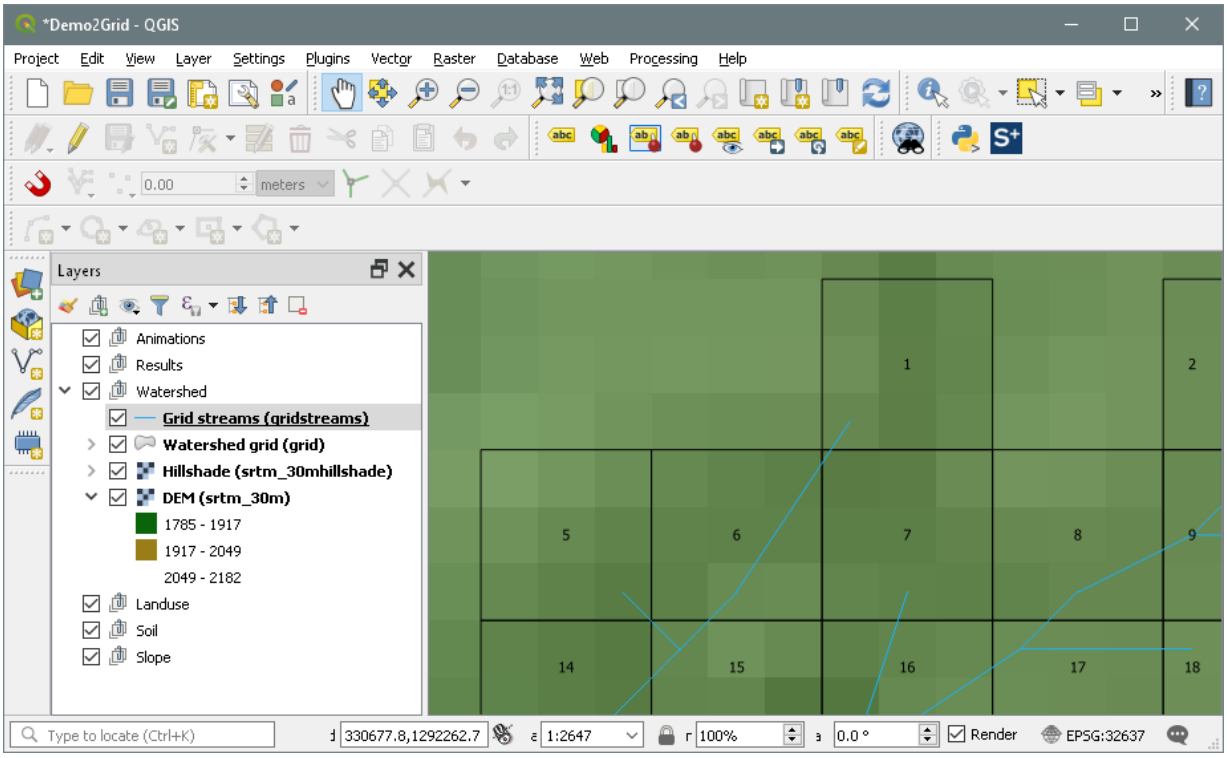


Figure 43: Subbasin number and drainage pattern

If we select for **Drainage definition** the option **Grid shapefile** then the option to add a streams shapefile disappears, and when we click the **Run** button a streams shapefile is created with links the centre points of the grid cells (Figure 44). This is created using only the watershed grid's PolygonId and DownId fields. We do not use any accumulation values, so the best we can do is link the centres of grid cells. You may think this gives a coarser model of the physical terrain (though to SWAT+ it will be just the same as the previous version) but the great advantage is that it can be changed by the user without any danger of terrain analysis contradicting what the user wants. So if you know that, for example, in part of the watershed the drainage has been artificially changed for irrigation, you can change the DownId values in the watershed grid shapefile. Just make sure that all rows have a valid DownId and there are no circularities! Then choose **Watershed shapefile** as the **Drainage definition** option.

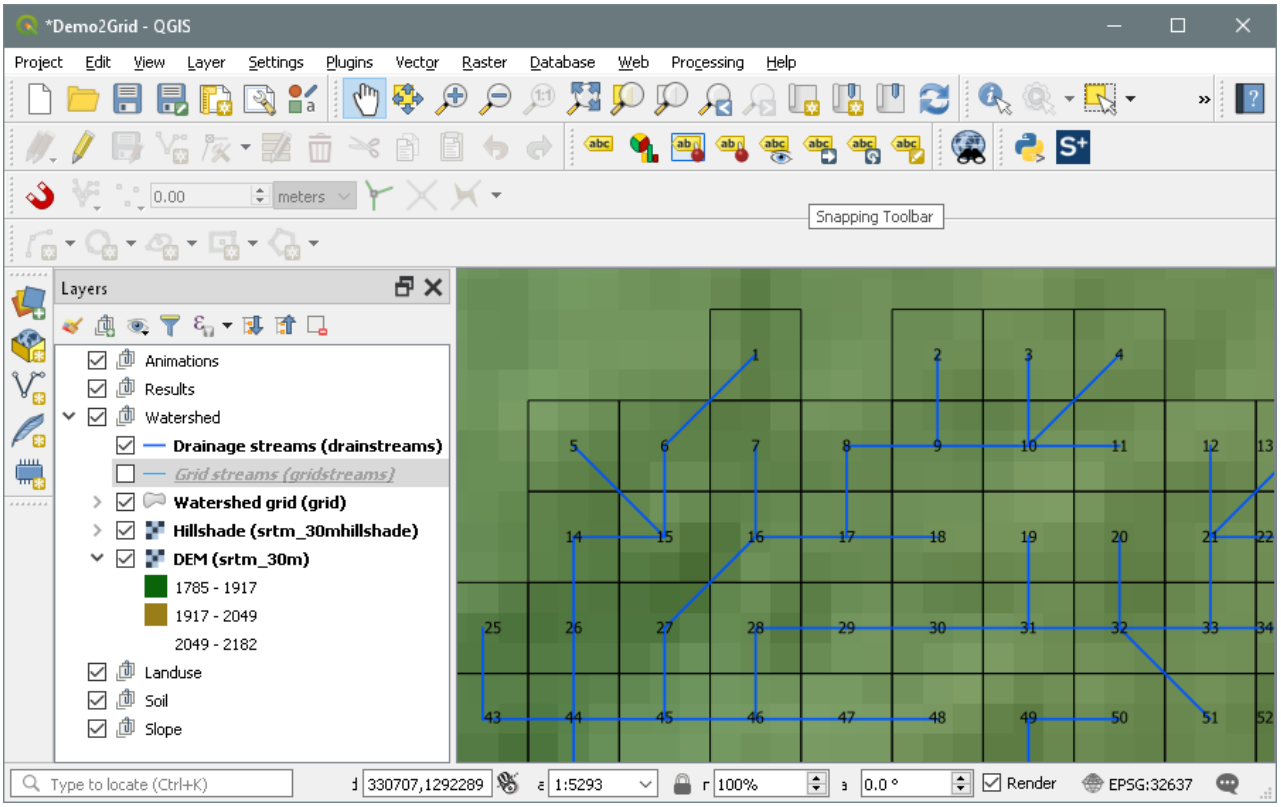


Figure 44: Drainage pattern using watershed grid to define drainage

The third **Drainage definition** option is **Drainage table**. This is functionally the same as using the watershed shapefile, but allows users to create a csv file containing the Subbasin and downstream subbasin values. When we select **Drainage table** we have a drainage table button allowing us to find and load a csv file. If we previously prepared our watershed grid shapefile using QSWAT+ then we will find in the *Watershed/Shapes* directory a file *<project>drainage.csv*, in our case *DemoGriddrainage.csv*, created when the watershed grid was created, with Subbasin and downstream subbasin values corresponding to PolygonId/DownId values. This can be edited and used as the drainage table. If you create such a file from scratch, the first line is ignored, and there should then be two integers on each line separated by a comma. The values in the first column should be non-negative and unique, and values in the second column should appear in the first or be -1 (indicating an outlet). Note that these values are subbasin numbers, as seen in the grid labels, to help users change the values while using the map as reference. Subbasin numbers may be the same as PolygonId values, but this is not necessarily the case.

11. Adding lakes

QSWAT+ allows for lakes to be included as shapefiles in watershed delineation. Here we describe the requirements on lake shapefiles, how to include them, the issues which can arise, and how water flows into and out of lakes are routed.

11.1 Polygon shapefiles

Lakes are defined in polygon shapefiles, and we first need to understand a little about how such shapefiles are structured. A shapefile contains a number of *shapes*, and each shape consists of 1 or more *parts*. This introduces an ambiguity into how shapefiles are constructed. A lake containing an island, for example, could consist of two shapes each with one part, or a single shape with two parts.

For QSWAT+ we need to represent each lake as a single shape (though it can be in a shapefile with several lakes). In addition, we will simplify it further by reducing it to a single part, its outer perimeter. We will be checking how channels cross the perimeter, and having them accidentally cross islands just makes things unnecessarily complicated. In the example from the previous paragraph, a lake and an island as two shapes should first be reduced to a single shape. This is the responsibility of the user, and we will demonstrate how it can be achieved, by *dissolving* shapes. The isolation of the perimeter as a single part is done automatically by QSWAT+.

11.2 Dissolving shapes

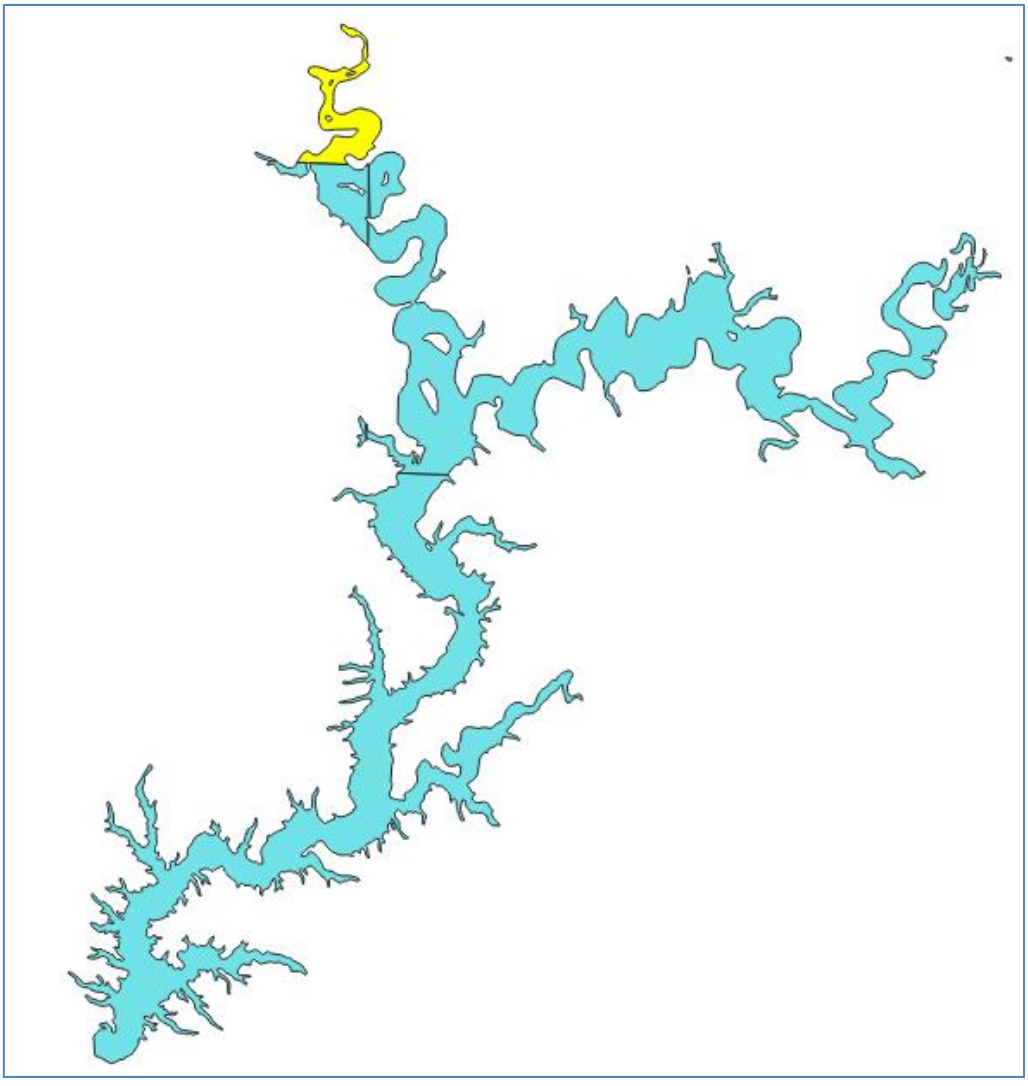


Figure 45: Lake composed of multiple shapes

Figure 45 shows a lake composed of 15 shapes. You can see how it is composed by loading the shapefile into QGIS, right clicking on its entry in the Layers Panel, and selecting *Open Attribute Table*. More than one entry in this table for what is considered a single lake indicates it is composed of several shapes. Selecting a line in the table will highlight the shape, as in Figure 45.

To dissolve the shapes, use *Vector -> Geoprocessing Tools -> Dissolve*. If the lake is the only set of shapes in your shapefile you can ignore the **Selected features only** option and the **Dissolve fields**. Otherwise you will need either to select the features in your lake in the map canvas and click **Selected features only**, or find (or add) a field, or collection of fields, that distinguish the shapes forming the lake. Select a location and name for the new shapefile and click *Run*. Opening the new shapefile and its attribute table will demonstrate that you now have a single shape for each lake.

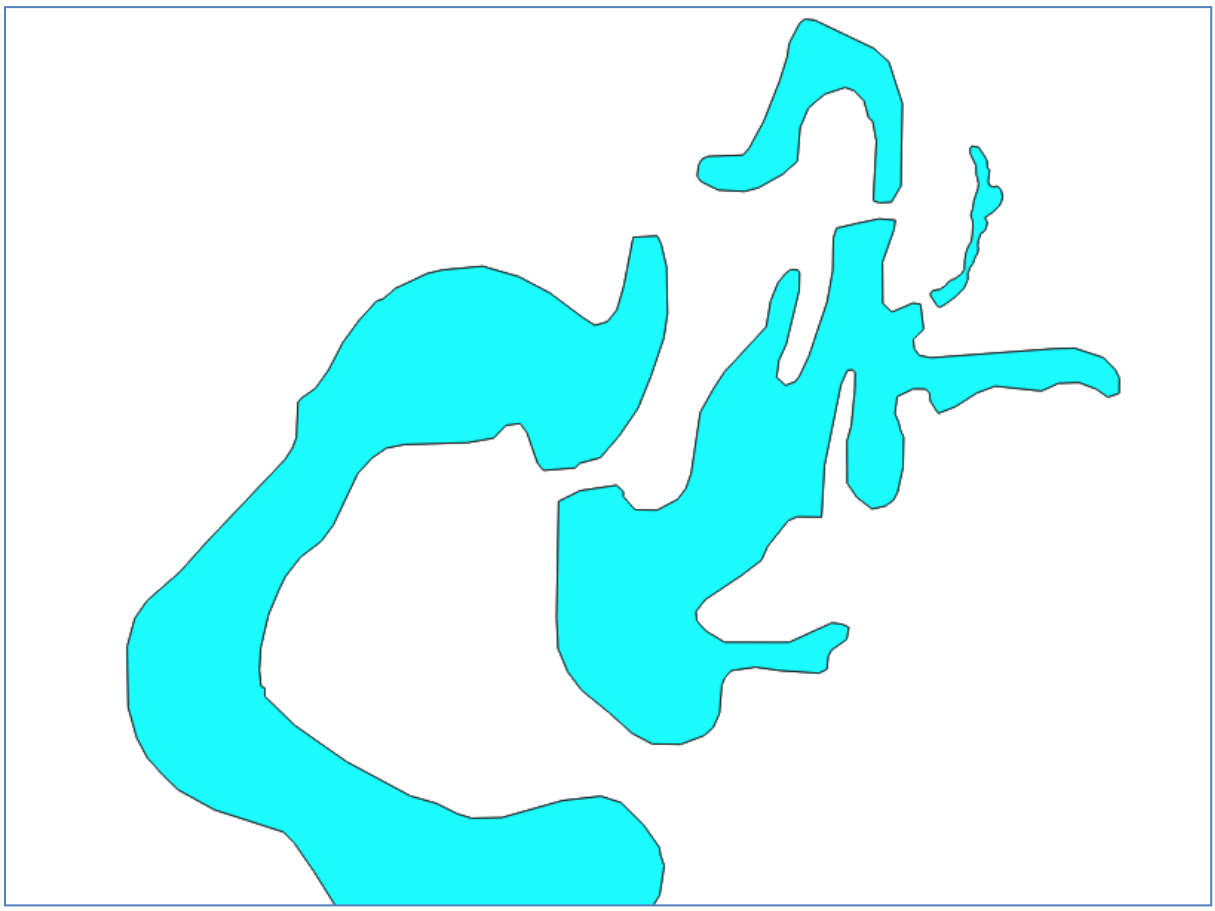


Figure 46: Dissolved shape (detail)

Although we have a single shape, in fact it still may have several parts, because dissolve only joins together parts that physically touch or overlap. Figure 46 shows the north-east corner of the dissolved lake, where we see that it was not very well digitized here: several parts are still separated. When QSWAT+ later selects the perimeter it will select the part with the largest area. But it will also warn if the next largest part (if any) has an area more than 5% of the largest part. To avoid a significant part being lost, go back to the original shapefile, go into editing mode, and drag some points on the edge of one part so they are inside the part you

want to join with. Save the edit and rerun the dissolve.

If you have your dissolved lakes in more than one shapefile, you can easily create a single file containing them all. Load them all as layers in QGIS, then use *Vector -> Data Management Tools -> Merge Vector Layers*. But first make sure they all have the same projection. This will need to be the same as your DEM's projection.

QSWAT+ needs an integer to identify different lakes. You can define this yourself by including in your lakes shapefile a field *LakeId* and assigning a different number to each lake. Or you can leave it to QSWAT+ to add such a field and numbers 1, 2, etc. This is only done to the copy of the lakes shapefile within the project. When you load the lakes shapefile each lake will be labelled *Lake n* where *n* is its LakeId value.

The lakes shapefile may also have a field *RES* indicating with value 1 if the lake is a reservoir,, or indicating with value 2 that it is a pond. If there is no such field the default is reservoir.

11.3 Delineation with lakes

The lakes shapefile is selected in the delineation form, and can be added after you have run delineation. The shapefile is loaded as a layer, and as with all input maps is copied, in this case into the *Watershed/Shapes* directory, keeping its file name. But it is not a simple copy: each shape is reduced to the outer perimeter of its largest (by area) part. As mentioned above, the user is warned if there are any ignored parts with an area more than 5% of the largest part. The rest of the processing happens when you click the final **OK** button in the **Add Lakes** tab. It proceeds as follows:

11.3.1 Channel splitting

It is required that each lake has an outlet. Examples which have a single outlet and cause no problems are illustrated in Figure 47, Figure 48, and Figure 49.

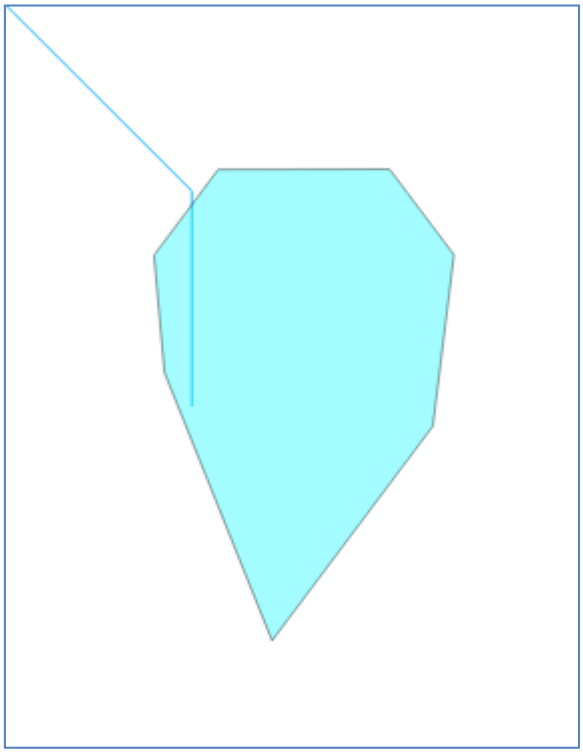


Figure 47: Single outlet; no inlets

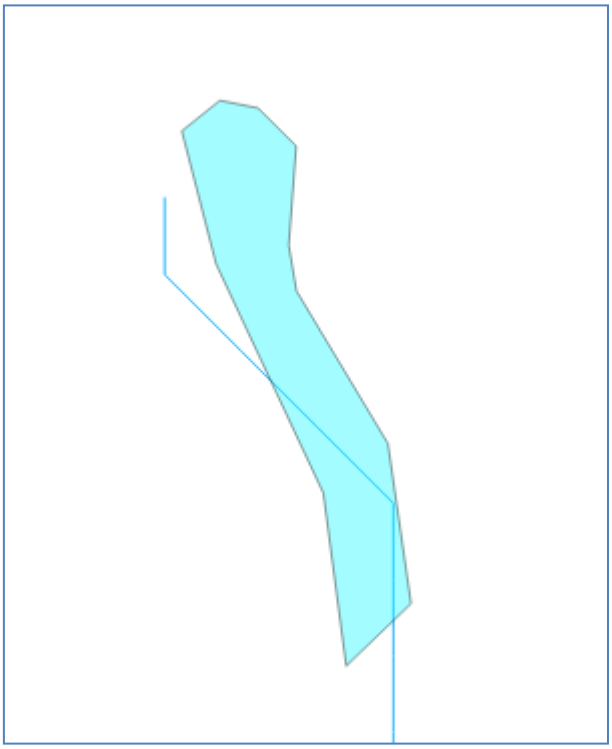


Figure 48: Single outlet; single inlet

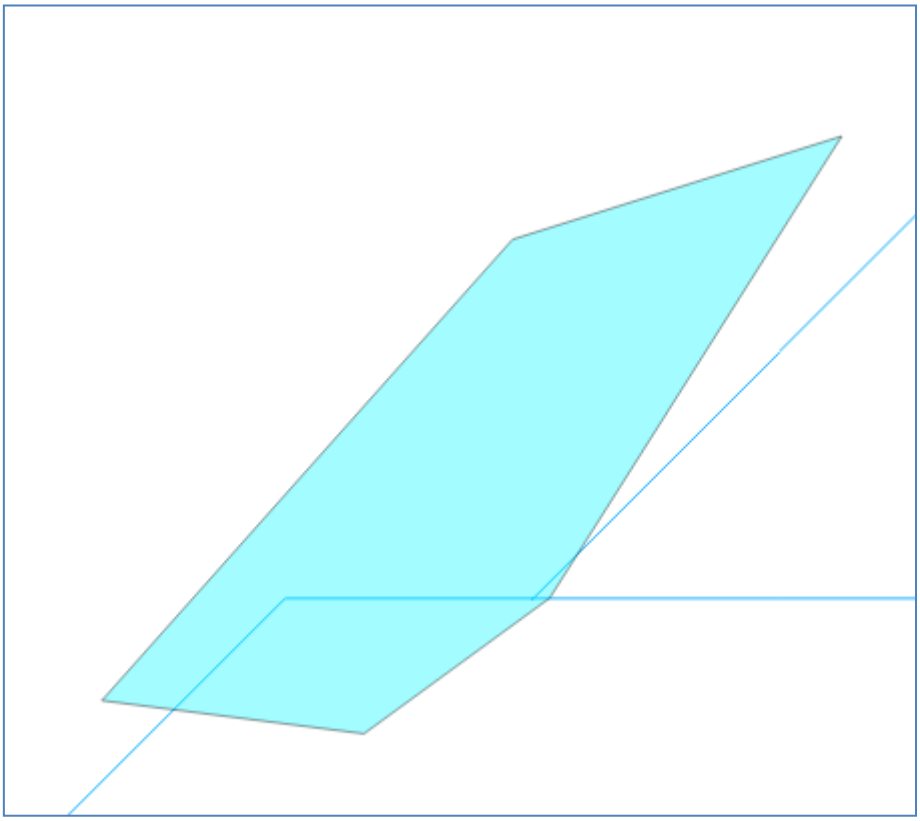


Figure 49: Single outlet; multiple inlets

The aim is to divide channels which interact with each lake into three groups: channels which enter the lake (which will terminate at the lake boundary), channels which leave the lake (which will start at the lake boundary) and channels entirely within the lake. So the single channel in Figure 47 will be split into a channel within the lake and a channel leaving it. The single channel in Figure 48 will be split into a channel entering the lake, a channel within it, and a channel leaving it. Figure 49 involves three channels: two entering the lake and one leaving it. These will each be divided where they cross the lake boundary, so we eventually have two channels entering, three inside, and one leaving.

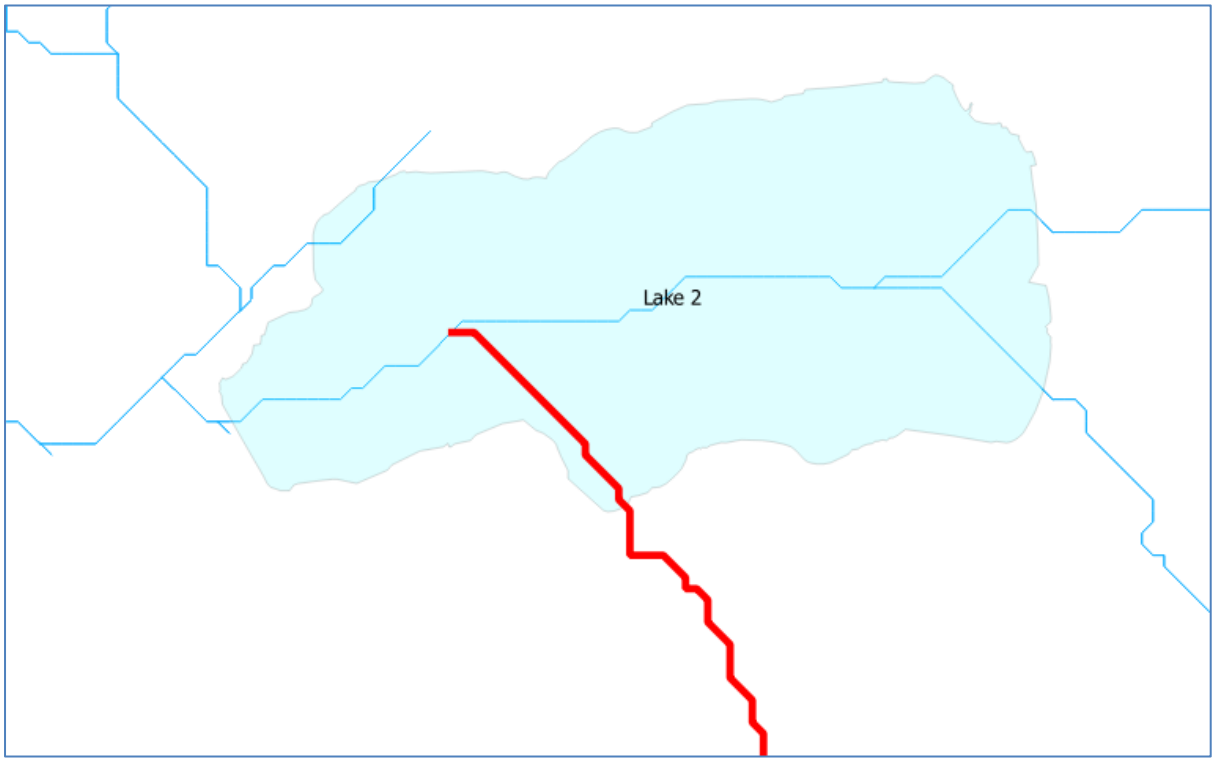


Figure 50: Before splitting channels

Figure 50 shows part of the channel network after delineation and before channel splitting, with one channel highlighted in red.

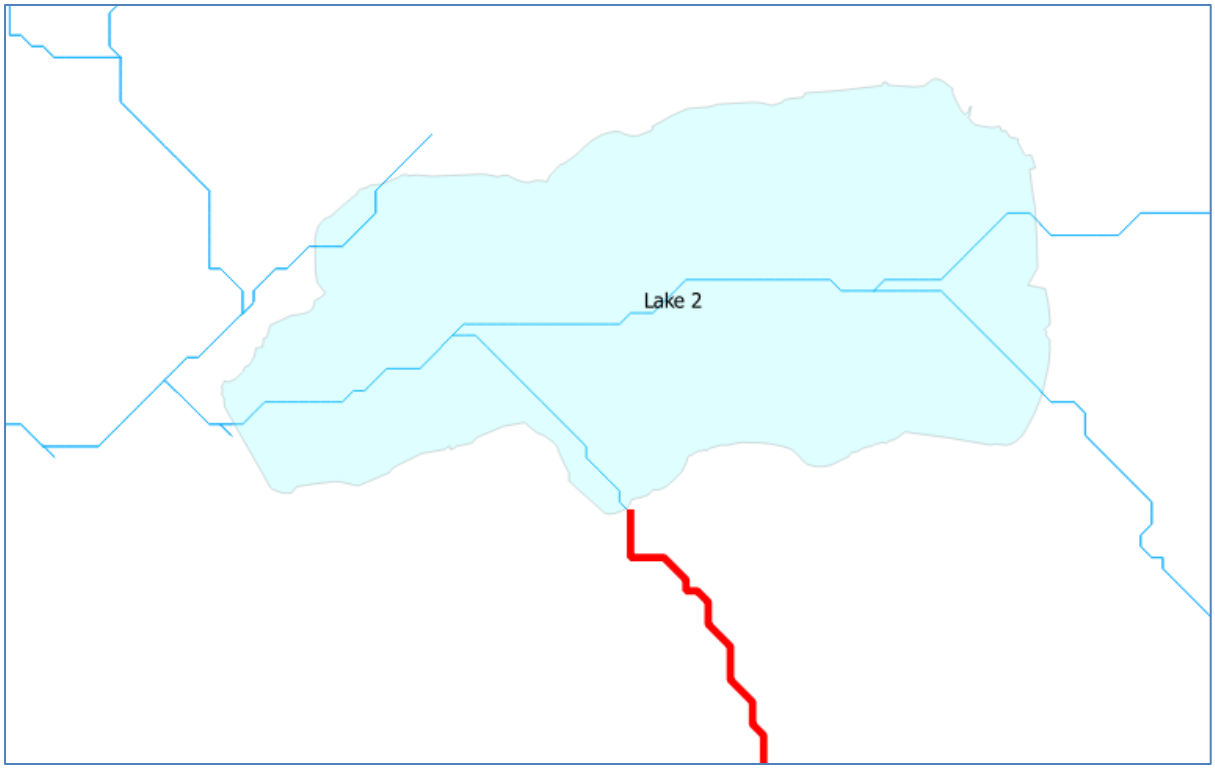


Figure 51: After splitting channels

Figure 51 shows the effect of splitting channels. We have highlighted the upstream channel of the pair that has replaced the previously highlighted channel. The downstream channel is within the lake.

The aim of splitting channels is to find out what areas drain to where. Areas that drain to the new upstream channel outside the lake will be routed to that channel and thence to the lake. Areas that drain to the new downstream channel inside the lake will instead drain directly into the lake. You might expect such areas to be themselves in the lake, but this is not the case. Figure 52 shows the original drainage area of the single channel highlighted in Figure 50. Figure 53 shows the drainage areas after splitting channels and then adding the lake. The lake area has been deleted from the drainage areas. The yellow area in Figure 53 is the area draining to the new upstream channel. The blue area is the area that remains of the yellow drainage area in Figure 52 that is outside the lake. This is the area that technically drains into the downstream channel, but in the SWAT+ model will drain directly into the lake. In fact in the SWAT+ model the channels inside the lake are removed.

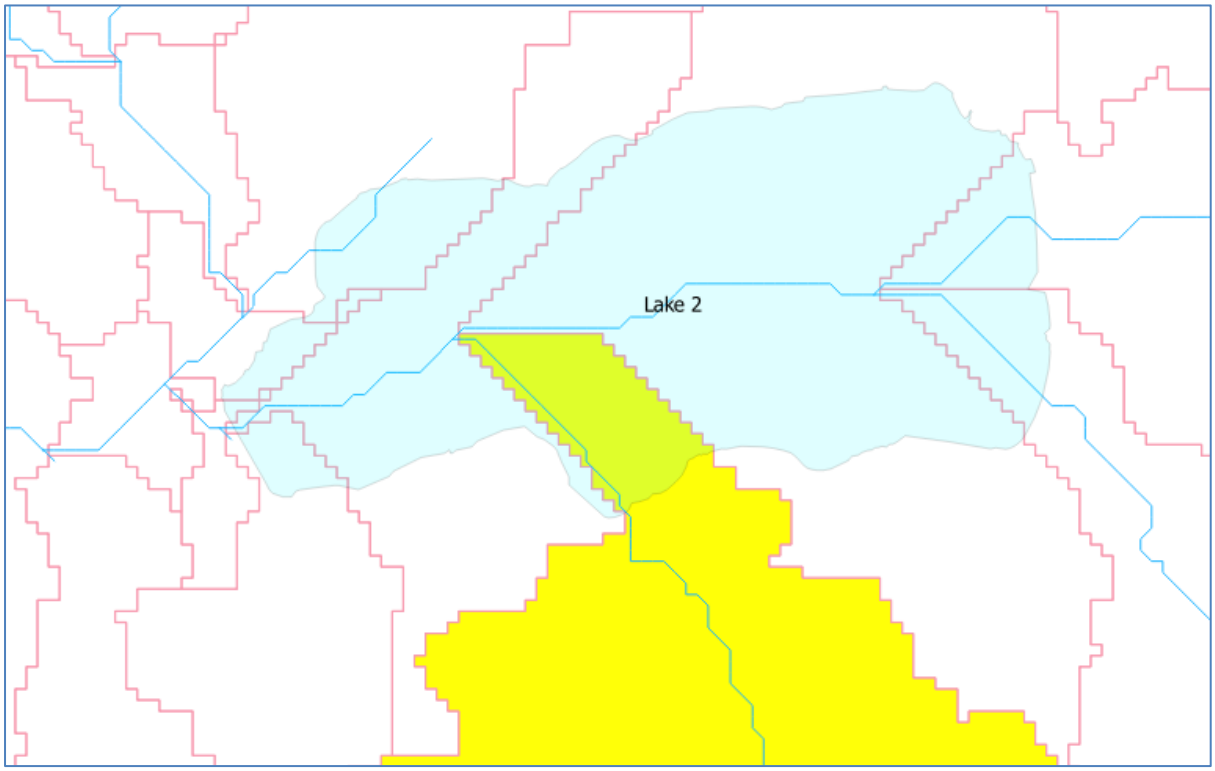


Figure 52: Channel drainage areas before channel splitting

Note that LSUs are formed from channel drainage areas: one LSU per channel if no floodplain is used, else two (floodplain and upslope) if a floodplain is used. LSU numbers are related to channel numbers: if a channel has number *n* then the corresponding single LSU will have number 10 * *n*. When a floodplain is used, the two LSUs will have the number 10 * n + 1 (floodplain) and 10 * n + 2 (upslope). This makes it easy to relate LSUs to channels.

A consequence of removing channels inside lakes while keeping channel drainage areas like the blue area in Figure 53 is that there will be LSU numbers that do not relate to any channel numbers: the blue area's LSU number(s) will be related to a channel number that has been removed. If no floodplain is used, the number of LSUs is normally equal to the number of channels. But if lakes are included the number of LSUs will typically exceed the number of channels.

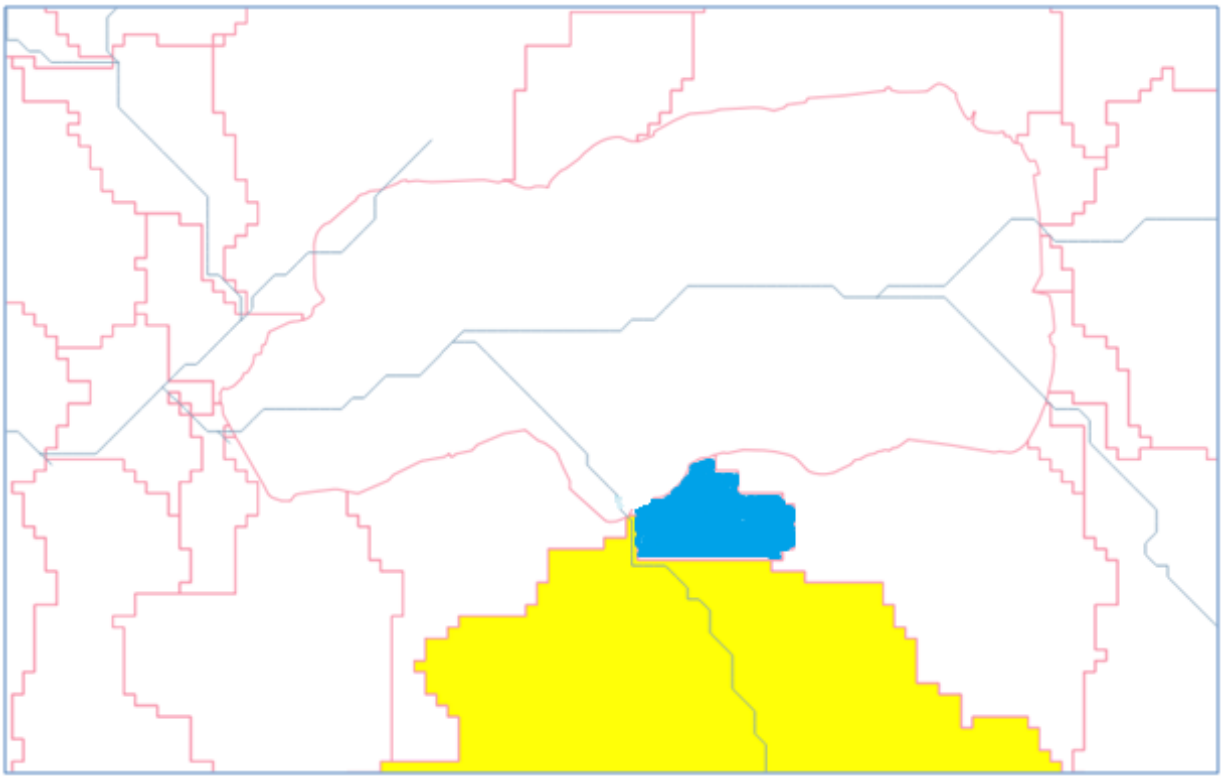


Figure 53: Drainage areas after channel splitting

11.3.2 Problems

Ideally, each channel interacting with a lake should fall neatly into one of the three categories: entering, leaving and within. But in practice things are not so simple. Lake boundaries may not coincide with the model of channels and basins created by delineation, which is always imprecise, and on flat areas like the surfaces of lakes particularly so. Neither can we rely entirely on the lake shapefile: separated parts can occur as we have seen, and in any case the perimeter of a lake at the end of the dry season may be significantly different from its perimeter at the end of the wet season. We have to accept compromises and use our judgement and local knowledge.

The first problem is when the requirement that every lake must have a channel outlet is not met. This might happen because a lake is outside the watershed boundary (Figure 54) or because no channel interacts with it (Figure 55). When there is no outlet QSWAT+ zooms to the lake with the problem and asks the user if they wish to remove the lake, or if they want to keep the lake and try to correct the problem. If the user clicks *Yes* to removal the lake is removed from the copy of the lakes shapefile in the project. Otherwise the lake remains and the user needs to correct the problem in some way, such as changing the channel threshold, burning in a channel map, or editing the lake polygon, and then rerunning delineation.

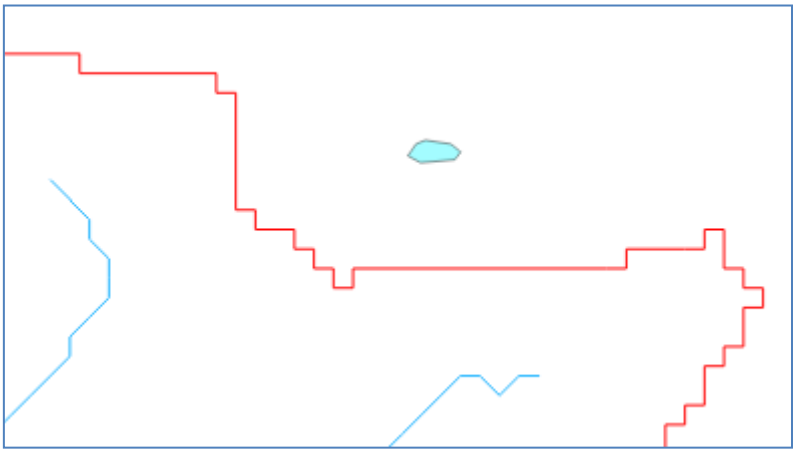


Figure 54: Lake outside the watershed

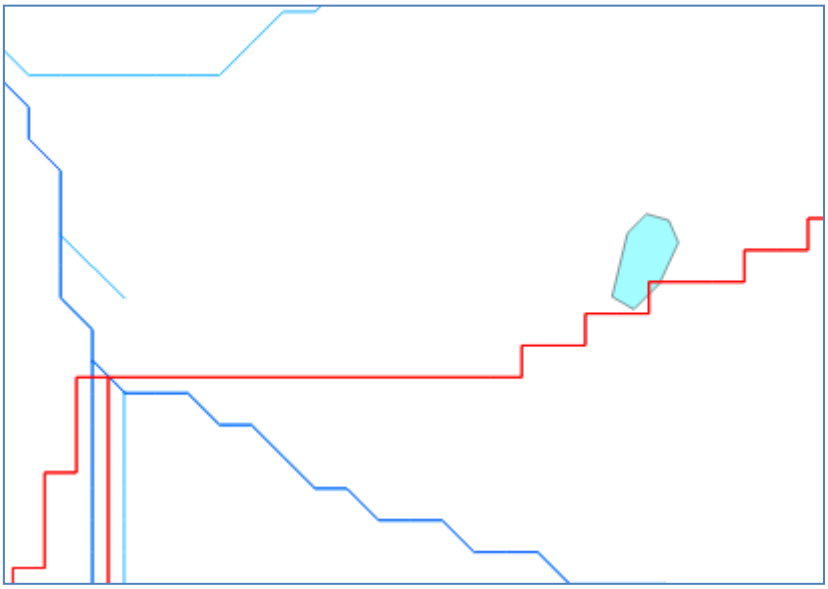


Figure 55: Lake not on a channel

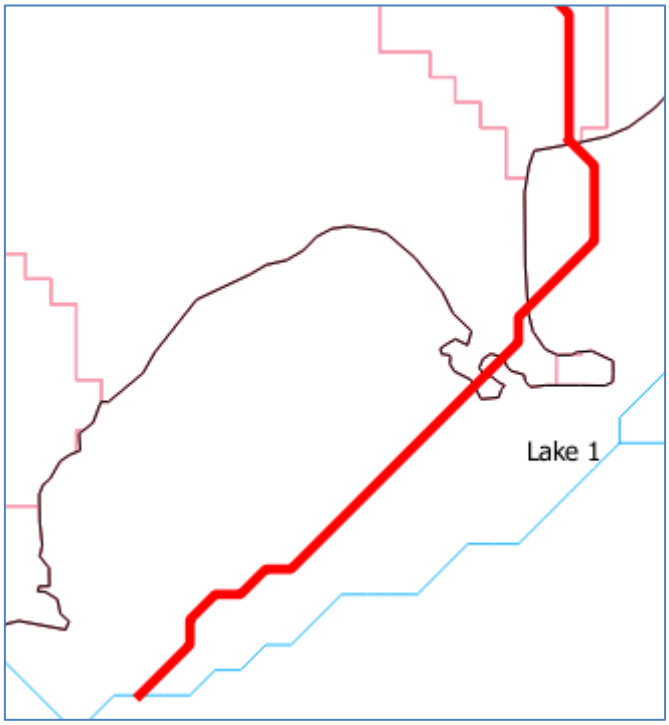
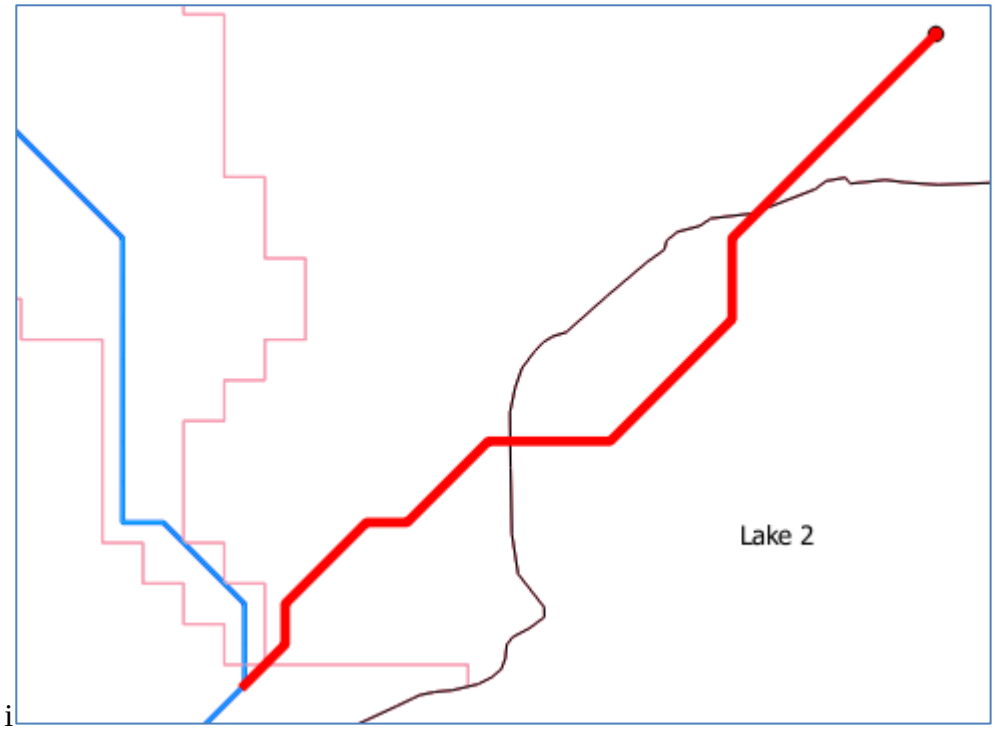


Figure 56: Multiple crossings

Figure 56 is an example of a channel (highlighted in red) that starts in the north outside the lake (perimeter in black) and crosses the lake boundary three times. Since the channel starts outside the lake and terminates within it, it will be designated an inflowing channel and split at the point where it crosses the lake boundary which is nearest to the channel source. A warning is given to the user that this designation has been made.



i

Figure 57: Another multiple crossing example

Figure 57 shows an example where a channel (again highlighted in red) starts and finishes

outside a lake. Unless there are no other channels leaving the lake (as in Figure 48), the designation made is that it should not in fact cross the lake boundary; all the water in its channel basin will be routed into the channel and thence to the channel outlet in the south-west. Again the user is informed of this designation.

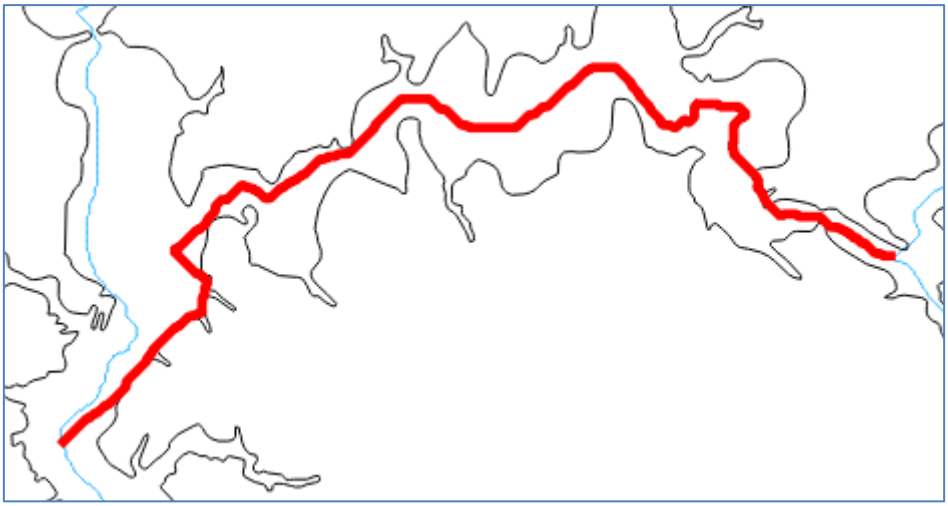


Figure 58: A channel internal to the lake with multiple crossings

In a similar fashion, a channel that starts and finishes within a lake but crosses the boundary is regarded as internal to the lake, with a warning to the user. Figure 58 is an example involving multiple crossings.

A channel that starts inside the lake and finishes outside it but crosses the lake boundary more than once is designated an outflowing channel, again with a warning, and split at the crossing point nearest its outlet.

11.3.3 Fixing problems

It may be that the designations selected by QSWAT+ in the cases above are not appropriate. There are also problems where an error is reported and the user needs to take action.

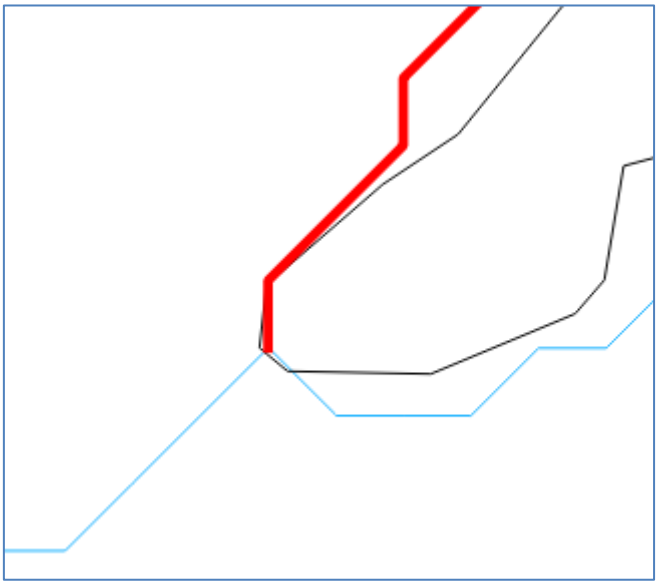


Figure 59: Channel in lake appears to be an outlet.

In Figure 59 the lake is the area in the left and bottom of the figure: there is a spit of land coming into the lake from the north-east. The channel highlighted in red, mostly internal to the lake and flowing north-east to south-west, appears to terminate on land, so terminates outside the lake. Similarly the channel flowing roughly east to west appears to terminate outside the lake, and the channel flowing to the south-east appears to be an inflow, since it starts outside the lake. All of these are wrong, and a warning about multiple lake outlets will be reported. We need to arrange it so that the channel junction is in the lake rather than on land. We could try to achieve this by editing the channels shapefile to move the ends of the channels, but if we ever rerun the delineation the error will recur. We could try to avoid this by burning in the adjusted channels shapefile, but this method is not always reliable. A better approach is to shift the lake boundary a little, in this case to move a few points at the end of the spit of land so that none of the three channels cross it.

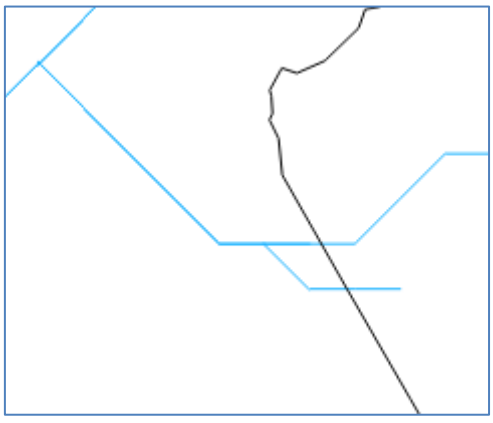


Figure 60: Multiple outlets

Figure 60 illustrates a situation which will also be reported as a lake (on the right) having multiple outlets. In this case there are three things we could do:

1. We can ignore the warning. The short lower channel (assuming it has a smaller drain area than the upper one) will by default receive 0% of the lake's outflow, and the upper chan-

nel (assuming no other outlets) will receive 100%. Effectively the lower channel is treated as internal to the lake. But the outlet routing of 0% remains, and we could change the percentages later with the SWAT+ editor.

2. The short lower channel might disappear if we increased the channel threshold a little and ran delineation again. However, the channel threshold applies across the watershed and the new threshold might not be suitable elsewhere.

3. We could edit the lake so that it crosses the upper channel just to the west of the junction of the two outlet channels.

11.3.4 Short outlet channels

Sometimes it is natural to make the outlet from a lake the outlet from the whole watershed. How can we achieve this while obeying the rule that each lake has one main outlet channel? We don't want to create a short channel with a correspondingly small channel basin. There is an extra condition that if a watershed outlet (an outlet with no other outlets downstream) is placed on a channel within a lake and close to the lake boundary, then the channel is extended to the boundary and the outlet moved to the channel's new outlet. The definition of "close" in this context is (approximately) the same as used for snapping points to channels: 300m by default. If a watershed outlet is placed further away from the boundary then the lake is considered to be closed: it may have inflows but it has no main outflow.

This solves the problem of placing a watershed outlet on a lake boundary, but there is a similar issue that lake outlets can often occur very close to subbasin outlets, with a short channel between them. We cannot easily shift the subbasin outlet, which is determined by delineation, so how can we avoid the small channel and (perhaps) correspondingly small channel basin? To deal with this there is another special condition: if the outflow channel from a lake has a drainage area less than 1% of the lake area, then the channel is regarded as internal to the lake, and the water flow from the channel basin will be into the lake instead of into the channel.

11.3.5 Multiple outlets

Normally one expects a lake to have a single outlet. But it is sometimes the case that lakes have other outlets, water being drawn for irrigation, for example. QSWAT+ allows for multiple outlets, but the user is always warned if the number of outlets is different from the most common case of 1. As we have seen, extra outlets are commonly the result of a mismatch between the results of delineation and the real world. So there is an assumption that less important outlets are probably mistakes. If there are multiple outlets, QSWAT+ compares the drainage areas of each (according to the original channel network, before lakes are added). The one with the largest drainage area is designated the main outlet. When the routing table is constructed, the main outlet will be assigned 100% of the lake's outflow. The other outlets will route 0% of the lake's outflow to their downstream channels, and so by default will be ignored. These percentages can of course be changed in the SWAT+ editor.

11.3.6 Water routing

The rules for routing water into and through lakes are as follows:

1. Remember that channels which flowed into a lake across its perimeter were split into inflowing channels and channels within the lake. The outflowing channels are split similarly. So channels that interact with the lake are either (a) inflowing but entirely outside, (b) within, or (c) outflowing and entirely outside.
2. Channel basins for channels outside the lake are routed just like channel basins not interacting with lakes. The only difference is that an inflowing channel has an outlet point created at its outlet end (on the lake boundary), the channel is routed to this point, and the point is routed into the lake. The main outflowing channel has a reservoir or pond point created at its source. 100% of the lake is routed into this point, which in turn is routed to the main outflowing channel. Other outflowing channels receive by default 0% of the lake outflow.
3. Channels within the lake are removed from the model, and so are not routed.
4. The channel basins for channels within the lake may be entirely within the lake, or may have some area outside the lake. In the former case they are removed from the model. In the latter case, the channel basin is reduced to the area outside the lake and all flows that would normally be routed to the basin's channel are instead routed directly into the lake.
5. In the cases where the outlet channel is taken to be within the lake, the lake's reservoir or pond point is routed directly to the subbasin outlet at the outlet end of the channel.
6. When a watershed outlet is placed initially within the lake, or at the end of a sufficiently short channel for the channel to taken into the lake, the lake's reservoir or pond point is routed directly to the watershed outlet.
7. By default in QSWAT+, all channels have a point source created at their source. If inputs are defined for any of these point sources, they are routed into their channel.

These rules only describe the default actions of QSWAT+. Users can freely edit the routing table using the SWAT+ Editor.

11.4 Files involved

The channels shapefile created from the delineation before lakes are added is called *DEMchannels.shp*, where the DEM raster is called *DEM.tif*, and is found in the *Watershed/Shapes* folder. Channel splitting creates a new channels shapefile *DEMchannels1.shp*. The original is retained because it can be used to locate the channels mentioned in messages about channels with multiple crossing points with lakes. The replacement channels shapefile is created by rerunning part of delineation and the LINKNO values of channel reaches are likely to be different.

Other files affected by adding lakes are the subbasins file *Watershed/Shapes/DEMsubbasins.shp* and the channel basins *Watershed/Shapes/DEMwshed.shp* (not normally loaded). All of these have the lake areas removed. The last also has channel basins split where channels were split. The subbasins and wshed shapefiles are later the basis for the subs1 and lsus1/2 shapefiles respectively.

The extra outlet points at the outlets of channels flowing into lakes, and the reservoir or pond points at the lake outlets, are added to the inlets/outlets shapefile created by snapping points to streams and channels. They can be made visible by reopening the delineation form, clicking the *Review snapped* button, and then clicking C*ancel* (to avoid trying to rerun lake processing).

11.5 Lakes in grid models

Lakes can also be added to grid models. This is done essentially by designating a number of cells in the grid as forming lakes.

To add lakes to a grid model, first delineate it as usual: this is the same as non-grid delineation except you check **Make grid** and pick a grid size. Then you select the **Add Lakes** tab, and click the '...' button to select and load the lakes shapefile. As in the non-grid case, each lake is reduced to its perimeter, and *LakeId* values are added if necessary. Then the grid cells which overlap with each lake are marked as forming part of the lake if the lake includes at least 50% of the cell.

In grid models, there is no distinction between streams and channels - there are only channels. Each channel runs from one grid cell to an adjacent one. So after defining which cells form a lake, it is possible to define channels as not interacting with the lake (neither end is a lake cell), inflowing (the outlet end only is a lake cell), within the lake (both ends are lake cells) or potentially outflowing (the source end is a lake cell, the outlet is not). As in the non-grid case, there can be several outflowing channels. If there are several, the main one is chosen as the one with the largest drainage area, the user is informed, and as in the non-grid case the default routing will be 100% to the main outlet, 0% to the others.

In the non-grid case, editing of the lake perimeter, typically by dragging some of the perimeter points, can be done easily using the shapefile editing capabilities of QGIS. For grid models it is not so easy: the user could edit the lake shapefile before loading it, but it is difficult to judge which grid cells will be included or excluded in advance. Instead a simple mechanism is provided to edit the lake when it has become just a set of the cells in the watershed grid. There are buttons **Remove lake cells** and **Add lake cells**. Clicking either of these puts QGIS in a mode where grid cells can be selected by left-clicking in them. You can hold the Ctrl key to make a multiple selection - the selected cells turn yellow - and then click **Ok** to make the change or **Cancel** to leave the lake unchanged. Since there may be more than one lake in your model, before adding cells you need to select the appropriate lake identifier in the *Lake number* box.

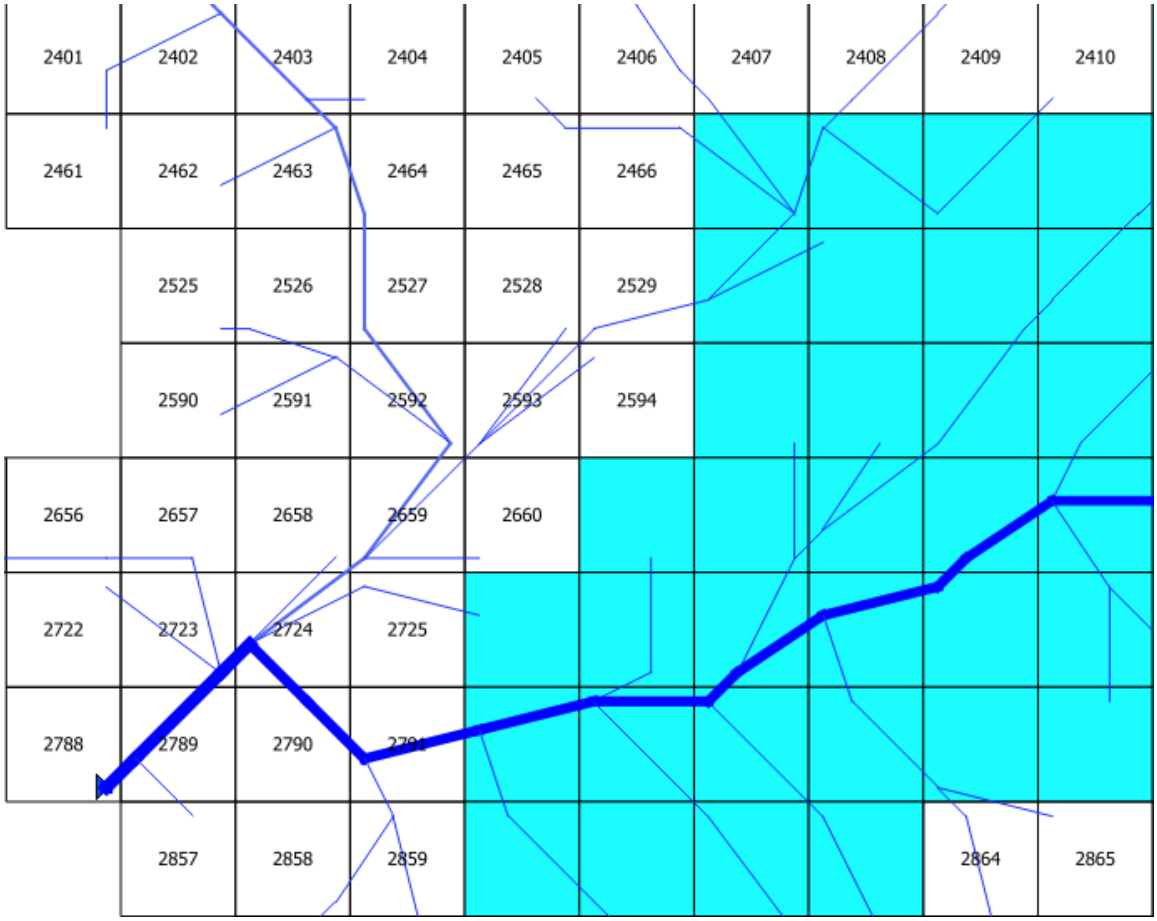


Figure 61: Initial grid lake

Figure 61 shows the initial state of a lake in a grid model. (Cell numbers are removed for lake cells, but we refer to some cells in the lake using the numbers they would have had before the lake was added: the numbering system should be obvious from the figure: in each row cells are numbered consecutively left to right.) We can see that the lake outlet should be from 2792 to cell 2791 (channel widths increase with their drainage areas), but there are also potential outlets into 2725 and 2529. QSWAT+ will settle on 2792 for the main outlet and warn about the others, and we can see that there are only a few cells involved and leave things as they are. Alternatively we could take the drainage csv file generated when we delineate grid models, edit it, and use it in a predefined watershed project. We could, for example, change the DownSubbasin of Subbasin 2726 to 2792 instead of 2725, and change that of 2530 to 2595. A third option is to decide that this north-west corner of the lake is inaccurate and encroaching into another natural basin: we need to edit the lake. We can remove the lake cells occupying cells 2726, and 2530, but then we see that 2530 is now a potential outlet, and we see that we need to also remove 2531, 2467, 2468 and 2469. The result is shown in Figure 62

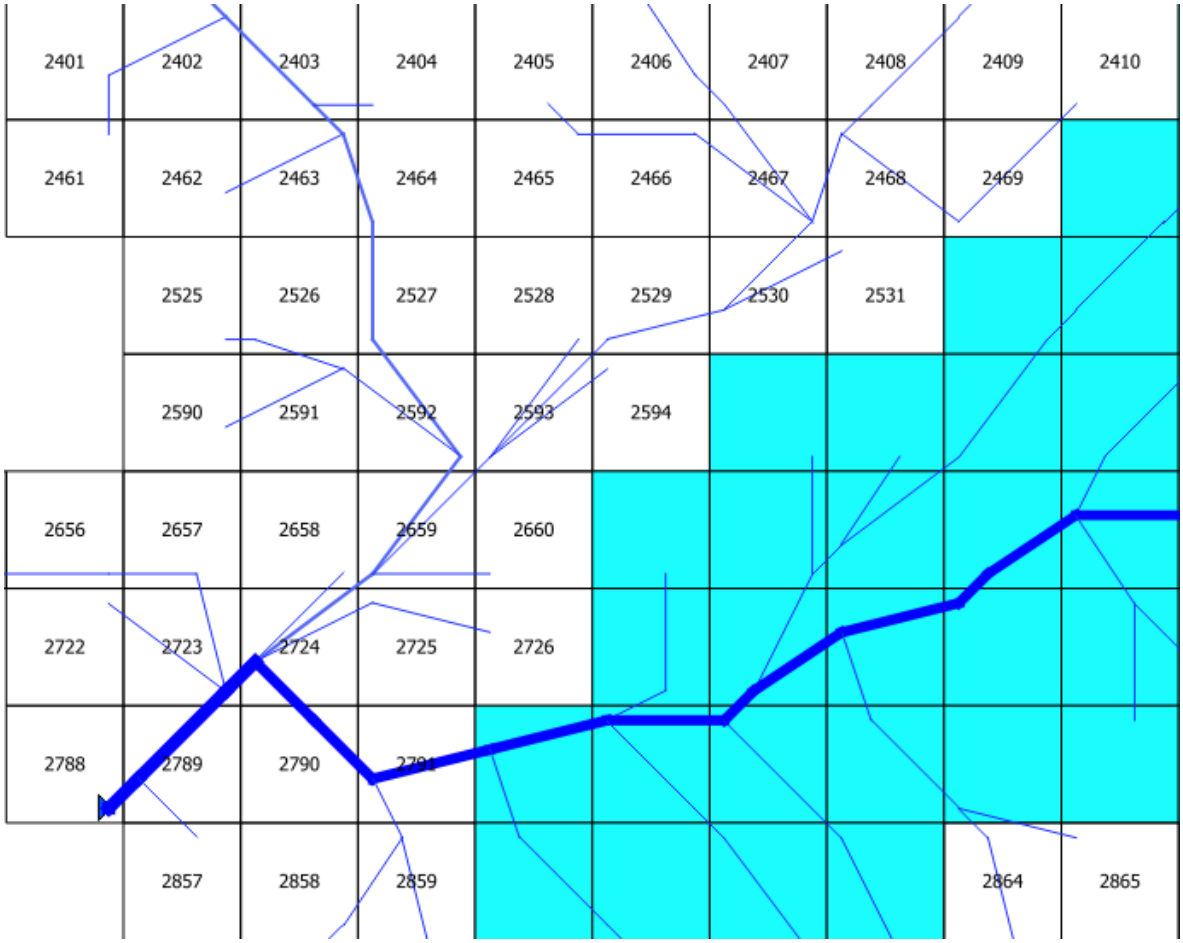


Figure 62: Grid lake after removing some cells

In the non-grid case channels were split at the lake boundary. We cannot split the grid channels, because they always run cell to cell, but we still place outlet points on inflowing channels and a reservoir or pond point on the main outflowing channel. For inflowing channels the point is placed at the channel outlet (so apparently inside the lake) and the routing is channel to point to lake. For the outflowing channel the point is placed at the source end of the channel (again apparently inside the lake) and the flow is lake to point to channel.

11.6 Lakes in predefined watersheds

QSWAT+ supports the use of predefined watersheds, in which the user supplies the streams and channels, subbasins and channel drainage areas as shapefiles, and no watershed delineation is done. Generally this is to avoid the delineation process changing the stream lines or the basin boundaries.

The principal that outputs from delineation can be used as inputs for predefined models is maintained: delineated models include all the necessary fields described below.

11.6.1 Predefined grid models

With predefined grid models adding lakes is very simple, and the process is the same as in the

delineated case. The user supplies the grid and the drainage pattern, and adds the lake shapefile. The lakes are converted into collections of grid cells by intersecting the lake with the grids, and the inflowing and outflowing channels identified. As in the delineated case, the lakes can be edited by adding or removing cells.

11.6.2 Predefined non-grid models

Adding lakes to predefined non-grid models is much more complicated. As we described earlier, we need to split channels that cross a lake boundary into a channel outside the lake and a channel inside it. The splitting is straightforward, but we also need to divide the region draining into the original channel into two regions, and this can only be done by delineation techniques. This contradicts the idea of a predefined watershed avoiding the use of delineation. So instead it is necessary for the user to provide a channel network that has already been split, and a corresponding watershed shapefile that defines the region draining to each channel. The following changes are necessary to the requirements on the input shapefiles for predefined non-grid models:

11.6.2.1 Lakes shapefile

This is required, with one shape for each lake. There must be a *LakeId* field in the attributes, with each lake having a unique strictly positive LakeId value. The shapefile is in fact only used to calculate the area and centroid of each lake, so it is not required, for example, to reduce the shapes to outer perimeters only. The area and centroid values are always calculated from the geometry of each lake, regardless of any other fields which might define them. The lakes shapefile should also have a *RES* field indicating with a value 1 if the lake is a reservoir or 2 if it is a pond. If there is no such field reservoir is assumed.

11.6.2.2 Channels shapefile

The channels shapefile requires four extra fields: *LakeIn, LakeOut, LakeWithin*, *LakeMain.* Values in these fields may be NULL (empty) or 0 (zero), indicating the channel has no interaction with any lake, or a value found in the LakeId field of the lakes shapefile, indicating that the channel flows into the lake, or flows out of it, or is within it. Channels marked as flowing into a lake are regarded as having their outflow point on the lake boundary. Channels marked as flowing out of a lake are regarded as having their source on the lake boundary. The LakeMain field is used to give the LakeId for which the channel is the main outlet: it is only read when LakeOut has a positive value, and must have the same value. The channels flowing into lakes may have DSNODEID values, but these are ignored. The reason is that the requirements above are met by *channels1* files created during delineation, and these have DSNODEID values which correspond to points added to the snapped inlets/outlets file. But this file is not commonly accessed by users, and in any case predefined watersheds typically do not need or use inlets/outlets files. So such values are ignored. Outlet points are created for the inflowing channels, as in the delineated case, but this is done internally and no inlets/outlets file is created. These points will of course be defined in the *gis_points* table and referenced in the *gis_routing* table of the project database.

11.6.2.3 Subbasins shapefile

This is assumed to correspond to the subbasins with the lake shapes removed, so that in particular the total watershed area will be the sum of the subbasin shape areas plus the lake areas.

11.6.2.4 Watershed shapefile

This defines the area draining into each channel. As with the subbasins shapefile it is assumed to correspond to the drainage areas for each channel with the lake shapes removed.

Acknowledgements

The addition of lakes to QSWAT+ was inspired by the development of SWAT2Lake by Eugenio Molina Navarro, Anders Nielsen and Dennis Trolle of Aarhus University . The author is grateful for their assistance in this work. He is also grateful to Juan Sebastián Acero Triana for supplying useful example data.

12. Parameter settings

A range of parameter settings is possible using the parameters form (Figure 63) that is opened by clicking the *QSWAT+ parameters* button on the main form (Figure 2).

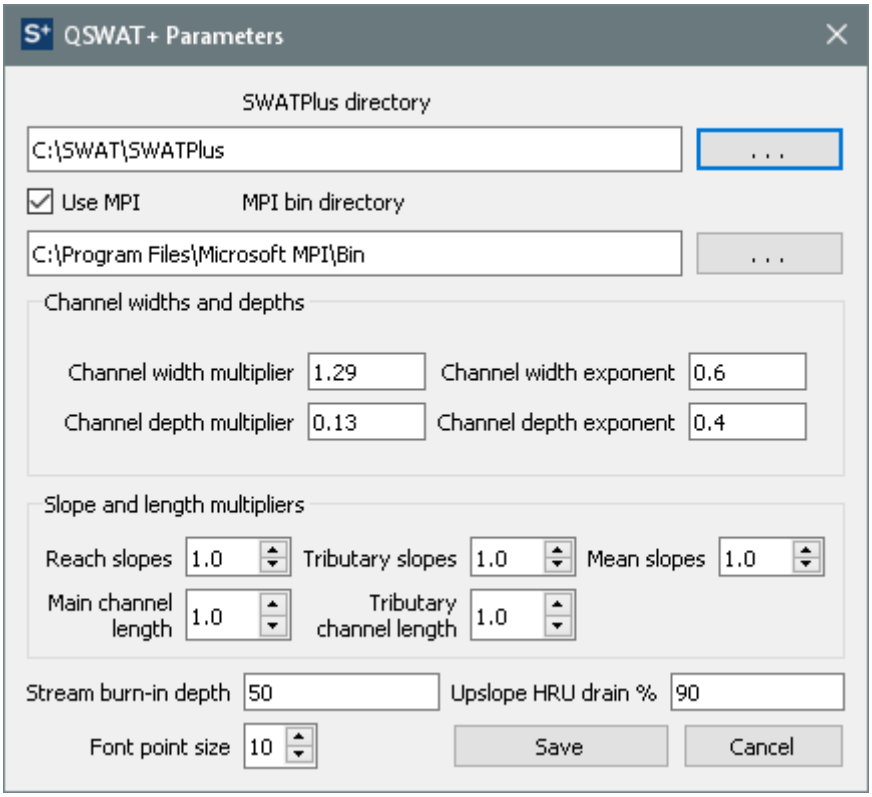


Figure 63: QSWAT+ parameter form

12.1 SWATPlus directory

This directory holds a number of directories needed for QSWAT+ and SWAT+:

- *Databases* contains template project and reference sqlite databases, plus soil and weather generator databases (if installed)
- *Documents* contains documents like this one
- *ExampleDatasets* currently contains just the *Robit* example used in this manual.
- *TauDEM5Bin* contains TauDEM executables used for watershed delineation. This directory is not necessary if QSWAT is also installed, since QSWAT has its own set of the same executables
- *Tools* currently contains *ConvertFromArc*, a tool to convert ArcSWAT projects to QSWAT+, and *SWATGraph*, a stand-alone version of the QSWAT+ feature for plotting SWAT+ outputs stored in csv files. SWATGraph is not necessary if QSWAT is installed.

The directory defaults to C:/SWAT/SWATPlus in Windows, and is used by the QSWAT+, SWAT+ Database and SWAT+ Editor installers. It is possible to install elsewhere (but you should use the same directory for all installers) and then use this form to tell QSWAT+ the location.

Setting this parameter sets the value for all QSWAT+ projects.

12.2 MPI bin directory

See Appendix IV: Installing and Using MPI. MPI is a tool for reducing the processing times with large DEMs.
Setting this parameter sets the value for all QSWAT+ projects. The remaining parameters described in this section are project-specific.

12.3 Channel widths and depths

QSWAT+ uses standard formulae for estimating the widths and depths of channels:

$$width = WM * DA^{WE}$$

$$depth = DM * DA^{DE}$$

where *DA* is the area draining into the channel in square km, *WM* is the *channel width multiplier*, *DM* the *channel depth multiplier*, *WE* the *channel width exponent*, and *DE* the *channel depth exponent*. The default values used for these four parameters are shown in Figure 63. The defaults are intended to provide typically appropriate values, but may not give good estimates in your area. Changing these values and clicking *Save* will cause the values to be saved and used for calculating channel widths and depths, and also saved in the project file for use in the future. Restarting the project will then place the new values in the parameters form if it is opened again.

The widths and depths so calculated are stored in the project database tables *gis_channels* and *gis_lsus*. In *gis_channels* they are the values called *wid2* and *dep2*, the *main channel* width and depth, and the area used is the total area draining into the channel. In *gis_lsus* they are the values called *wid1* and *dep1*, the *subsidiary channel* width and depth, and the area used is the area of the channel's LSU.

If either or both of the tables *gis_channels* and *gis_lsus* exist when *Save* is pressed the tables are updated. It is therefore possible to complete delineation or HRU creation and then update these tables, without having to rerun delineation or HRU creation.

The values in the database table *gis_channels* are also stored in the attributes of a shapefile *rivs1.shp*, which can be found in the *Watershed/Shapes* directory. This is updated as the database table is updated. Similarly, *lsus2.shp* in the same directory has the values from *gis_lsus* in its attributes, and is updated with the table. . (The use of the number 1 or 2 in these shapefile names reflects their formation. Number 1 is created from reading files for subbasins, landuse, soil, etc., and a new version with number 2 is added for lsus since channel merging changes the shapes, which show the areas draining into channels.)

12.4 Slope and length multipliers

Mean slopes and channel lengths are calculated as part of delineation. DEMs are obviously only small samples of the actual terrain elevations, and so these values may poorly estimated. They can be changed individually in the relevant project database tables, but it may also be useful to correct a persistent bias by multiplying the estimated figures by a constant. There is a range of multipliers that can be adjusted from their default values of 1.0:

- *Reach slopes* multiplies *slo2* values in *gis_channels*, the mean slope of the channel, which is the difference in elevation between the ends of the channel divided by its length.
- *Tributary slopes* multiplies *csl* values in *gis_lsus*. This is calculated as the longest flow path in the LSU divided by the drop along this flow path.
- *Mean slopes* multiplies the *slo1* values in *gis_subbasins*, and the *slope* values in *gis_lsus*, and *gis_hrus*. In each case the slope is a mean of the slope values recorded for every DEM pixel in the subbasin, LSU or HRU respectively. The *sll* (slope length) values in *gis_subbasins* are also adjusted, since they are calculated from the subbasin's mean slope.
- *Main channel length* multiplies *len2*, the channel length (and also divides *slo2*) in *gis_channels*.
- *Tributary channel length* multiplies *len1*, the longest flow path (and also divides *csl*) in *gis_lsus*.

As with the width and depth parameters described above in section 12.3, when these values are changed and *Save* is clicked, if the relevant tables already exist they are immediately updated.

Also, again as mentioned in section 12.3, *rivs1.shp* in *Watershed/Shapes* contains values from *gis_channels*, and *lsus2.shp* values from *gis_lsus* and are updated together with the tables. *subs1.shp* has a similar relation to the table *gis_subbasins*.

12.5 Font point size

This enables users to select a larger or smaller font size for QSWAT+. It only takes effect if *Save* is clicked and QGIS (and QSWAT+) restarted. An attempt is made to immediately show the effect in the *Font point* size label, but this is not always accurate. The result is dependent on the font sizes available.

12.6 Stream burn-in depth

This is the depth in metres by which elevations are decreased along the path of the streams to be burned in. Changing this parameter and clicking Save only has effect the next time the delineation step is done (with the burn-in option selected).

12.7 Upslope HRU drain %

When a floodplain is included in calculating LSUs and HRUs, HRUs in upslope LSUs have their outflowing water routed by default 90% to the channel (or reservoir or pond if the channel forms one) and 10% to the floodplain LSU. The figure of 90 is changed by this parameter, provided it is an integer in the range 0..100 inclusive, and internally the figure of 10 changed so their sum is 100. This affects the relevant entries in the *gis_routing* table. The table is adjusted immediately if it exists when *Save* is clicked.

13. Exporting SQLite database tables

Clicking the **Export Table** button on the main form (Figure 2) opens the **Export SQLite table to csv** form (Figure 64). You can use this form to create a csv (comma separated value) file from any SQLite database table. Then, perhaps after editing, you can import the form into its original table or to another database (which will need to have the table data structure already defined). This is mainly intended to support the import of csv files into QSWAT+ projects. QSWAT+ supports the import of csv files for landuse lookup tables, soil lookup tables, usersoil tables, plant tables and urban landuse tables: see section 7.2.1. .

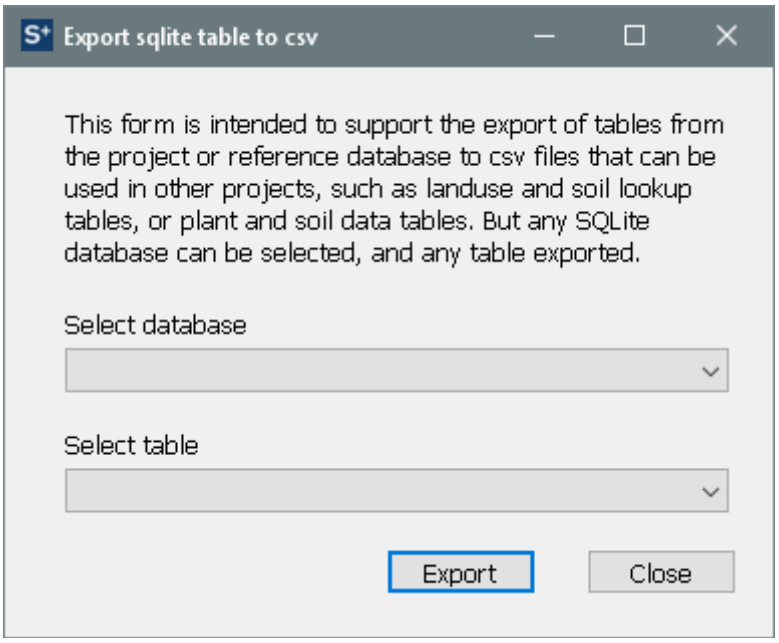


Figure 64: Export SQLite table to csv form

To export a table, first use the **Select database** pull-down menu, which offers the *Project database*, *Project's reference database*, and *Other SQLite database* as options. The first two refer of course to the current project, and the third allows the user to browse for any SQLite database. The second pull-down menu will then contain all the table names from the selected database, allowing the user to select one. Clicking **Export** will then save the database table to a csv file, with the user selecting its name and location. Clicking **Close** closes the form.

SQLite tools like SQLiteStudio also support export of tables, but this QSWAT+ feature is intended as a convenience for users who do not wish to spend the time to acquaint themselves with such tools.

14. Appendix I - Adding your own data

14.1 Adding weather generator data

We need to create and populate tables *weather_wgn_cli* and *weather_wgn_cli_mon* in the project database. The first contains information about the (simulated) weather generator station, name, location, etc., and the second has the data for each month.

weather_wgn_cli will be added using *Robit/WGEN_Robit_stat.csv, and weather_wgn_cli_mon* using *Robit/WGEN_Robit_mon.csv.* This section shows how tables can be imported from .csv files..

WGEN tables define data for simulating weather using a weather generator. They are used by SWAT+ when observed data is missing (either entirely, or on particular dates). The tables define data for one or more simulated weather stations. The SWAT+ Editor will assign the same station to all subbasins if there is only one in the table, or find the nearest to each subbasin if there are more than one.

1. Use SQLiteStudio to open *Demo1.sqlite* in the project folder. The database is added to SQLiteStudio using the *Database* menu, item *Add a database.* See Figure 65.

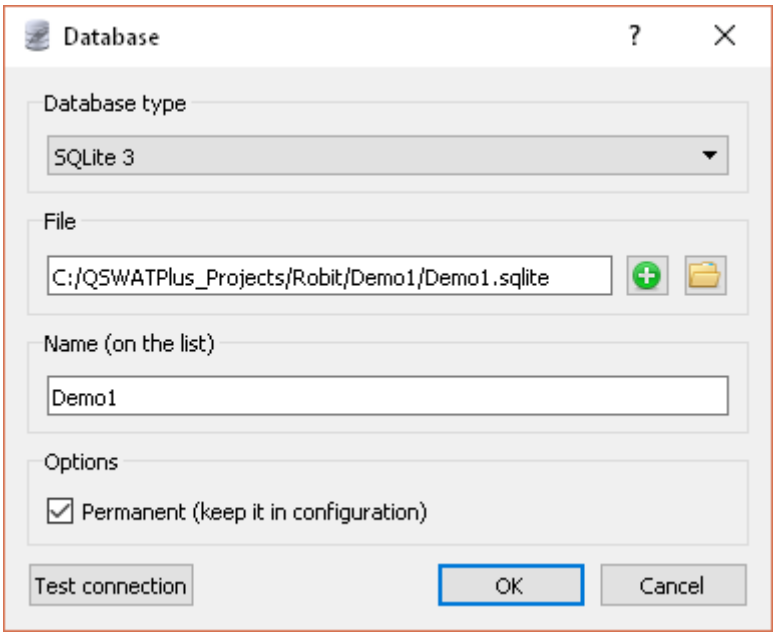


Figure 65: Adding a database

2. Select the database and open it, by double clicking on its entry in the list of databases, or by selecting it and using the open database symbol above. This will show the list of tables.
3. To import *WGEN_Robit_stat.csv* we first need to create or overwrite a table with the right structure - field names and types. If you have already started to use the SWAT+ Editor on this project, you may find that *weather_wgn_cli* and *weather_wgn_cli_mon* already exist. If so, skip to step 5 below. We have *WGEN_user*. We could overwrite it, but we will copy it as we need a new name anyway. Right click on *WGEN_user* and select **Create similar table**. The structure appears in the main panel. Type *weather_wgn_cli* into the **Table name** box and click the green tick box just above. You are presented with the SQL query that will create the table. Click **OK**.
4. In the same way, create *weather_wgn_cli_mon* as a copy of *WGEN_User_mon.*

5. Right click on *weather_wgn_cli* which has just appeared in the left panel and select **Import into the table.** Check the database name is *Demo1* (which may have an extra digit if you have loaded other databases with this name) and the table name is *weather_wgn_cli* and click **Next**.
6. In the new form, check the type is *CSV*, browse to *Robit/WGEN_Robit_stat.csv* for the input file, click the box to ignore errors, click the box to confirm that the first line is column names, leave the field separator as comma, and don't change anything else. See Figure 66 Click **Finish**. You will probably see a message about failure to import data row 2. This is caused by a blank last line: there is only one data row in this file. This was why we chose to ignore errors.

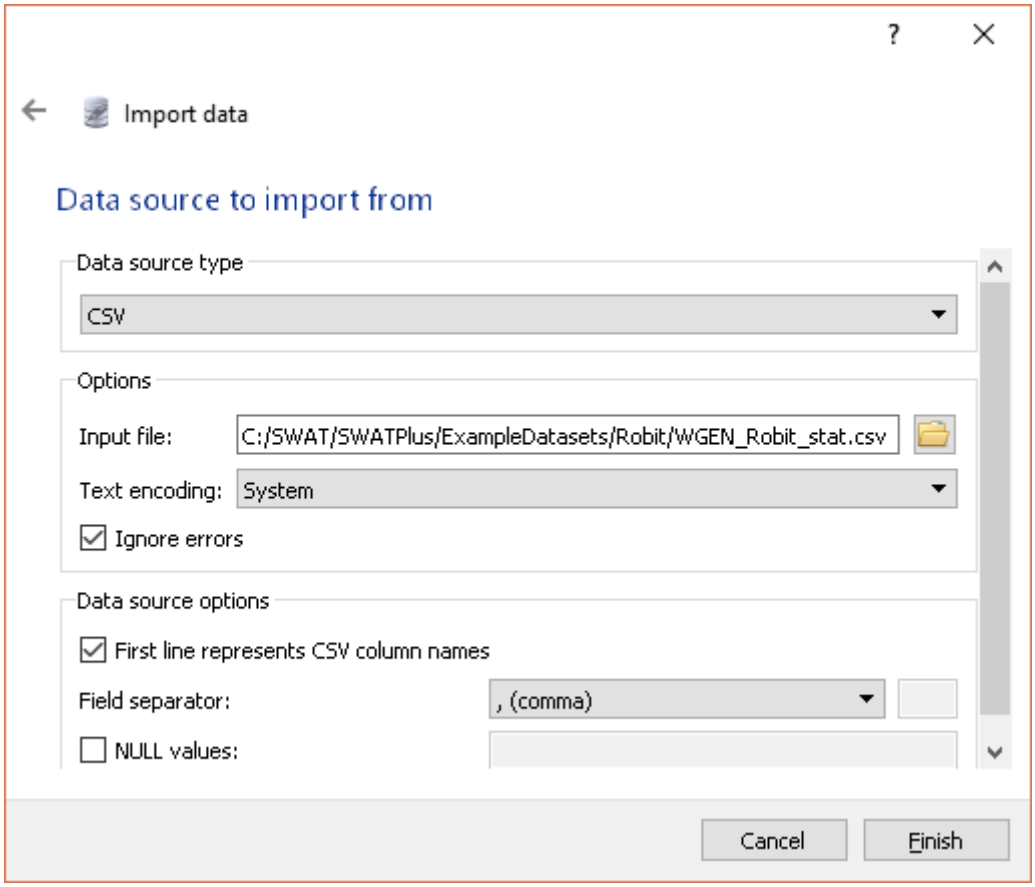


Figure 66: Importing WGEN_Robit_stat.csv

7. The Status panel should show that the import was successful. You can click on the Data tab above the main panel to see the imported data - there should be just one line as there is only one station. You may also need to click the blue button with circular arrows to refresh the data.
8. In the same way, import *Robit/WGEN_Robit_mon.csv* into *weather_wgn_cli_mon*. This time there will be 12 data rows, one for each month.

14.2 Preparing daily climatic data for QSWAT+

1. The user should initially prepare station information. The station information should be prepared for all the climatic variables for which the user has local data. The header for the table should consist of columns ID (station ID), NAME, LAT (latitude), LONG (longitude) and ELEVATION. Refer to the example dataset in the folder *Robit/ClimateRobit*, e.g. *pcp.txt*.

2. The actual climate data should have the same file name as the station NAME in the station data (e.g. station name robpcp corresponds to file *robpcp.txt*). The format of the climate data should be as follows: the first line should be the date in the format yyyymmdd followed by the daily climatic data values. Missing values will be replaced by -99. The data format for the precipitation, solar radiation, wind speed and relative humidity is the same. Refer to the example dataset for precipitation data in the folder at *Robit/ClimateRobit*, *robpcp.txt*. The only exception is for maximum temperature and minimum temperature, in which case the maximum temperature and minimum temperature are separated by a comma. Refer to the example dataset for temperature dataset, *Robit/ClimateRobit/robtmp.txt*.

3. The actual climatic data should be saved in the same folder as the station files.

4. The units for precipitation, temperature, solar radiation, wind speed and relative humidity should be mm, degree centigrade, MJ/m², m/s, and decimal fraction, respectively.

5. For each subbasin, SWAT+ uses observed climatic data from the closest station. If there is no observed station climatic data, SWAT+ will use simulated data using the weather generator. The algorithm used by SWAT+ is as follows: for example, suppose there are weather stations with identifiers A, B and C, and tables Apcp, Bpcp, Atmp, Btmp, and Cslr. Then the result will be:
   - For precipitation: subbasins will use measured data from Apcp or Bpcp, according to whether station A or B is closer.
   - For temperature: subbasins will use measured data from Atmp or Btmp, according to whether station A or B is closer.
   - For solar radiation: all subbasins will use measured data from Cslr.
   - For relative humidity: all subbasins will use simulated data.
   - For wind speed: all subbasins will use simulated data.

This makes it easy for users to vary climate data if they wish, and to combine measured data from a number of sources. It is even possible to combine simulated and measured data for different subbasins: in the example above, for example, if all the values in Apcp are -99 (missing data), while Bpcp is measured data, then subbasins nearer to station A will use simulated precipitation data and those nearer to B will use measured data.

14.3 Preparing landuse and soil tables

For each landuse or soil map the user can prepare a lookup table like *Robit_landuses* or *Robit_soils* and put it into the project database *Demo1.sqlite* in the *Demo1* folder (*C:/QSWATPlus_Projects/Robit/Demo1*). The project database is created by copying *QSWATPlusProj2018.sqlite* when *New Project* in the main QSWAT+ form is clicked.

Standard landuse and soil tables, namely *global_landuses* and *global_soils* are already included in the project database. These are for use with the global landuse and soil maps available from the WaterBase website on its data page http://www.waterbase.org/download_data.html. If you use these maps then you can use these lookup tables, and the existing tables *global_usersoil*, *plant*, and *urban* in the project database are appropriate. The global maps are in latlong projection and will need clipping and reprojecting to your project's projection as described for global DEMs in Appendix II.

In the case of a landuse map, the lookup table should have the string *landuse* in its name. Then it will be offered as an option for a landuse table. It must have the same structure as the table *global_landuses* in *QSWATPlusProj2018.sqlite*. So it must contain at least the columns LANDUSE_ID (type INTEGER) and SWAT_CODE (type TEXT). The LANDUSE_ID corresponds to the values in the landuse grid. It is possible that more than one of the LANDUSE_IDs maps to the same SWAT_CODE, where your data makes more distinctions than are supported by SWAT+. The SWAT_CODE strings are mostly 4 letters long, but can be up to 16 characters, and all the ones used in your map must be found in the table *plant* or the table *urban*, or in your own replacements for either or both of these in the project database. Searching for these codes is independent of case, so that for example *RICE* in your lookup table will match *rice* in the *plant* table.

In the case of a soil map, the lookup table should have the string *soil* (but not *usersoil*) in its name. Then it will be offered as an option for a soil table. You should copy the structure of the table *global_soils* in *QSWATPlusProj2018.sqlite*. So it must contain at least the columns SOIL_ID (type INTEGER) and SNAM (type TEXT). The SOIL_ID corresponds to the values in the soil grid. All the SNAM strings you use must be as follows:

- found as *SNAM* values in the table *global_usersoil* found in the SWAT+ project database, or the table you have chosen to use instead, or
- found as *name* values in the first *usersoil* table if you have used the two table option: see below, or
- if you are using STATSGO, one of the following four options:
  1. *muid* values such as AL001
  2. *muid+seqn* values such as AL001+1
  3. *muid+name* values such as AL001+DICKSON
  4. *s5id* values such as TN0042

  The lookup table must use one of these options consistently. Lookup is in the *statsgo* table in *swatplus_soils.sqlite*. Where lookup target matches more than one row in the *statsgo* table the first (as ordered by increasing *id* values) is taken. So the four examples above would all give the first row (*id* 1) in the table

Unless your project is in the USA and you are using SSURGO or STATSGO soil data, you will need a *usersoil* table to define your soil properties. QSWAT+ uses a structure which divides soil properties into two tables: a *usersoil* table giving properties for the whole soil, such as its hydrographic group and texture, and a *usersoil_layer* table giving properties for each layer. But QSWAT+ also accepts the old SWAT *usersoil* structure in which these are combined. You can use tables already provided in the project database, like *global_usersoil*, or you can create and select your own table. The table name should include the string *usersoil*, and if there is a separate layer table its name should be the first table name extended with *_layer*. The structure of the two tables is illustrated by the csv files in the Robit dataset *Robit_usersoil_main.csv* and *Robit_usersoil_main_layer.csv.*

An alternative to preparing these tables in the project database is to use .csv (comma separated value) files as described in section 7.2. Note that the .csv file *Robit_usersoil.csv* used the single usersoil structure. If you choose *Use csv* in the usersoil menu, you can select *Robit_usersoil_main.csv* as the usersoil table, and you will then be prompted for the second layers .csv file, and tables *Robit_usersoil_main* and *Robit_usersoil_main_layer* will be created in the project database.

15. Appendix ll – preparing global DEM data for QSWAT+

15.1 Introduction

You may have your own digital elevation map (DEM) for your watershed, but if you don't this section explains how to obtain and prepare DEM data obtainable free from the web. The techniques for clipping and reprojecting DEMs may also be useful for DEMs from other sources.
There are also notes about masking DEMs and about dealing with large DEMs in the final two sub-sections (7.7) that should be of general interest.

15.2 Obtaining SRTM DEM data

SRTM (Shuttle Radar Topography Mission) is a good source of DEM data for almost anywhere in the world. It is available at 90 metre and, since the end of 2014, 30 metre resolutions.

The first thing to do is to make a note of the extent of the watershed you want to model: the maximum and minimum latitudes and longitudes. Decimal longitudes that are west of the Greenwich meridian should be expressed as negative numbers: if you have a longitude above 180°, subtract 360. Decimal degrees, rather than degrees, minutes and seconds, will be required for clipping later, so convert yours if necessary (just divide the minutes by 60 and round to 1 decimal place) and make a note of them.

Choose or create a folder for the maps to be stored. We will refer to this as the *map folder*.

15.2.1 90m data

You might like to use the less detailed 90m data if your watershed is large or if you are new to QSWAT+. Go to http://srtm.csi.cgiar.org/, and select the *SRTM Data Search and Download* menu item. Then select *GeoTiff* as the file format. You can now select tiles by selecting *Multiple Selection* and clicking on them, or by selecting *Input Coordinates* and entering the extent. Then click *Begin Search* and on the next page select the *Data Download (FTP)* or *(HTTP)* for each of your tiles. You will not need the mask. Each tile comes in a zipped archive file containing a text file with information about SRTM (including citation data) and a .tif file, plus .hdr and .tfw files. Unzip each archive and put the files in the map folder.

15.2.2 30m data

Go to http://earthexplorer.usgs.gov/ and register if it is your first visit, or log on. You won't be able to download anything until you are logged on.

Under *Coordinates*, select *Decimal* and click *Add Coordinate*. Enter, the latitude and longitude of one of the 4 corners of the extent. Add the other 3 corners similarly. Then above *Enter Search Criteria* select the second tab, *Data Sets.*

Find *Digital Elevation*, click on the plus sign to expand and find *SRTM*. Expand that and check *SRTM 1 Arc-Second Global*. Then go straight to the fourth tab, *Results*.

After a few seconds you should see the tiles you need listed. For each one. Click on *Download options* (the 5th symbol) and then click *Download* beside *GeoTIFF*. *Save* the file, close the *Download Options* form, and repeat for each of your tiles. Note that the save form says the file is a zipped archive, but its extension is .tif and it is not zipped. Move the .tif files to the map folder.

15.3 Clipping the grids

Start QGIS and load the first grid (using *Layer → Layer → Add Raster Layer*, or the corresponding button on the left of the QGIS window). Select *Raster → Extraction → Clip Raster by Extent* to bring up the clip form (Figure 67). Set the *Input layer* to the grid you just loaded, set the *Clipped (extent)* by browsing to where you want to store the result, and use something like <grid_name>_clip_ll.tif as the file name. It is a good idea when manipulating grids to set the nodata value, so include in the nodata box -32768 for 90m data and -32767 for 30m data[8]. You can find the appropriate value by right clicking on the entry in the *Layers* panel, selecting *Properties*, and selecting the *Transparency* section (or look under *Bands* in the *Information* section.)

You also need to enter the extent you used to download the grids. It is a good idea to widen these somewhat, say by 0.2 of a degree, to make sure there are no problems. Make a note of the values you use as you may need them again. Check *Open output file after running*

8 The value should not matter, as long as it is less than any occurring in the actual data, but if it is different from the existing no data value it can cause problems with the existing value being accepted as an actual data value.

algorithm and click *Run*. Right click on the original, unclipped grid in the *Layers* panel, select *Remove Layer* and confirm. You will be left with the clipped portion. Load each of the other grids, and clip in the same way.

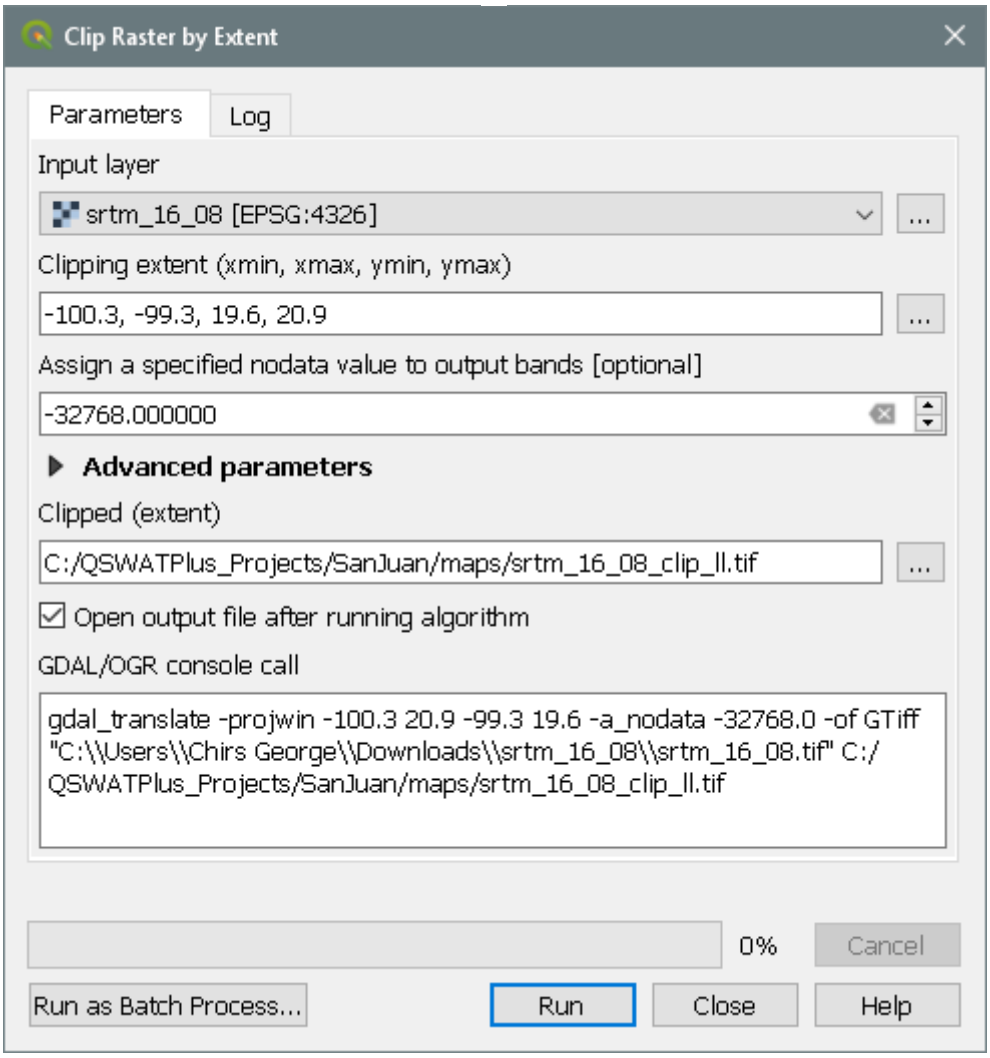


Figure 67: Clip form

15.4 Merging the grids

You should have all the clipped grids loaded. Use *Raster → Miscellaneous → Merge* to open the *Merge* form (Figure 68). Select all the layers you want to merge. Ignore the two check boxes. Check the type of one of the clip files by right clicking on its legend entry, selecting *Properties -> Information:* the type is the **Data type,** probably Int16, and select the *Output data type* accordingly. For *Merged*, browse to the map folder and choose a suitable name, perhaps ending in dem_ll to remind you what it is Check *Open output file after running algorithm* and click *Run*.

Check your merged DEM is OK by removing the visibility of the other layers.

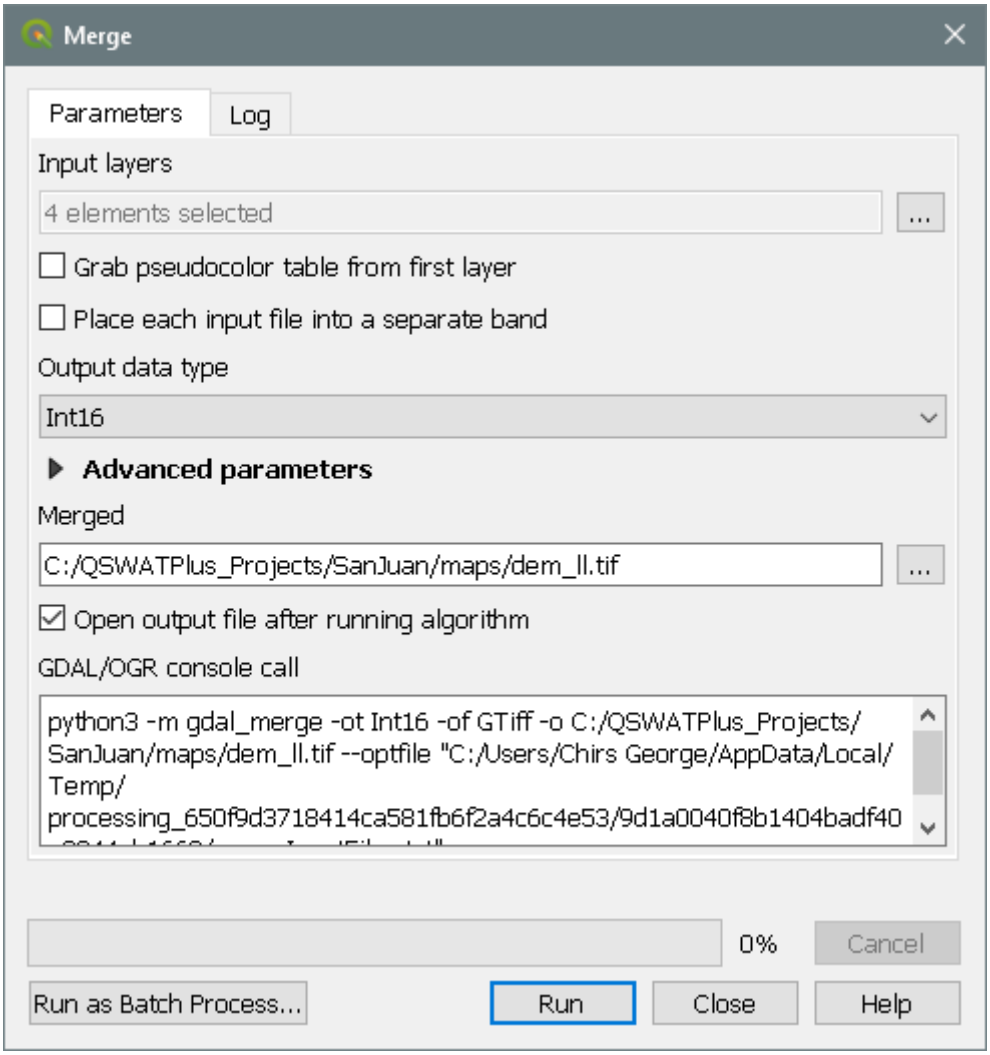


Figure 68: Merge form

15.5 Reprojecting the grid

Our final task is to reproject the merged grid into an "equal area" projection, i.e. one in which each grid cell has the same area. The current lat-long projection is not appropriate because at different latitudes the same amounts of longitude will represent different distances. Universal Transverse Mercator (UTM) is a good general choice for the new projection. It is available almost everywhere, and while not exactly an equal area projection is sufficiently close for most SWAT modelling. We need a suitable UTM zone. We can find the zone from a map like http://www.dmap.co.uk/utmworld.htm, or use the formula *(x + 180)/6* rounded up to the next integer, for a reasonably central longitude *x* in our area. So, for example, if *x* is -100 the UTM zone will be 80/6, or 13.3, rounded up, to give 14. Then we note whether we are in the northern or southern hemisphere.

Use *Raster → Projections → Warp (Reproject)* to open the *Warp (Reproject)* form (Figure 71). Select the merged grid as the *Input layer* from the pull-down menu. You can leave the *Source CRS* unselected as it will default to the lat-long projection of the merged raster. The *Target CRS* will need to be set to our chosen UTM projection. This is unlikely to be in the pull-down menu unless you have used it recently, so click the *Select CRS* button to the right

to bring up the *Coordinate Reference System Selector* (Figure 69).

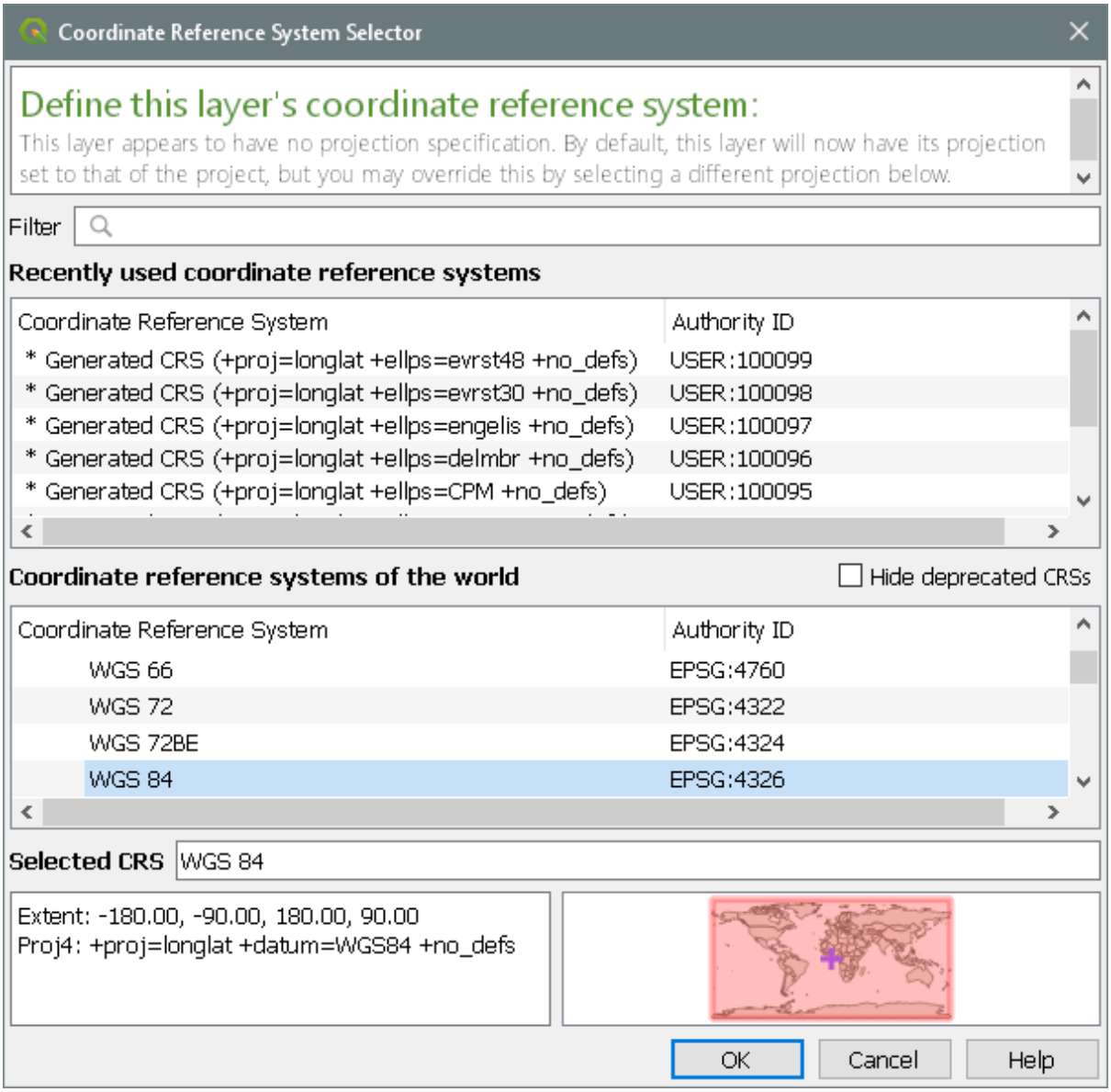


Figure 69: Coordinate Reference System Selector

For zone 14 in the northern hemisphere we need to find WGS84 / UTM zone 14N. If we have used it recently it will appear in the upper *Recently used* box. Otherwise we have to find it in the lower box. It will appear under ***Projected Coordinate Systems*** rather than ***Geographic*** and can be found much closer to the bottom than the top. However, there are a lot of projections and it can be tedious to find. Instead we use the fact that a WGS84 / UTM zone *nn* in the northern hemisphere has an EPSG number 326*nn*, while the corresponding zone in the southern hemisphere has an EPSG number of 327*nn*. So if you know the zone and the north or south hemisphere you can calculate the EPSG number. Type 32614 in the *Filter* box and the one we want will immediately appear in the middle panel. Select it there, and it will appear as the **Selected CRS** at the bottom, along with its extent and its location shown on the world map (Figure 70). Click *OK*.

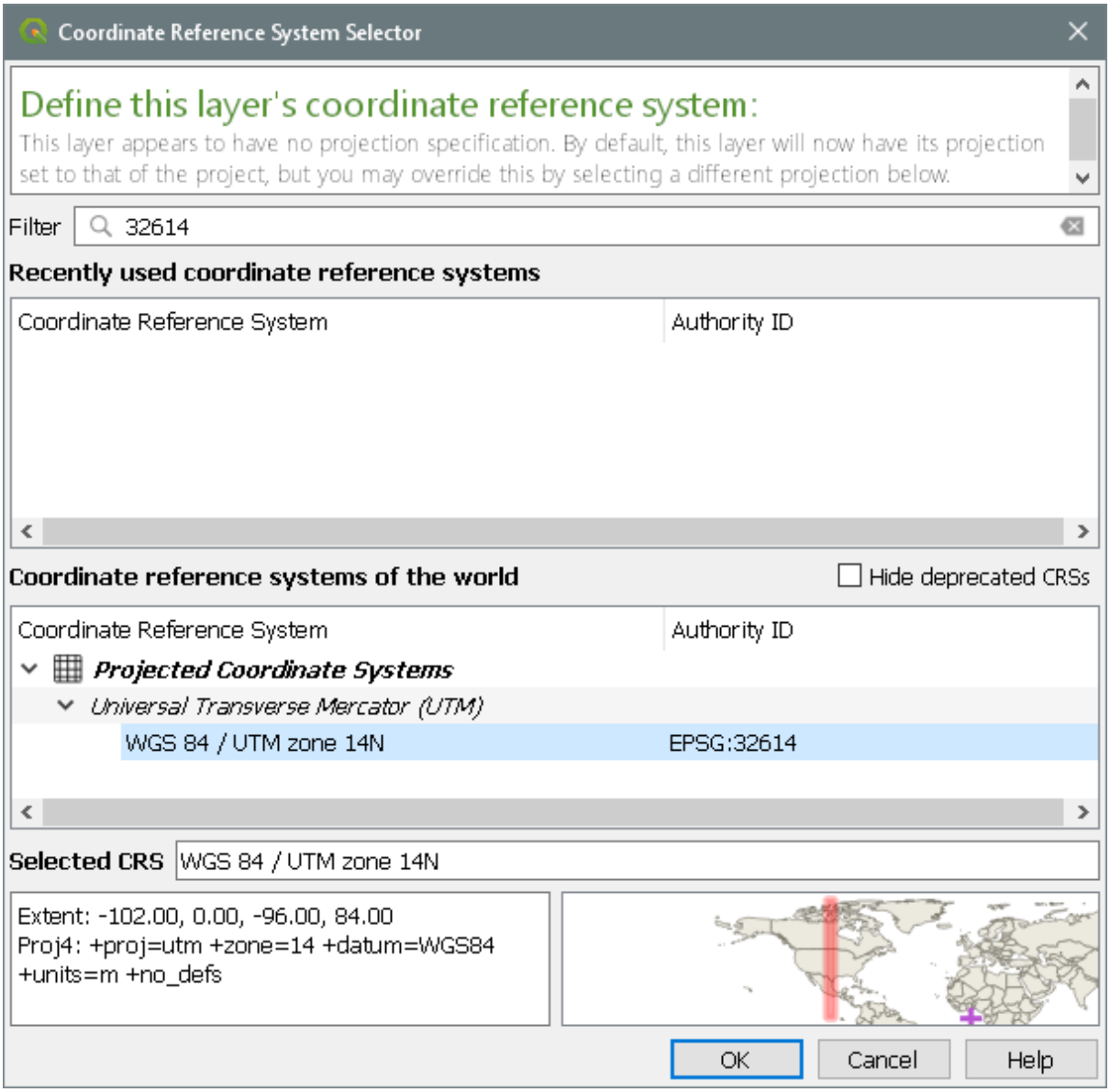


Figure 70: Coordinate Reference System selected

Leave the *Resampling method* as *Nearest neighbour.* Since we are keeping the original resolution there will be little resampling, if any. Set the nodata value: -32768 for 90m, or -32767 for 30m. Set the *Output file resolution* to 90 for 90m SRTM, 30 for 30m. This will ensure that the output has square pixels of the appropriate size.

For *Reprojected*, browse to the map folder and type in a suitable name, perhaps ending in *dem_utm* to remind you what it is

Uncheck *Open output file after running algorithm*: the projection is different from the current lat-long projection and should be viewed in a new QGIS window. Click *Run.*

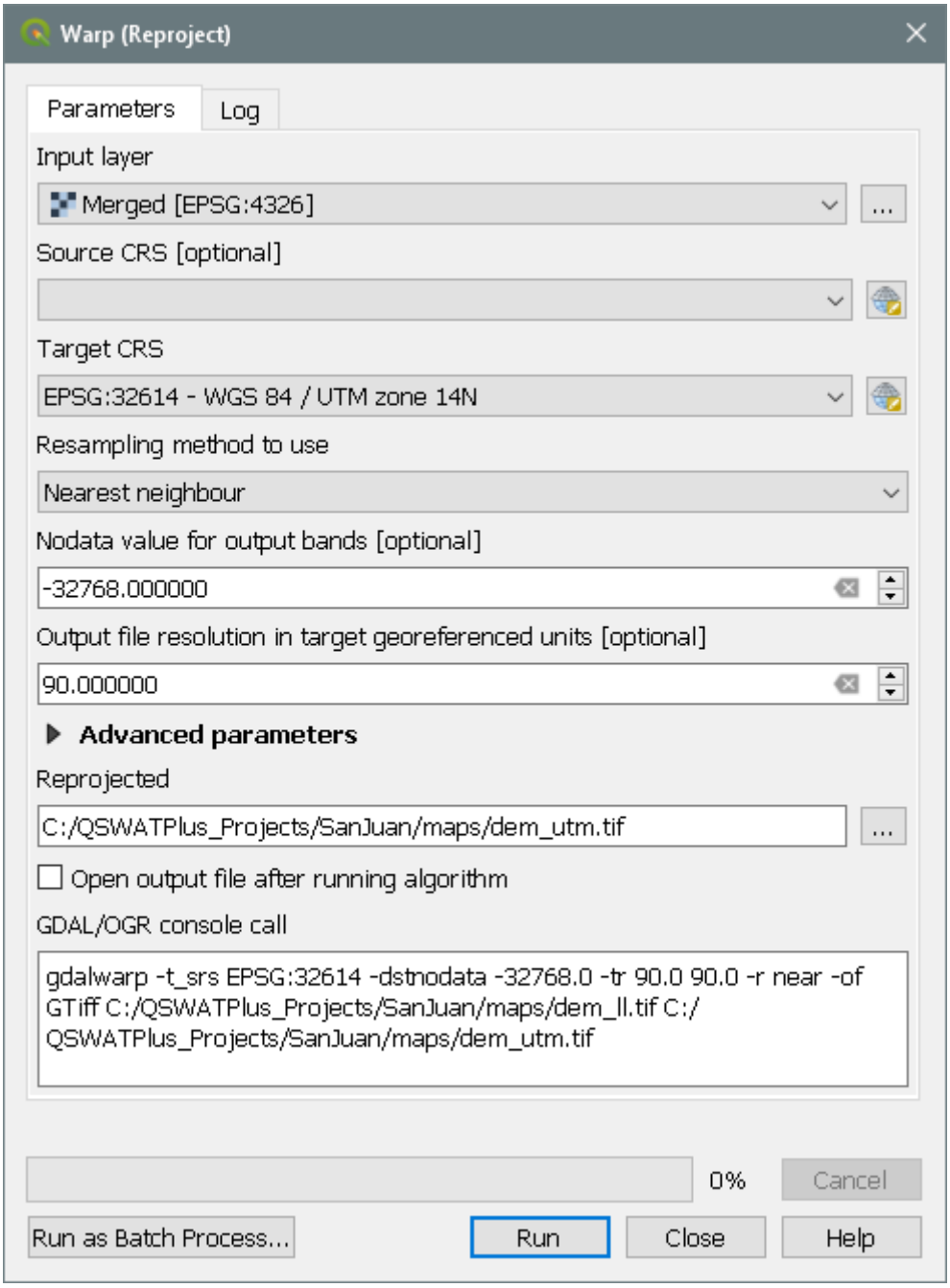


Figure 71: Reprojection form

Start a new QGIS instance (leaving the old one in case you find you made an error and need to redo something). Load the dem_utm file and check it looks OK. Note that coordinates in the box at the bottom are now shown as metres instead of degrees (you won't recognise the numbers but you will see the difference from the previous instance of QGIS). If all looks well close the previous instance of QGIS. Choose *Discard* when asked if you want to save the project.

15.6 Masking a DEM

QSWAT+ does not itself provide a facility for masking a DEM, because the capability is already in QGIS. Before showing how to mask, a note about when to do it. Masking, or reducing the area of the DEM to be delineated, is done for two reasons:

- To reduce the amount of time that delineation takes.
- To improve the accuracy of delineation.

The first reason is much less true now than it was some years ago because of the improved speed of computers. Unless your DEM is very large and detailed it will typically take a few minutes at most to delineate, and as you don't often repeat the process (and QSWAT+ tries to reduce the amount of recalculation when it is repeated, as well as supporting multi-core processing) reducing the area more than the rough clipping described in this document is not likely to be worth doing. See section **Error! Reference source not found.** for more nformation about large DEMs.

Masking is also dangerous. If after delineation you find that part of your watershed boundary is on the edge of your DEM then it is quite possible that you have excluded some of the watershed, and it will be a good idea to reclip the DEM more generously and run again. So the general advice on masking your DEM is don't do it!

The second reason is much more compelling. DEMs are far from perfect representations of the real terrain, and delineation can easily include in the watershed areas which are known to be outside it. In this case, if you have a reliable map of the actual watershed boundary, you can use it to clip the DEM precisely.

So we assume you have a good map of the watershed boundary, which we assume is a shapefile with a single shape in it. We further assume it is in the projection of your final DEM: if it is in some other projection you can use Vector -> Data Management Tools -> Reproject Layer to create a new shapefile with the right projection.

We can now use this boundary shapefile to clip our DEM. Start QGIS, load the projected DEM, and then load the projected watershed shapefile (Figure 72) Check the shape lies within the DEM, or we made a mistake somewhere!

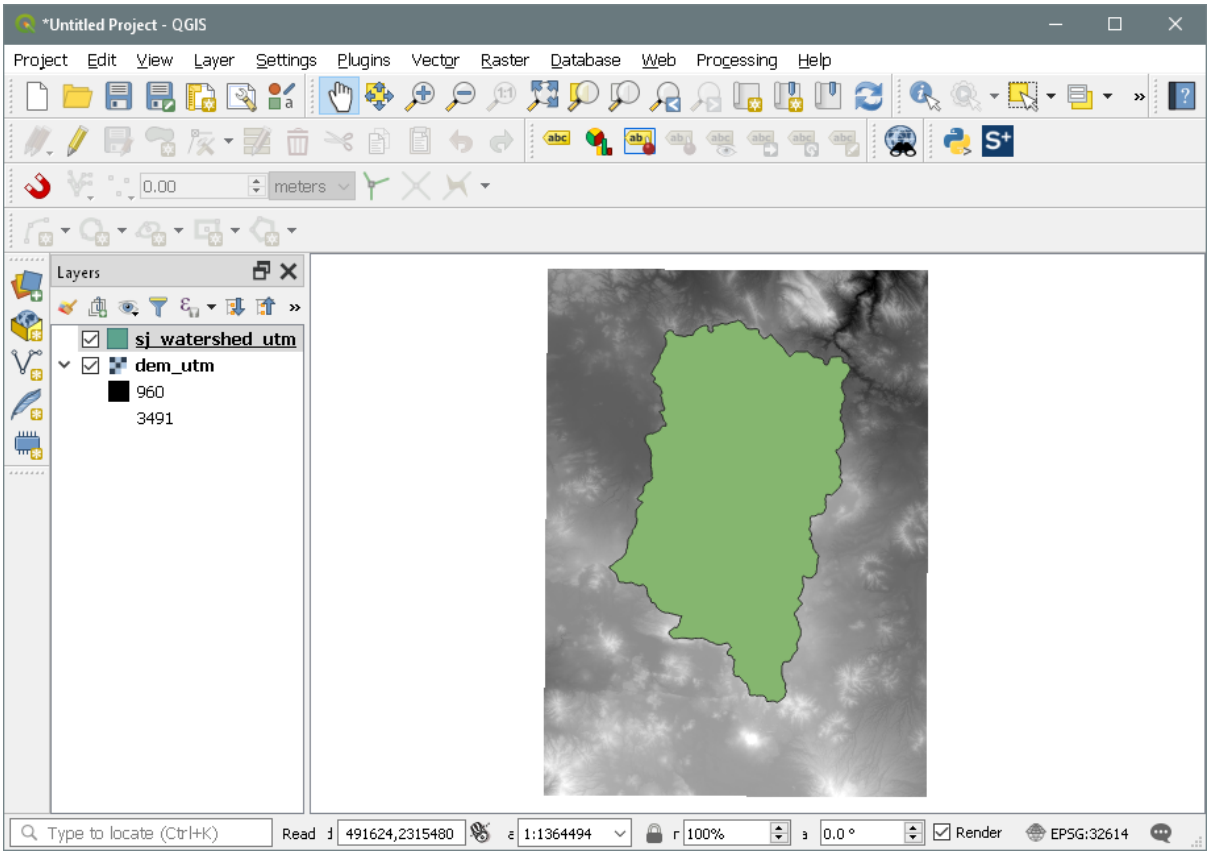


Figure 72: DEM and mask shape loaded

Then use *Raster → Extraction → Clip Raster by Mask Layer* to open the clip form (**Error! eference source not found.**). Select the dem_utm from the pull-down menu as the *Input layer*, and the shapefile layer as the *Mask layer* Choose the appropriate nodata value for the raster you are clipping, such as -32768 for a 90m SRTM. You don't need an alpha band, you should match the extent of the clipped raster to the mask (or you will just create a DEM of the same size but with a lot of nodata) and you should keep the resolution: these will probably all be the defaults. For *Clipped (mask)* browse to the map folder and provide a name like *dem_clip_utm*. Check *Open output file after running algorithm*. Click *Run*. Remove the dem_utm to check that we have a DEM clipped to the shape.

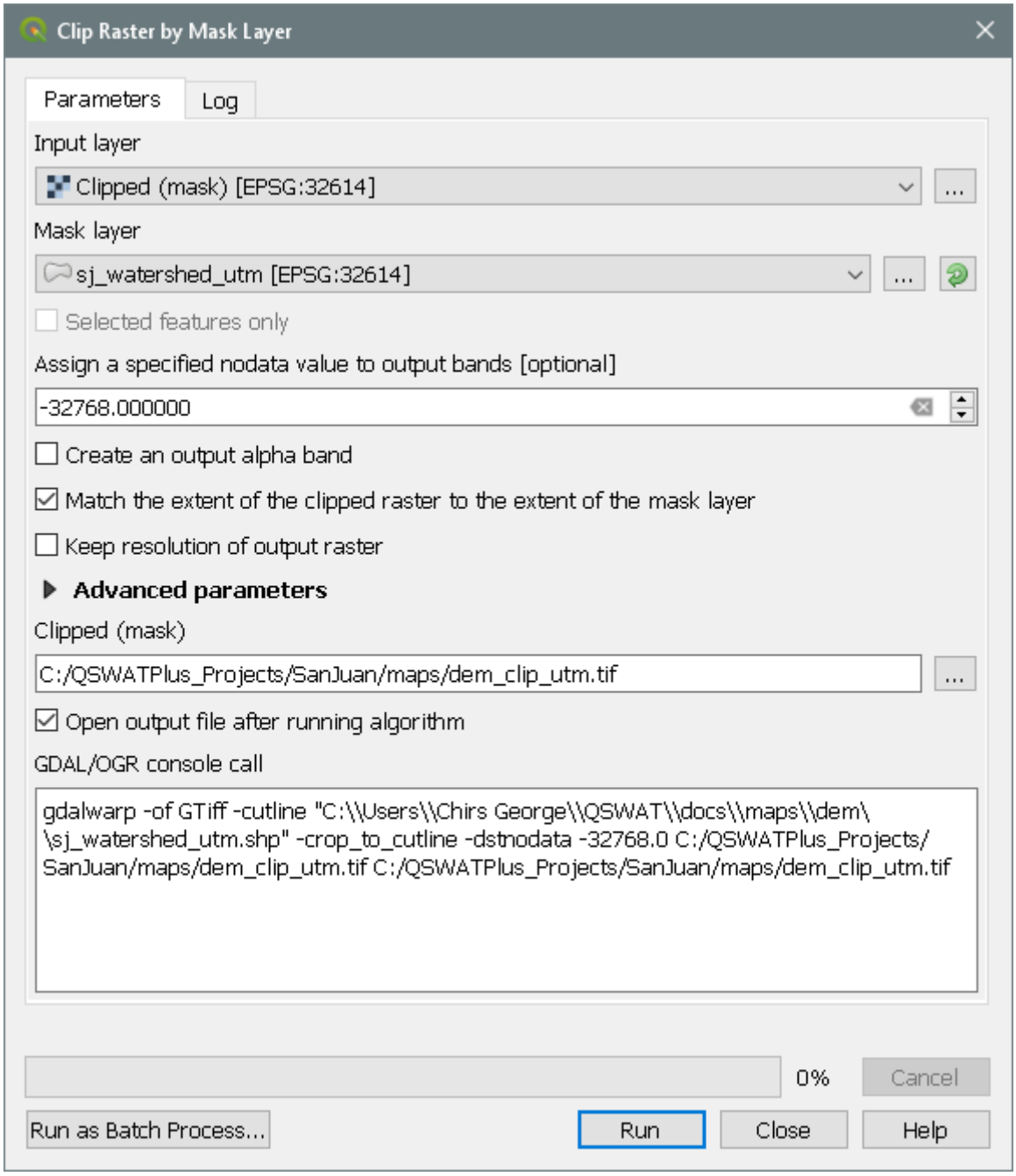


Figure 73: Clipping the DEM by mask

15.7 Large DEMs

Some processing times for large DEMs are shown in Table 1. These are not intended to be anything more than indicative: while the number of cells in a DEM is probably the main factor influencing processing time, it is not the only one. DEM sizes are in millions of cells: for example the 100 million DEM has 10000 rows and 10000 columns. You can see the size of your DEM by loading it in QGIS, right clicking the entry in the *Layers* panel and selecting *Properties*. The *Layer info* section under the *General* tab shows the number of columns and rows.

The times in Table 1 are for running from after loading the DEM to the end of stream delineation: this includes the two most long-running TauDEM functions (D8FlowDir and

DinfFlowDir). Note that these two functions only need to be rerun if the DEM changes. If you want to change the delineation threshold, or the inlets/outlets, the rerun will be much faster than these times. Where there is a second time in brackets, this is for running without using the Message Passing Interface (MPI), i.e. with a single process. The machines used to obtain these figures are:

A. A 32-bit laptop with an Intel T9400 2.53GHz processor with 2 cores and 2 GB of memory. MPI was used with 4 processes.

B. A 64-bit PC with an Intel i3 3.3GHz processor with 2 cores, 4 logical processors and 6GB of memory. MPI was used with 8 processes.

C. A 64-bit PC with an Intel i7 3.4GHz processor with 4 cores, 8 logical processors and 8GB of memory. MPI was used with 16 processes.

See Appendix IV: Installing and Using MPI..

Machine/ DEM size	A	B	C
100	2100(*)	255(560)	104(450)
25	*(480)	64(140)	28(111)
12	159(205)	28(61)	12(48)
6	42(57)	8 (16)	4 (13)
4	13(22)	3(5)	1(3)
2	7(9)	1(2)	1(2)
1	3(4)	1(1)	1(1)

Table 1: Times in minutes for processing large DEMs (sizes in millions of cells)

** - run failed*

There are two conclusions to be drawn from these figures:

- In this range of DEM sizes the times increase initially linearly with the size of the DEM, but later much more sharply. A DEM of 369 million cells processed on machine C took over 15 hours, much worse than the rough '1 minute per million cells' that you would expect for this machine from the table. Typically 'Not Responding' will appear at the top of the main QGIS window and/or the delineation form. This is a result of the TauDEM processes running in the background but using almost all the CPU resources. Ignore the lack of responsiveness and be patient!
- MPI only makes a substantial difference when the DEMs are above a few million cells.

16. Appendix III: Using a Predefined Watershed and Channel Network

The delineation form allows you to opt for a predefined watershed and channel network by selecting the *Use existing watershed* tab.

We deal first with non-grid models in section 16.1. Existing grid models were discussed earlier in section 10.2; we present precise requirements for them in section 16.2.

16.1 Non-grid models

On this tab, with *Use grid model* not checked, you can select the shapefiles containing the watershed and its subbasins (the *Subbasins shapefile*), the regions draining into channels (the *Watershed shapefile*) and the channel network (the *Channels shapefile*). Remember that channels include the streams that define the subbasins, so that the watershed shapefile is a finer partitioning of the whole watershed than the subbasins shapefile. These files need to have the same projection as the DEM. If you want to use files generated by TauDEM in a previous project, then such files can be found in the *Watershed/Shapes* subfolder of that project, with names ending *subbasins.shp, wshed.shp* and *channel.shp* respectively.

It is also possible, but not necessary, to specify an O*utlets/Inlets shapefile*, and again a file from the previous project may be used. If such a file is not specified, only main outlet(s) from the watershed will be used as main outlets in the new project. You will not be able to mark inlets, reservoirs or ponds, but you will later be able to add automatic reservoirs by setting an appropriate threshold for WATR areas.

The *subbasins shapefile* must be a polygon shapefile (each polygon representing a subbasin) with the following fields:

Necessary/Optional	**Name**	**Type**
Necessary	PolygonId	Integer
Optional	Subbasin	Integer

The *watershed shapefile* must be a polygon shapefile (each polygon representing the region draining into a channel reach) with the following field:

Necessary/Optional	**Name**	**Type**
Necessary	PolygonId	Integer

The *channels shapefile* must be a line shapefile (each line representing a channel reach) with the following fields:

Necessary/Optional	**Name**	**Type**
Necessary	WSNO	Integer
Necessary	LINKNO	Integer

Necessary	DSLINKNO	Integer
Optional	DSNODEID	Integer
Optional	Length	Double
Optional	Drop	Double
Necessary	BasinNo	Integer
Necessary for lakes	LakeIn	Integer or NULL
Necessary for lakes	LakeOut	Integer or NULL
Necessary for lakes	LakeWithin	Integer or NULL
Necessary for lakes	LakeMain	Integer or NULL

An *outlets/inlets shapefile* must be a point shapefile with the following fields:

Necessary/Optional	**Name**	**Type**
Necessary	ID	Integer
Necessary	RES	Integer
Necessary	INLET	Integer
Necessary	PTSOURCE	Integer

Note that the REC field has changed from QSWAT, where it had the value 1 to indicate a reservoir point, else 0. Now it can also take the value 2, to indicate a pond.

A *lakes shapefile* must be a polygon shapefile with the following fields.

Necessary/Optional	**Name**	**Type**
Necessary	ID	Integer
Necessary	RES	Integer

The RES field may be 1 (for a reservoir) or 2 (for a pond).

Necessary fields must be defined in the original files. *Optional* fields may or may not be defined originally. With two exceptions defined below, if optional fields are defined, and if the *Reuse* button is checked then their values will be assumed to be correct and reused. If optional fields are not defined originally, or if the *Recalculate* button is checked, then they will be defined if necessary, calculated, and the calculated values inserted[9]. Any file may have other fields defined: these will be ignored. The fields may be in any order.

The first exception is the DSNODEID field. This is *necessary* if an outlets/inlets shapefile is

9 Changes to these files are only made to the copies of the originals in the *Source* folder of the project.

specified, in which case it is assumed to be correct and its non-negative values must refer to ID values in that file. DSNODEID values are never recalculated.

The second exception is the Subbasin field in the subbasins shapefile. Even if present and *Reuse* is selected it is checked to have suitable values for the SWAT+ basin numbers. These must be strictly positive and unique, except for basins which are empty, when the Subbasin values must be zero. If the Subbasin values do not meet these requirements they are recalculated.

There are some conditions imposed on the fields. The PolygonId values in the watershed shapefile must be unique, non-negative, and must equal the corresponding WSNO values in the channels shapefile: each polygon must have exactly one channel reach, and each channel reach must have exactly one polygon. These two fields provide the connection between the watershed polygons and the channel reaches. The LINKNO values must also be unique and non-negative (and will typically but not necessarily be the same as the WSNO numbers.). The DSLINKNO (downstream link) values must each be another LINKNO value or -1, where -1 indicates a watershed outlet.

The BasinNo field in the channels shapefile provides the link between the channels and the subbasins: each BasinNo value must be a PolygonId in the subbasins shapefile, defining the subbasin within which the channel is located.

Files previously generated by TauDEM will meet all these conditions, and their values may be safely reused.

There is also an assumption made about the channels shapefile. Each channel reach is either a single line, represented as a sequence of points, or a collection of such lines. The first point in each line may represent the upstream (source) end and the last represent the downstream (outlet) end, or vice versa. TauDEM creates stream reaches with the first point at the outlet end, but either orientation is accepted by QSWAT+. The assumption is that the orientation is the same for all the reaches. The orientation is checked by finding a reach with an upstream or downstream reach where the ends of the reaches are within one dem cell of each other: normally such ends will exactly coincide. When a reach is a collection of lines, these must link together.

The last four entries for the channels shapefile are only necessary if one or more lakes are included. The values may be positive integers, in which case they must be lake identifiers in the lake shapefile (which needs a field LakeId defining the lake identifier), or 0, NULL or empty, indicating no interaction with a lake. At most one of the three fields LakeIn, LakeOut and LakeWithin may have a positive value, indicating that the channel flows into, out from, or entirely within the identified lake respectively. One of the LakeOut channels for each lake should have the same lake identifier in its LakeMain field, indicating that it is the main outlet for the lake. Subbasin and watershed shapefiles need no additional fields for lakes, but must cover only the region outside the lakes. See section 11.6.2.

The optional fields are (if necessary) calculated as follows:

Field	Calculation
Subbasin	For subbasins which are not empty and not upstream of an inlet, a unique positive integer. These will be the subbasin numbers for use by SWAT+. For subbasins which are empty or upstream of an inlet, 0.
Length	The total length in metres of the straight line segments forming the reach.
Drop	The absolute difference in metres between the elevations of the end points of the reach.

Note that if a predefined watershed is chosen then the length of the longest flow path along channels in the subbasin is used for the maximum flow length in that subbasin.

16.2 Grid models

Setting up an existing grid model was described in section 10.2. In this section we describe the requirements on the existing files used.

The *grid shapefile* is a polygon shapefile defining the square shapes that together define the watershed. Each such polygon is treated as a subbasin, and so has a single stream reach associated with it. With grid models there is no further subdivision based on channels: channel and stream are the same.

The grid shapefile has the following fields:

Necessary/Optional	Name	Type
Necessary	PolygonId	Integer
Optional	DownId	Integer
Optional	Subbasin	Integer

PolygonIds must be non-negative unique integers. DownIds are optional (and ignored) if drainage is defined by *Stream drainage* or by *Drainage table* but otherwise must be either another PolygonId value (indicating that drainage from this polygon is to the polygon with that PolygonId value) or -1 (indicating a watershed outlet). The drainage pattern so described must not be circular: following the pattern from any polygon must eventually lead to an outlet.

If subbasin values are included they will be used for the SWAT+ subbasin numbers if they are unique and positive. Otherwise they will be recalculated.

If drainage definition is by *Stream shapefile* then a stream shapefile is required, with the following fields:

Necessary/Optional	Name	Type

Necessary	WSNO	Integer
Necessary	LINKNO	Integer
Necessary	DSLINKNO	Integer
Optional	DSNODEID	Integer
Optional	Length	Double
Optional	Drop	Double

The *WSNO* values must be unique and each must be a *PolygonId* value in the grid shapefile (and each polygon in that file must have its id as a WSNO value in the streams shapefile). The *LINKNO* - *DSLINKNO* values define the drainage pattern, so the LINKNO values must be unique and non-negative, and the DSLINKNO values all -1 (indicating a watershed outlet) or another LINKNO value. Drainage must not be circular.

If *DSNODEID* values are included they are used as references to the inlets/outlets shapefile and should exist as *ID* values in that file.

If *Length* and *Drop* values are provided and *Reuse* is checked then these will be used as the length in metres and the drop in metres of their stream reach. Otherwise these values will be calculated.

An *outlets/inlets shapefile*, if provided, must be a point shapefile with the following fields:

Necessary/Optional	**Name**	**Type**
Necessary	ID	Integer
Necessary	RES	Integer
Necessary	INLET	Integer
Necessary	PTSOURCE	Integer

If drainage definition is by *Drainage table* then a comma-separated value (*.csv*) file must be identified. Its first line is assumed to be column headings and is ignored. The following lines must consist of pairs of integers, the first being a PolygonId value from the grid shapefile, and the second either another PolygonId value or -1. All polygons in the grid shapefile must be represented. The drainage file is just another way of presenting the PolygonId - DownId relationship that may be contained in the grid shapefile, and is provided as an alternative for users who wish to define a particular drainage pattern on a grid model. A suitable initial drainage table is created when a grid model is created by delineation, and may be found as *<proj>drainage.csv* in the *Watershed/Shapes* directory, where *<proj>* is the project name.

17. Appendix IV: Installing and Using MPI

QSWAT+ uses TauDEM, Terrain Analysis Using Digital Elevation Maps, as the main component for doing watershed delineation.

The current version of TauDEM is designed for modern multicore architectures. A *multicore* CPU has more than one processor: 2, 4, even more. By using an implementation of the *Message Passing Interface* (MPI), TauDEM can take advantage of multiple cores, splitting tasks into several processes distributed between them, and speeding up calculations. It is also able (provided your machine has enough memory) to process very large DEMs: up to 4GB.

The implementation of MPI used by TauDEM is MS-MPI from Microsoft, which is free to download and use.

By default MS-MPI is not installed when you install QSWAT+: TauDEM will run without it, in "single process" mode.

You may obtain and install MS-MPI, and we give some instructions below. Then QSWAT+ will take advantage of it. There is no point in doing this unless you have a multicore CPU, and probably not much unless you have large DEMs: see Appendix II for some examples of processing times on large DEMs.

It is possible the first time you run one of the TauDEM executables after installing MS-MPI that you get a TauDEM error, and in the details you will see

Fatal protocol error: check version between Mpiexec.exe, Msmpi.dll, and Smpd.exe.

The way to fix this is to remove or make inaccessible the copy of *msmpi.dll* that is stored with the TauDEM executables in order to enable them to run without MS-MPI. Go to the directory *C:/SWAT/SWATEditor/Taudem5Bin* and rename *msmpi.dll* to, say, *msmpidll* (which is better than removal as you can restore it later if necessary).

17.1 Installing MS-MPI

Note that you need to be an administrator to install MS-MPI.

- Download the MS-MPI installer *MSMpiSetup.exe* from http://msdn.microsoft.com/en-us/library/bb524831%28v=vs.85%29.aspx and install it.

- Restart the machine (otherwise it will not be able to find the executable *mpiexec.exe*).

- On a 32-bit machine there can be a problem installing and running MS-MPI. If *MSMPISetup* will not run you need the 2008 version: see below. If the installer runs, after restarting your machine you should check all is OK by opening a command

window and running the command *mpiexec -help*. If there is an error saying that mpiexec is not a valid 32-bit command then download and install the 2008 version of MS-MPI from http://www.microsoft.com/en-gb/download/details.aspx?id=14737.

17.2 Configuring QSWAT+ to use MPI

On the QSWAT+ main screen there is a *QSWAT parameters* button. Clicking this enables users to set the locations of the SWAT Editor (by default *C:/SWAT/SWATEditor*) and of *mpiexec.exe* (by default *C:/Program Files/Microsoft MPI/Bin*). Users should also make sure *Use MPI* is checked: this is checked by default if the MPI *Bin* directory exists. Then MPI will be used during watershed delineation provided the *Number of processes* spinbox in the *Delineate watershed* form is set to some positive number. This should be left at zero if MPI is not installed. See Appendix II for some example times for large DEMs on different machines with typical settings of the number of processes.

17.3 Blocking network access

MS-MPI supports multi-process computations across several machines. So you may get a Windows firewall query as to whether you want to allow the MS-MPI program manager, *smpd*, access to networks. Since TauDEM only runs multiple processes inside your machine you may *Cancel* access (unless, of course, you are using MS-MPI for other purposes for which it needs network access).

Separate TauDEM executables (*D8FlowDir*, *DInfFlowDir*, etc) may also raise the same query, which is annoying - but it only happens the first time you use them!

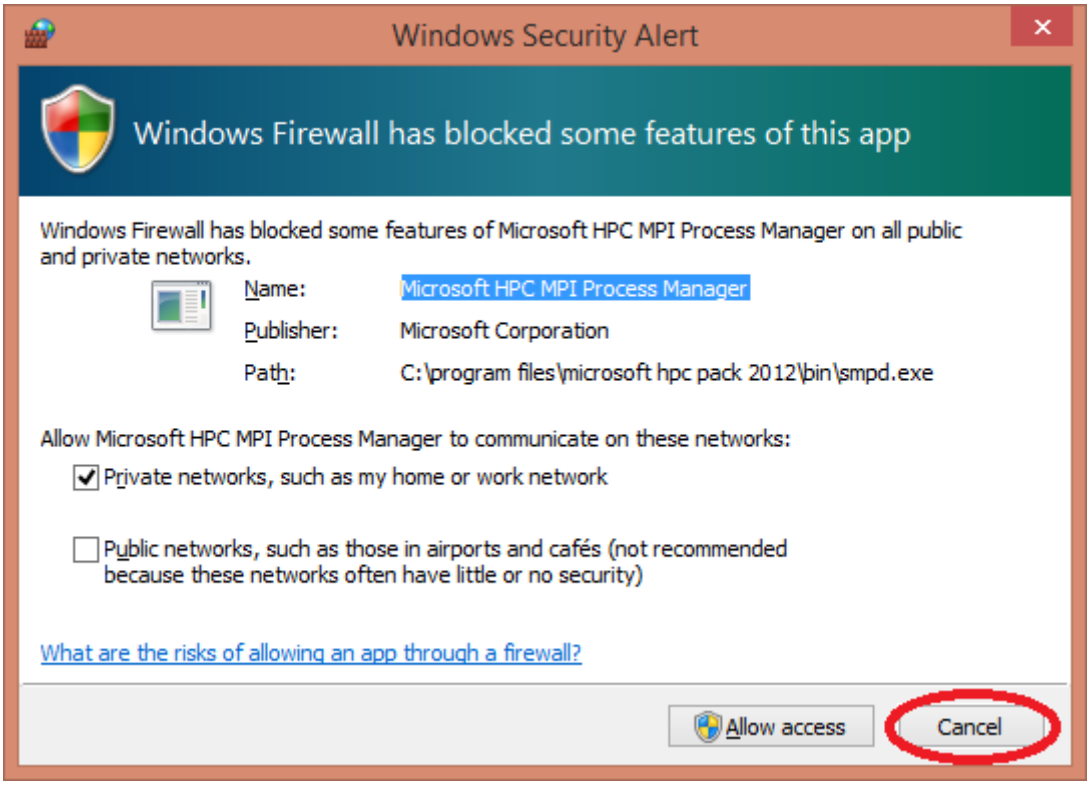


Figure 74: Cancelling network access for MPI

18. Appendix V: Conversion from ArcSWAT to QSWAT+

18.1 Purpose

The purpose of this tool is to help users migrate from using ArcSWAT to using QSWAT+. It is only concerned with reusing data developed for ArcSWAT projects. As well as reading this guide, users will also need to acquaint themselves with QGIS and QSWAT+ documentation. The essential steps in QSWAT+ are much the same as ArcSWAT. Creating the model consists of first delineating the watershed, then combining the delineated watershed with landuse and soil data, and defining HRUs. At this point the work with GIS is complete and an editor is used to add weather data, edit inputs as required, and then run SWAT+.

The conversion tool has three particular purposes in mind, for which it offers **Full** , **Existing,** and **No GIS** options.

Full conversion is for users who want to rebuild their ArcSWAT project from scratch, and take full advantage of the new features of QSWAT+. After conversion, they will be at the start of their SWAT+ project, about to do watershed delineation. The data they need that could be collected from the ArcSWAT project: the maps, soil data, weather data, etc. will be available, and they will be able to create a model as close to or as different from their ArcSWAT project as they wish.

Existing conversion is an option for those who aim to keep their model as close as possible to the ArcSWAT one. An immediate problem with shifting from Arc GIS to QGIS is that the watershed delineation software is different, and can give different

results. Watershed boundaries may shift, streams may move, the numbers and boundaries of subbasins may change. To avoid this, the existing option uses the subbasins and streams generated from Arc GIS as an existing watershed, and so preserves the watershed boundary, the streams and the subbasins. No delineation is done in QGIS, so multiple channels within subbasins cannot be created: all HRUs will drain into the main channel, as in ArcSWAT. Adding landuse and soil maps is done within QSWAT+, as is HRU calculation, so the details of HRUs may differ from the ArcSWAT model. A major advantage of the existing option (shared with the full option) is that since there are subbasin, stream and HRU maps inherited or created that are consistent with the model, they may be used for geographic visualisation of results.

No GIS conversion is an option for users who just want to run SWAT+ on their existing model without any preparation like delineation or defining HRUs. Nothing is done in GIS, so the subbasins , streams and HRUs are not changed. But no geographic visualisation of results will be possible.

The conversion tool copies much of the ArcSWAT project, and then changes the contents of some copied files, but it does not affect any of the files in the ArcSWAT project. Nor does it rely on them afterwards: QSWAT+ always maintains the files it uses within its own project structure. So your ArcSWAT project is safe, and you can change it or delete it later without affecting the QSWAT+ project.

18.2 Instructions

Run *C:/SWAT/SWATPlus/Tools/ConvertFromArc/runConvertFromArc.bat* by double-clicking on it or using it as a command in a command or shell window. If you didn't use a command or shell window one will open, and you will see messages there as the conversion progresses. You will be asked to locate the existing ArcSWAT project directory. You can recognise you are in the right place as this directory will contain within it at least the directories called *Watershed, Scenarios*, and *RasterStore.idb*. Then you are asked to identify a parent directory for your QSWAT+ project. This should be an existing directory (though you can create one on the fly if you wish). Note that you are not selecting the QSWAT+ project directory, since this is about to be created, but only where that project directory is to be placed. Do not try to arrange the ArcSWAT and parent directories so that one is a descendant of the other, as this would cause problems (and the tool will prevent you in any case).

Once you have selected the parent directory you will be invited to reuse the ArcSWAT project name or create a new one. Once the name is chosen, the tool checks to see if the QSWAT+ project directory exists already. If it does, the tool checks that you wish to delete it, and does so if you confirm that it should.

Deletion of an existing directory to make the QSWAT+ project directory sometimes seems to fail, with a message being displayed to say so. Sometimes this message appears although the deletion has in fact taken place. If this happens you can do the following:

- Check with Windows Explorer if the directory exists and if so use Windows Explorer to delete it.
- If there is still a problem, check that there is nothing else (like another Windows Explorer window) accessing the directory or one below it.
- Open the Task Manager (right click in the taskbar to find it), switch to the Details tab, and kill any processes with Name *python.exe*. Python processes seem often not to die when they are exited.
- If all else fails and you still cannot delete a directory, reboot the machine.

When the QSWAT+ project directory has been created, as a directory with the project name within the parent directory, the conversion tool creates all the subdirectories needed, and sets up a new project database, and reference database.

Then the conversion tool opens a form for you to make the choice between **Full, Existing**, or **No GIS** conversion.

If you choose **Full** or **Existing**, a new project file is created (a .qgs file with the project name in the project directory) and the DEM, landuse and soil rasters and shapefile maps from the ArcSWAT project are copied in, rasters into *Watershed/Rasters* and shapefiles into *Watershed/Shapes*. In this process the DEM is converted to a GeoTiff, as this is required by QSWAT+ delineation.

QSWAT+ normally uses two shapefiles to mark points. The first is the inlets/outlets shapefile and includes the main outlet(s) and any inlets that were defined in the ArcSWAT model. Note the Monitoring Points shapefile generated by ArcSWAT includes points at stream junctions as well as main outlets and any extra points defined by the user. QSWAT+ does not include in its inlets/outlets shapefile the points which are stream junctions: they are implicit in the model. This means that if you have marked any additional points as monitoring points (and which have created subbasin boundaries) and are using the **Full** option you will need to add them by hand: QSWAT+ includes a facility for adding more outlets manually.

If you are using the **Full** option, delineation will use the inlets/outlets shapefile. It is possible that some points in that file will not snap to the stream network that QSWAT+ calculates. If this happens, you can fix it either by editing the inlets/outlets shapefile using the shapefile editing capabilities of QGIS, and dragging the affected points to their streams, or it may be sufficient to increase the snap threshold from its default value of 300m.

If you have reservoirs or point sources they will be added to the inlets/outlets shapefile. QSWAT+ can also define reservoirs itself if it finds areas of water above a user-definable threshold percentage of a landscape unit, so you may find later that your reservoirs in the inlets/outlets shapefile become unnecessary. If you choose this option in QSWAT+ and it creates a reservoir from one or more channels where you marked a reservoir, then it may be a good idea to return to delineation, remove the reservoir from the inlets/outlets file and rerun delineation. The marking of a reservoir and then choosing to create reservoirs on channels with a threshold amount of water in their drainage basins is a way of stopping adjacent

channels being merged to form single reservoirs, so unless this is what you want the marked reservoir should be removed.

The numbering of reservoirs and point sources is likely to be different from the ArcSWAT numbers. You can check what numbers have been given to a point source or reservoir by selecting the item in the inlets/outlets shapefile in QGIS and using the *Identify Features* button to find the *ID* value.

If in ArcSWAT you marked a subbasin outlet and later added a reservoir to that subbasin, the reservoir is placed at the same position as the outlet. This causes problems with QSWAT+ watershed delineation (when using the **Full** option) because it will only use one of the two points. To prevent this, move such reservoirs a little upstream before running delineation. To do this in QGIS 3, first select the inlets/outlets shapefile in the *Layers* panel, uncheck the box in the inlets/outlets legend beside Outlet, so that outlets are not visible and you won't select one by accident. Then click the yellow pencil *Toggle editing* button to start editing. Click the *Vertex tool* button (third right from *Toggle editing)*. When you then slide the mouse over a reservoir in the map canvas, it turns red. Left click on it and slide the mouse a little upstream. A small red cross slides with the mouse, showing that you have grabbed the reservoir. Left click again to place the reservoir in the new location (on a channel, of course). Finally click the *Toggle edit* button again, and confirm the changes are to be saved. Make the outlets visible again so you can see they have not been moved.

If you have added point sources to all subbasins in ArcSWAT and are using the **Full** option, you might like to remove them before doing delineation. QSWAT+ will add point sources to all channels, so you are very unlikely to need the ones you added, and they will cause unnecessary divisions between landscape units. To delete a point source, select the inlets/outlets shapefile in the *Layers* panel, click the yellow pencil *Toggle editing* button to start editing, hold Ctrl and click on each of the point sources in the map canvas in turn. Each will turn yellow. If you make a mistake release Ctrl and click away from any point to cancel the selection. When you have selected all you want to remove, click *Delete Selected* (red dustbin), then click the *Toggle edit* button again, and confirm the changes are to be saved.

If you use the **Existing** option the inlets/outlets shapefile is not used. The subbasins shapefile from ArcSWAT is used as both the *Subbasins shapefile* and the *Watershed shapefile*, and the stream network is used as the *Channels shapefile*. QSWAT+ allows one stream but multiple channels within subbasins, and the *Watershed shapefile* defines the areas draining into channels. Since ArcSWAT did not support such a feature, channels are set the same as streams. Since we also use the subbasin numbering from ARCSWAT, reservoirs and point sources should be correctly placed: no correction as with the **Full** option is necessary. The conversion tool adds the attributes required by QSWAT+ to the various shapefiles, so running the delineation form should involve no more than clicking *Run* and then *OK*.

18.3 Landuse and soil data

For both **Full** and **Existing** options, the landuse and soil maps from ArcSWAT are copied.

We also need the relevant lookup tables (unless the soil map is SSURGO). Users are invited to choose to use their own comma separated value(CSV) file for each of landuse and soil if they have one (they can supply it when they run QSWAT+). Otherwise the following procedure is used to create lookup tables:

- The landuse and soil maps are read, and percentages of each value occurring in them are calculated.
- The percentages of the watershed for each landuse and soil are read from the ArcSWAT project's *LanduseSoilsReport.txt*
- The two lists are matched by being placed in descending order of percentage, which should give the relation between map value and landuse or soil name that comprises the lookup table.
- If the lengths of the lists differ the percentages from the map are reported in *landuse_map_percentages.txt* or *soil_map_percentages.txt* which is placed in the QSWAT+ project directory for manual comparison with the ArcSWAT *LanduseSoilsReport.txt*, and no CSV file is generated.
- If consecutive percentages in a list differ by less than one percent, or if percentages in corresponding lists differ by more than one percent, the user gets a warning message. If there are percentages which are themselves less than one percent, the user is also warned.
- The CSV files are saved as *landuse_lookup.csv* and *soil_lookup.csv* in the project directory.
- When, later, QSWAT+ is run there is an option *Use csv* for the landuse and soil lookup tables which allows these files to be read (and stored as database tables for future runs of QSWAT+). In the meantime users can inspect these files and correct them if necessary. This will certainly be necessary if in the ArcSWAT project more than one map value was mapped to the same soil or landuse.

The landuse and soil maps saved in the ArcSWAT project have been clipped to the ArcSWAT watershed boundary. This is convenient for creating lookup tables, since the percentages calculated from maps are correct for the watershed, but a disadvantage when using the **Full** option as the QSWAT+ watershed may not agree precisely with the ArcSWAT one. QSWAT+ will report if the percentage of the watershed covered by the landuse or soil map falls below 95%. If this happens users should, if possible, replace the landuse and soil maps with unclipped maps. QSWAT+ does not initially create HRUs where soil or landuse is not defined, but it will later enlarge all HRUs proportionately so that their total area is the same as the watershed. So falling below this 95% threshold may not be too much of a problem if the unclipped maps are not available.

18.4 Other data

For all three options (**Full, Existing,** and **No GIS**) weather generator, observed weather, and some reference data is recovered from the ArcSWAT project.

18.4.1 Weather generator data

The ArcSWAT *Default*/*TxtInOut* directory is searched for *.wgn* files, each of which contains data for a weather generator station. Each different station has its data added to the *weather_wgn_cli* and *weather_wgn_cli_mon* tables in the QSWAT+ project database.

18.4.2 Observed weather gauges

The ArcSWAT *Default*/*TxtInOut* directory is searched for *pcp1.pcp, tmp1.tmp, hmd.hmd, slr.slr* and *wnd.wnd* files. If any are found, it (and for precipitation and temperature its successor files if any) are used to create the data for weather gauges of the appropriate category. Data about the gauge is placed in *weather_files* and *weather_sta_cli* tables in the QSWAT+ project database, and the observed data itself placed in appropriate files in the QSWAT+ project's *TxtInOut* directory.
The conversion tool also attempts to read the tables *SubHmd, SubSlr* and *SubWnd* tables in order to discover the locations of the gauge stations for relative humidity, solar radiation and wind speed. If the ArcSWAT project did not have any observed data in any of these categories, the corresponding table will not exist, and failure to read the table is reported during conversion. These messages may be ignored.

18.4.3 Reference data

Data in the tables *crop, fert, pest, septwq, till,* and *urban* tables in the ArcSWAT project's reference database may have been edited by the user. To make such changes available in the QSWAT+ project, data from these tables is extracted and placed in tables *arc_plant, arc_fertilizer, arc_pesticide, arc_septic, arc_tillage* and *arc_urban* in the QSWAT+ project database.

18.4.4 Inlet and point source data

In each ArcSWAT scenario's *TxtInOut* directory, the *fig.fig* file is read to find *recday, recmon, recyear* or *reccnst* commands. The corresponding data files are converted into the *csv* format used by SWAT+. Daily, monthly and yearly files are converted into files with the same name, but with extension *.csv*, placed in the corresponding SWAT+ project's scenario's *TxtInOut* directory. Average annual data files (from *reccnst* commands) are amalgamated into a single file called *rec_const.csv* in the SWAT+ project's scenario's *TxtInOut* directory. Each data file contributes one record to this file, and the name of this record (its first field) is generated as follows. If the data file's name was *Np.dat*, where *N* consists of one or more digits, then the name is *ptN*. Similarly *Ni.dat* gives a name *inN*. Otherwise the new name is the old one prefixed by *x*, to make sure it starts with a letter.

18.4.5 Dates

Start and end dates, plus the *NYSKIP* value for the simulation are read from the ArcSWAT project's *file.cio* and placed in the QSWAT+ project's *time.sim* and *print.prt* files respectively.

19. Appendix VI: Conversion from QSWAT to QSWAT+

19.1 Purpose

The purpose of this tool is to help users migrate from using QSWAT to using QSWAT+. It is only concerned with reusing data developed for QSWAT projects. As well as reading this guide, users will also need to acquaint themselves with QSWAT+ documentation.

The essential steps in QSWAT+ are much the same as QSWAT. Creating the model consists of first delineating the watershed, then combining the delineated watershed with landuse and soil data, and defining HRUs. At this point the work with GIS is complete and an editor is used to add weather data, edit inputs as required, and then run SWAT+.

The conversion tool has two particular purposes in mind, for which it offers **Full** and **No GIS** options.

> **Full** conversion is for users who want to rebuild their QSWAT project from scratch, and take full advantage of the new features of QSWAT+. After conversion, they will be at the start of their SWAT+ project, about to do watershed delineation. The data they need that could be collected from the QSWAT project: the maps, settings like thresholds that are common to QSWAT and QSWAT+, soil data, weather data, etc. will be available, and they will be able to create a model as close to or as different from their QSWAT project as they wish.
>
> **No GIS** conversion is an option for users who just want to run SWAT+ on their existing model without any preparation like delineation or defining HRUs. Nothing is done in GIS, so the subbasins, streams and HRUs are not changed. But no geographic visualisation of results will be possible.

19.2 Instructions

19.2.1 Prerequisites

We assume you already have QSWAT: the conversion tool is in fact supplied with QSWAT version 1.8.. You will need to install QSWAT+, and a suitable version of QGIS, probably QGIS 3.4 as instructed in the QSWAT+ user manual. You will also need to install the QSWAT+ Editor from the SWAT website.

19.2.2 Conversion

Run *C:/SWAT/SWATEditor/runConvertToPlus.bat* by double-clicking on it or using it as a command in a command or shell window. If you didn't use a command or shell window one will open, and you will see messages there as the conversion progresses.
You will be asked to locate the existing QSWAT project file (.qgs). Then you are asked to identify a parent directory for your QSWAT+ project. This should be an existing directory

(though you can create one on the fly if you wish). Note that you are not selecting the QSWAT+ project directory, since this is about to be created, but only where that project directory is to be placed. Do not try to arrange the QSWAT and parent directories so that one is a descendant of the other, as this would cause problems (and the tool will prevent you in any case).

Once you have selected the parent directory you will be invited to reuse the QSWAT project name or create a new one. Once the name is chosen, the tool checks to see if the QSWAT+ project directory exists already. If it does, the tool checks that you wish to delete it, and does so if you confirm that it should.

Deletion of an existing directory to make the QSWAT+ project directory sometimes seems to fail, with a message being displayed to say so. Sometimes this message appears although the deletion has in fact taken place. If this happens you can do the following:

- Check with Windows Explorer if the directory exists and if so use Windows Explorer to delete it.
- If there is still a problem, check that there is nothing else (like another Windows Explorer window) accessing the directory or one below it.
- Open the Task Manager (right click in the taskbar to find it), switch to the Details tab, and kill any processes with Name *python.exe*. Python processes seem often not to die when they are exited.
- If all else fails and you still cannot delete a directory, reboot the machine.

When the QSWAT+ project directory has been created, as a directory with the project name within the parent directory, the conversion tool creates all the subdirectories needed, and sets up a new project database, and reference database.

Then the conversion tool opens a form for you to make the choice between **Full** or **No GIS** conversion.

If you choose **Full** , a new project file is created (a .qgs file with the project name in the project directory) and the DEM, landuse and soil rasters and shapefile maps from the QSWAT project are copied in, rasters into *Watershed/Rasters* and shapefiles into *Watershed/Shapes*. All scenarios are copied, as in the **No GIS** option: see below.

If you choose **No GIS**. an empty project file is created and the QSWAT+ directory structure is created. The DEM, landuse and soil rasters are not copied, and neither are most of the shapefile maps created in the QSWAT project. But the *Scenarios* directory is copied in its entirety, which means in particular that the *Results* directories in the default and any saved scenarios are copied. The *Default* scenario will be overwritten when you run SWAT+, but any saved scenarios (plus the default one if you save a copy before running SWAT+) will be available for visualisation and comparison with the result of running SWAT+.

19.3 Data

For both options (**Full** and **No GIS**) landuse, soil, weather generator, observed weather, and some reference data is recovered from the QSWAT project.

19.3.1 Landuse data

The data from the *crop* and *urban* tables in the QSWAT project reference database is copied into tables *plant* and *úrban* in the QSWAT+ project database.

19.3.2 Soil data

If the QSWAT project did not use STATSGO or SSURGO soils, the data from table *usersoil* in the QSWAT project reference database is copied into a table *usersoil* in the QSWAT+ project database.

19.3.3 Weather generator data

The QSWAT *Default/TxtInOut* directory is searched for *.wgn* files, each of which contains data for a weather generator station. Each different station has its data added to the *weather_wgn_cli* and *weather_wgn_cli_mon* tables in the QSWAT+ project database.

19.3.4 Observed weather gauges

The QSWAT *Default*/*TxtInOut* directory is searched for *pcp1.pcp, tmp1.tmp, hmd.hmd, slr.slr* and *wnd.wnd* files. If any are found, it (and for precipitation and temperature its successor files if any) are used to create the data for weather gauges of the appropriate category. Data about the gauge is placed in *weather_files* and *weather_sta_cli* tables in the QSWAT+ project database, and the observed data itself placed in appropriate files in the QSWAT+ project's *TxtInOut* directory.

19.3.5 Reference data

Data in the tables *fert, pest, septwq,* and *till* tables in the QSWAT project's reference database may have been edited by the user. To make such changes available in the QSWAT+ project, data from these tables is extracted and placed in tables of the same name in the QSWAT+ project database.

19.3.6 Inlet and point source data

In each QSWAT scenario's *TxtInOut* directory, the *fig.fig* file is read to find *recday, recmon, recyear* or *reccnst* commands. The corresponding data files are converted into the *csv* format used by SWAT+. Daily, monthly and yearly files are converted into files with the same name, but with extension *.csv*, placed in the corresponding SWAT+ project's scenario's *TxtInOut* directory. Average annual data files (from *reccnst* commands) are amalgamated into a single file called *rec_const.csv* in the SWAT+ project's scenario's *TxtInOut* directory. Each data file contributes one record to this file, and the name of this record (its first field) is generated as follows. If the data file's name was *Np.dat*, where *N* consists of one or more digits, then the name is *ptN*. Similarly *Ni.dat* gives a name *inN*. Otherwise the new name is the old one

prefixed by *x*, to make sure it starts with a letter.

19.3.7 Dates

Start and end dates, plus the *NYSKIP* value for the simulation are read from the QSWAT project's *file.cio* and placed in the QSWAT+ project's *time.sim* and *print.prt* files respectively.