QGIS Interface for SWAT (QSWAT)

Version 1.5
January 2018

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

This guide describes QSWAT, which uses the 2012 version of the SWAT model.
This version describes QSWAT version 1.5, which needs SWAT Editor version 2012.10.19.

QSWAT (QGIS SWAT)

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

Contents

1. Environment and Tools Required 4
2. Installation 4
3. Structure and Location of Source Data 6
4. Structure and Location of Output Data 6
5. Setup for Robit Watershed, Lake Tana Basin 7
   - 5.1. Watershed Delineation 9
   - 5.2. Create HRUs 19
   - 5.3. SWAT Setup and Run 31
   - 5.4. Output Visualisation with QSWAT 36
     - 5.4.1. Preparing to visualise 37
     - 5.4.2. Running visualise 37
     - 5.4.3. Visualisation period 38
     - 5.4.4. Static visualisation 38
     - 5.4.5 Printing 41
     - 5.4.6 Animation 48
     - 5.4.7. Plotting 55
   - 5.5. SWAT Output Visualization with SWAT Error Checker 61
6 Appendix I 66
   - 6.1 Adding tables into SWAT reference database 66
   - 6.2 Preparing daily climatic data for QSWAT 70
   - 6.3 Preparing landuse and soil look-up tables 72
7 Appendix II – preparing global DEM data for QSWAT 73
   - 7.1 Introduction 73
   - 7.2 Obtaining SRTM DEM data 73
     - 7.2.1. 90m data 73

7.2.2. 30m data 73
7.3. Clipping the grids 74
7.4. Merging the grids 75
7.5. Reprojecting the grid 76
7.6. Masking a DEM 80
7.7. Large DEMs 83
8 Appendix III: Using a Predefined Watershed and Stream Network 86
9 Appendix IV: Installing and Using MPI 89
9.1 Installing MS-MPI 89
9.2 Configuring QSWAT to use MPI 90
9.3 Blocking network access 90

1. Environment and Tools Required

- Microsoft Windows (any version, as far as we are aware)
- Microsoft Access, as the interface uses an Access database
- Text editor (e.g. WordPad, NotePad, NotePad++) that enables you to read and edit ASCII text files.
- A tool like WinZip that can uncompress.zip files (not needed with recent versions of Windows)

System requirements for SWAT Editor Version 2012.10.19

- Microsoft Windows XP to 10
- Microsoft .Net Framework 3.5
- Adobe Acrobat Reader Version 7 or higher. Acrobat Reader may be downloaded for free from: http://www.adobe.com/products/acrobat/readstep2.html

2. Installation

- Install QGIS by running **QGIS-OSGeo4W-2.6.1-1-Setup-x86.exe**. It can be downloaded from http://qgis.org/downloads/QGIS-OSGeo4W-2.6.1-1-Setup-x86.exe. Use the default folder *C:\Program Files\QGIS Brighton* as the installation folder, or C*:\Program Files (x86)\QGIS Brighton* on a 64-bit machine. Currently you must use the 32-bit version on 64-bit machines. We recommend version 2.6 rather than a later version. From now on we will use *Program Files* as a folder name, even though it will be *Program Files (x86)* on a 64-bit machine.
- Install SWATEditor 2012 version in its standard place *C:\SWAT\SWATEditor*. For the SWAT 2012 version, unzip the installation archive swateditor_install_201210_219.zip (or a later version if available from http://swat.tamu.edu/software/arcswat/swateditor/) and double-click on the "**setup.exe**" program to install the SWAT Editor. This will launch the SWAT Editor install program. Use the default folder *C:\SWAT\SWATEditor* as the installation folder.

The SWATEditor folder will consist of Databases folder, SwatCheck folder, SWATEditorHelp, and other SWAT executables. The database folder consists of SWAT2012 databases which require updating based on user available information (e.g. user soil, weather generator). Refer to **Appendix I** (page 66) on how to update a database. Refer to the **SWATEditor_Documentation.pdf** document in SWATEditorHelp for important information on how to get started with the SWAT Editor.

Make sure you have the system requirements listed above for the SWAT Editor to work properly. Failure to have the above system configuration may result in failure to install the SWAT Editor or errors in the SWAT Editor interface.

Users can join the user group for help at https://groups.google.com/forum/#!forum/qswat

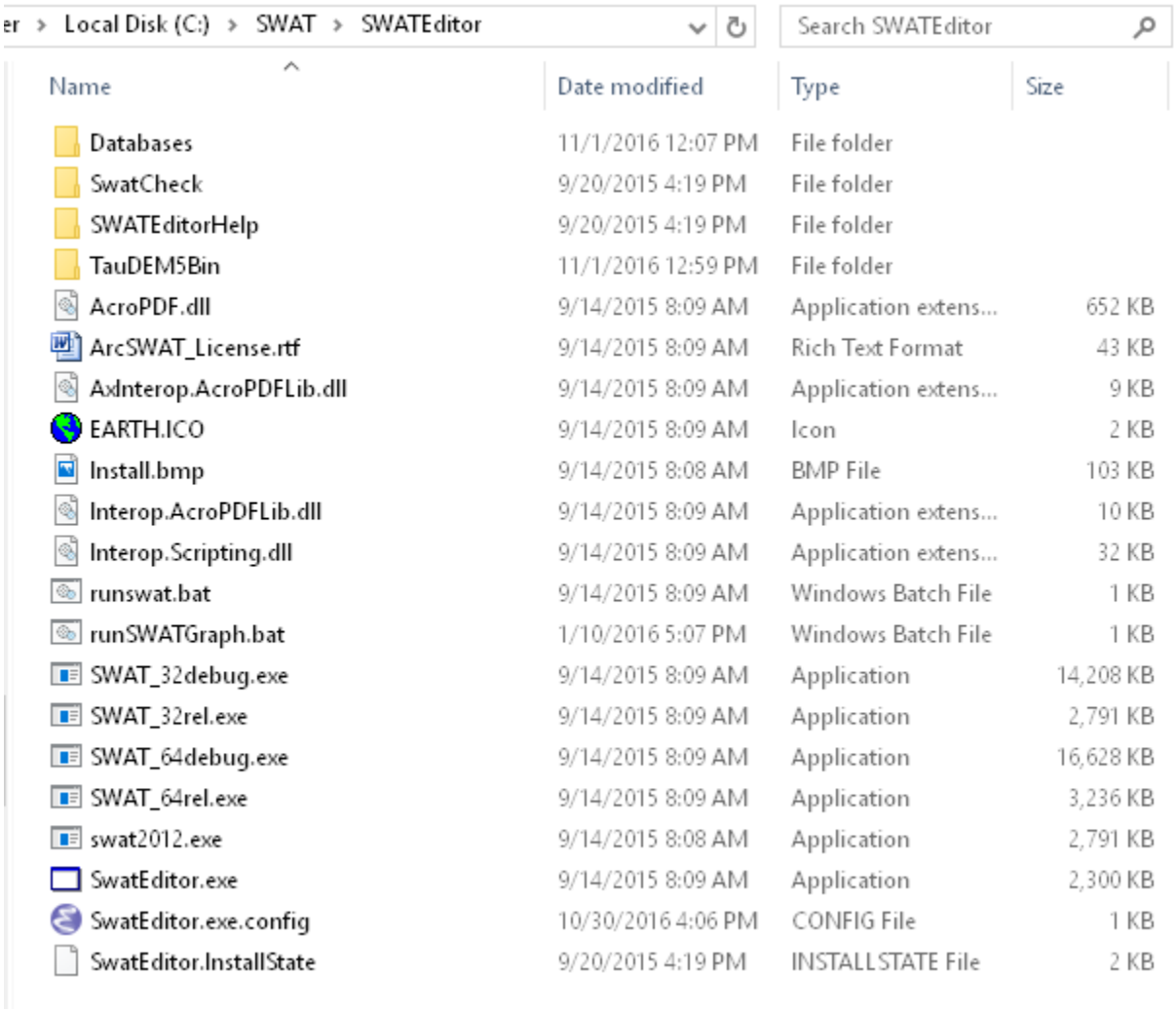


Figure 1: SWAT Editor installation folder (after installing QSWAT)

- Install QSWAT by running **QSWATinstall1.5**.exe or a later version. This adds some files to the SWATEditor: a project database and a reference database in *Databases*, and also stores the TauDEM executables in a new folder *C:\SWAT\SWATEditor\TauDEM5Bin*. The QGIS plugin QSWAT is put into the user's home folder in *.qgis2\python\plugins\QSWAT*,

which we will refer to as the *QSWAT folder*. The latest version of QSWAT can be downloaded from the SWAT website http://swat.tamu.edu/.

3. 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** (page 66) 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** (page 73) for preparing global data for QSWAT. Users can also refer to the **Arcswat Interface for SWAT2012 User's Guide** (pages 11-36) for detailed guidance. The example files used in this manual are available in the QSWAT folder in a folder *ExampleDataset*. This folder contains:

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

It is a good idea while following this guide to add the *ExampleDataset* folder to your Favorites or Quick Access list, as we will be regularly accessing files within it.

4. Structure and Location of Output Data

We will be establishing a project called *Demo*, say, in a folder *C:\ QSWAT_Projects\Robit\Demo.*. This will create:

1) In the folder *C:\QSWAT_Projects\Robit* the project file *Demo.qgs* and a folder *Demo*. If we later want to reopen the project *Demo.qgs* will be the file we look for. The folder *Demo* is the *project folder*.
2) A folder *C:\QSWAT_Projects\Robit\Demo\Scenarios\Default\TxtInOut* that will contain all the SWAT input and output files.
3) A folder *C:\QSWAT_Projects\Robit\Demo\Scenarios\Default\TablesIn* that may be used by the SWAT Editor.
4) A folder *C:\QSWAT_Projects\Robit\Demo\Scenarios\Default\TablesOut* that may be used by the SWAT Editor and is used to hold results files from visualisation.

5) A folder *C:\QSWAT_Projects\Robit\Demo\Source* that will contain copies of our input maps and a number of intermediate maps generated during watershed delineation.
6) If we choose to save a SWAT run as *Sim1*, say, then a folder *C:\QSWAT_Projects\Robit\Demo\Scenarios\Sim1* will be created (a copy of *C:\QSWAT_Projects\Robit\Demo\Scenarios\Default*, including its sub-folders).
7) A folder *C:\QSWAT_Projects\Robit\Demo\Watershed* with several sub-folders

5. Setup for Robit Watershed, Lake Tana Basin

1. Start **QGIS** and select **Plugins** menu → **Manage and Install Plugins** and find **QSWAT**. Click its checkbox to install, which takes a few seconds. The SWAT icon will appear in the toolbar. If it fails to appear check you are running a 32 bit version of QGIS.

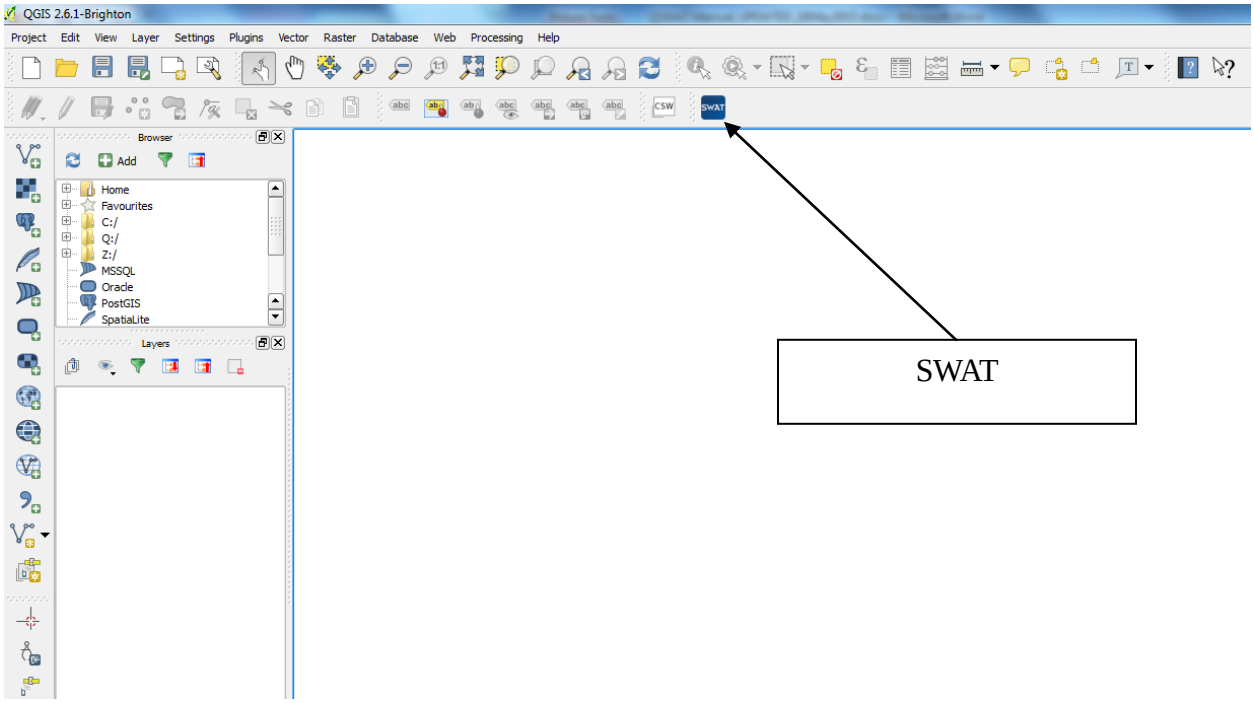


Figure 2; QGIS with the SWAT icon

2. Start **QSWAT** by clicking the **SWAT** icon.
3. The main QSWAT interface will be displayed. Click the box **New Project**.
4. A browser will be displayed requesting a name for the new project. Type **Demo** in the text box labelled *File name* (under the *C:\QSWAT_Projects\Robit* folder). See Figure 3.

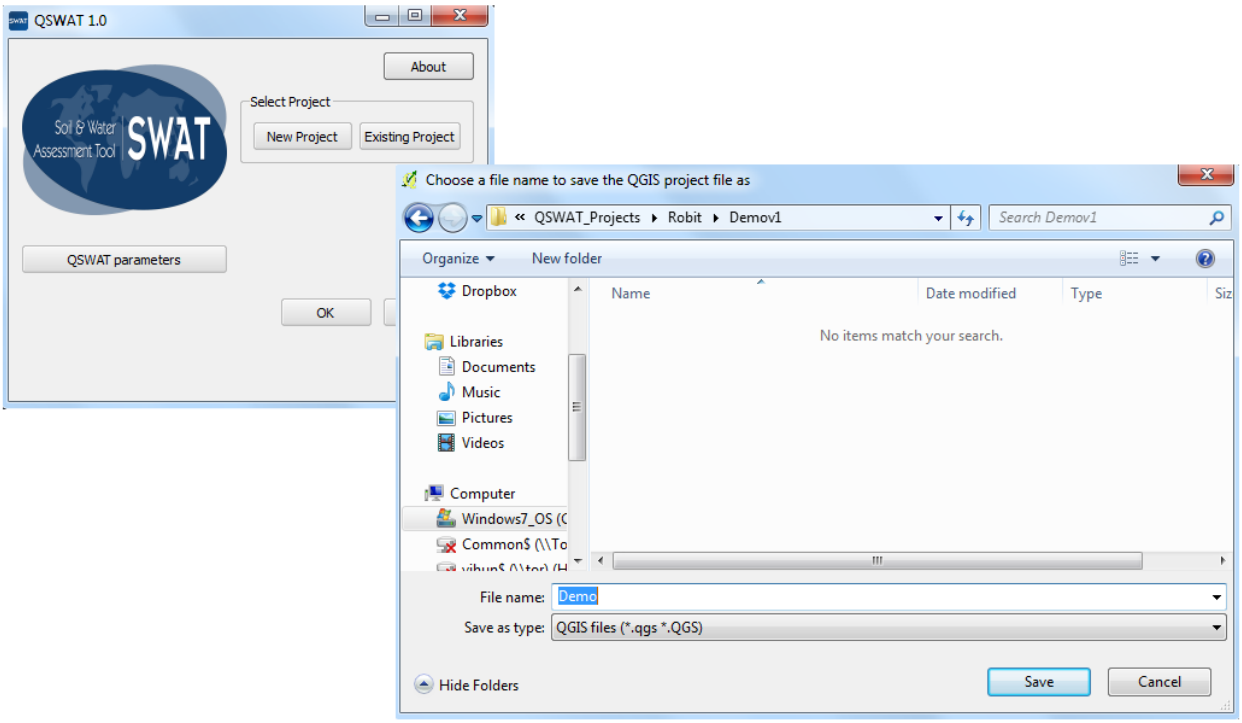


Figure 3: Creating a QSWAT project

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.

The interface now presents a step-by-step configuration to be followed in order to prepare the SWAT simulation, starting with Step 1 (Figure 4).

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

Figure 4: Step 1 interface

5. At this stage the project database is created as *Demo.mdb* in the project folder, and a copy of the SWAT reference database *QSWATRef2012.mdb* is also created there. For this example you need to update the reference database with details of the soil data we are using and also possible weather generation stations. Refer to **Appendix I** (page 66) on how to do this..

5.1. Watershed Delineation

6. To start automatic watershed delineation click the **Delineate Watershed** button. When the prompt box is opened browse to the data source in the dialogue box next to **Select DEM**.

7. Browse to the *ExampleDataset\DEM\srtm_30m* folder and open the file *hdr.adf*. This digital elevation map (DEM) is an ESRI grid and consists of a number of files. The file called *hdr.adf* is the one to choose for such a grid. DEMs are raster files (also called grids) and come in a large variety of formats. A common format is GeoTiff, when you would be looking for a file with siffix *.tif*.

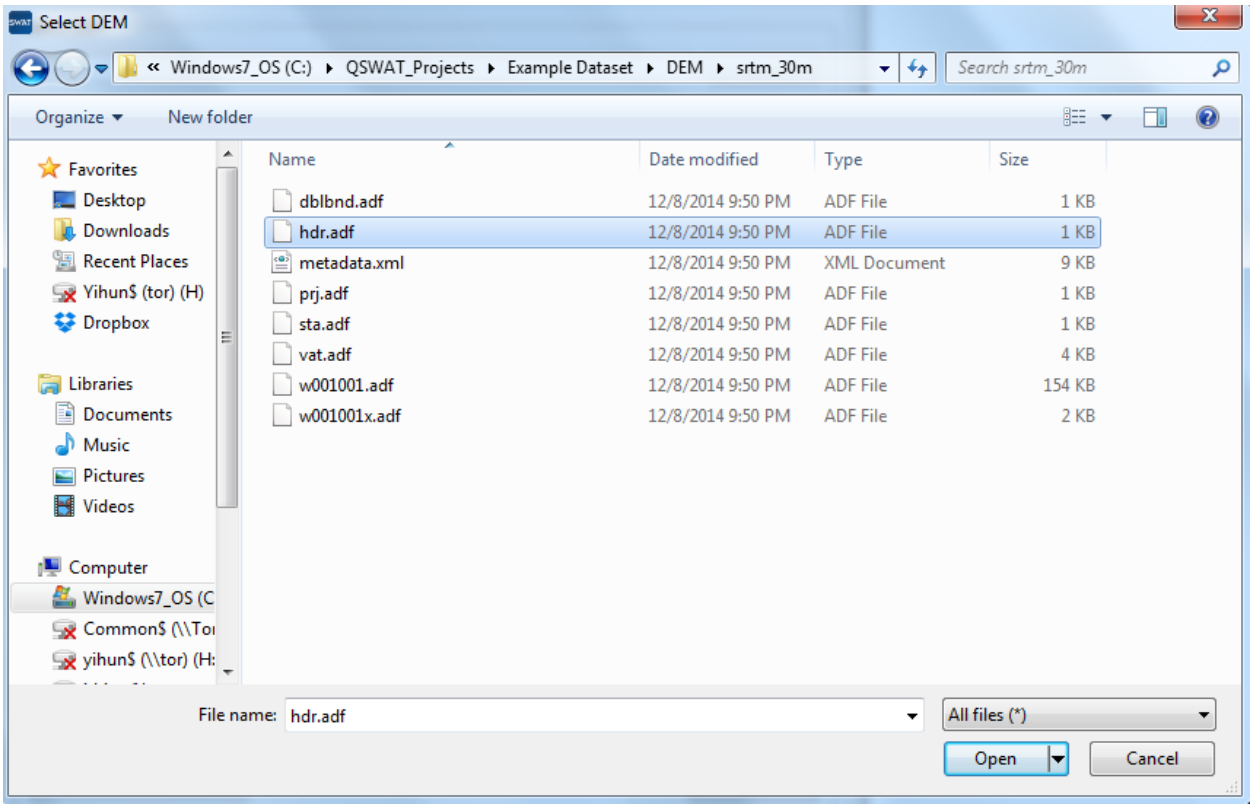


Figure 5: Selecting the DEM

8. 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.

9. Users can burn in an existing stream network using the option **Burn in existing stream network***;* use a file with the extension **.shp* (Figure 6). Check the *Burn in existing stream network* option and select the file *ExampleDataset\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 Stream Network, page 86).

We will not use MPI, which enables multiprocessing during watershed delineation (see Appendix

IV, page 89) and so the *Number of processes* spinbox should be set to 0.

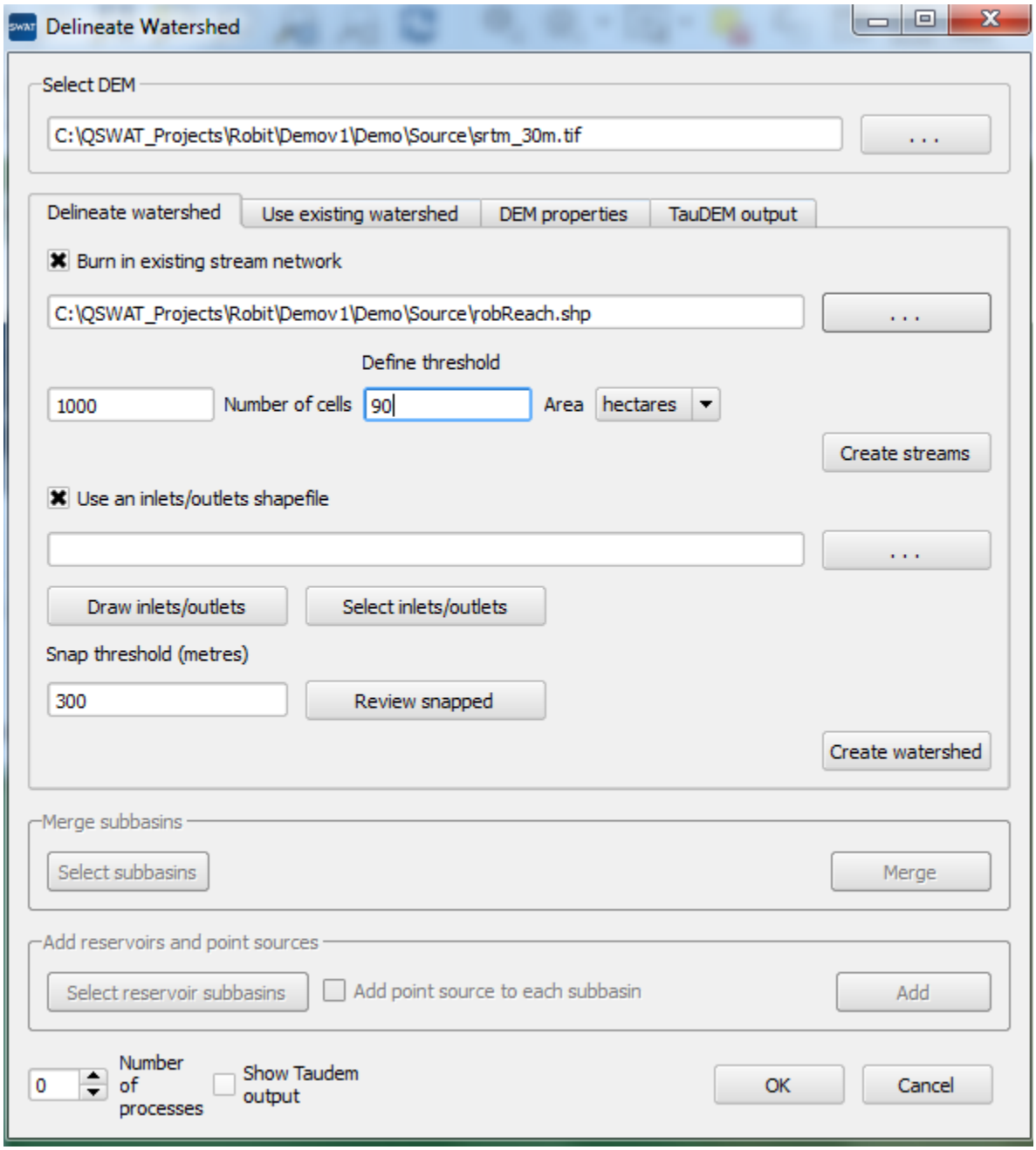


Figure 6: Creating stream networks

10. The threshold size for creating subbasins should be set next. It 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 number of hectares to 90, and press **Create Streams**: the number of cells will be adjusted to the corresponding value (1000). The threshold is the number of cells (or area) required to form a stream: a cell will be made part of a 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, stream junctions, and inlet, outlet, point source, and reservoir points that we may add later. A subbasin is an area draining into a stream reach. This step creates streams (Figure 7). It also loads the stream burn-in shapefile if selected.

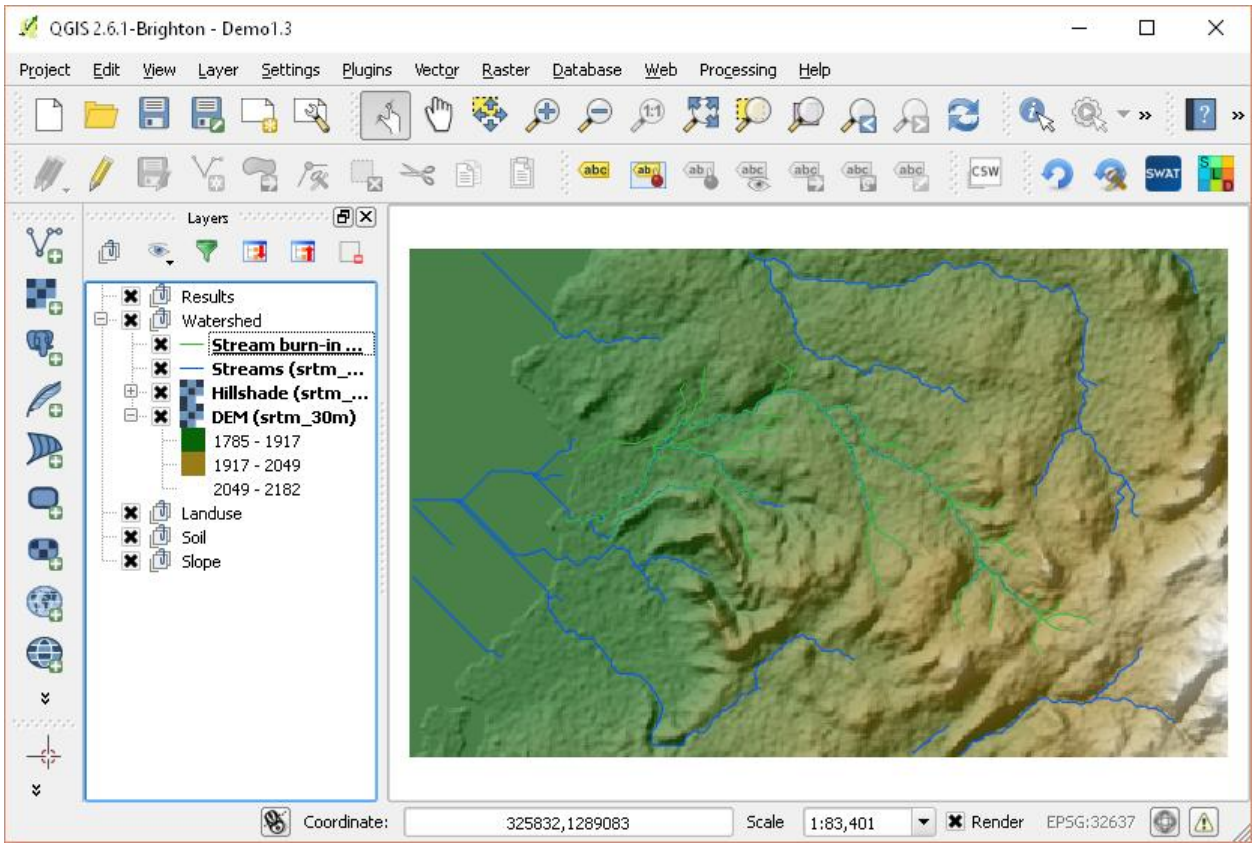


Figure 7: Stream networks displayed

11. 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 *MainOutlet/MainOutlet.shp*

12. Users can add outlets, reservoirs, 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. Points need to be placed on the stream network, and you may need to zoom in to place them precisely. Only points within the snap threshold of a stream 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** (page 86) for details of the fields needed in an inlets/outlets file. We do not add any additional outlets, reservoirs, inlets, or point sources in this example.

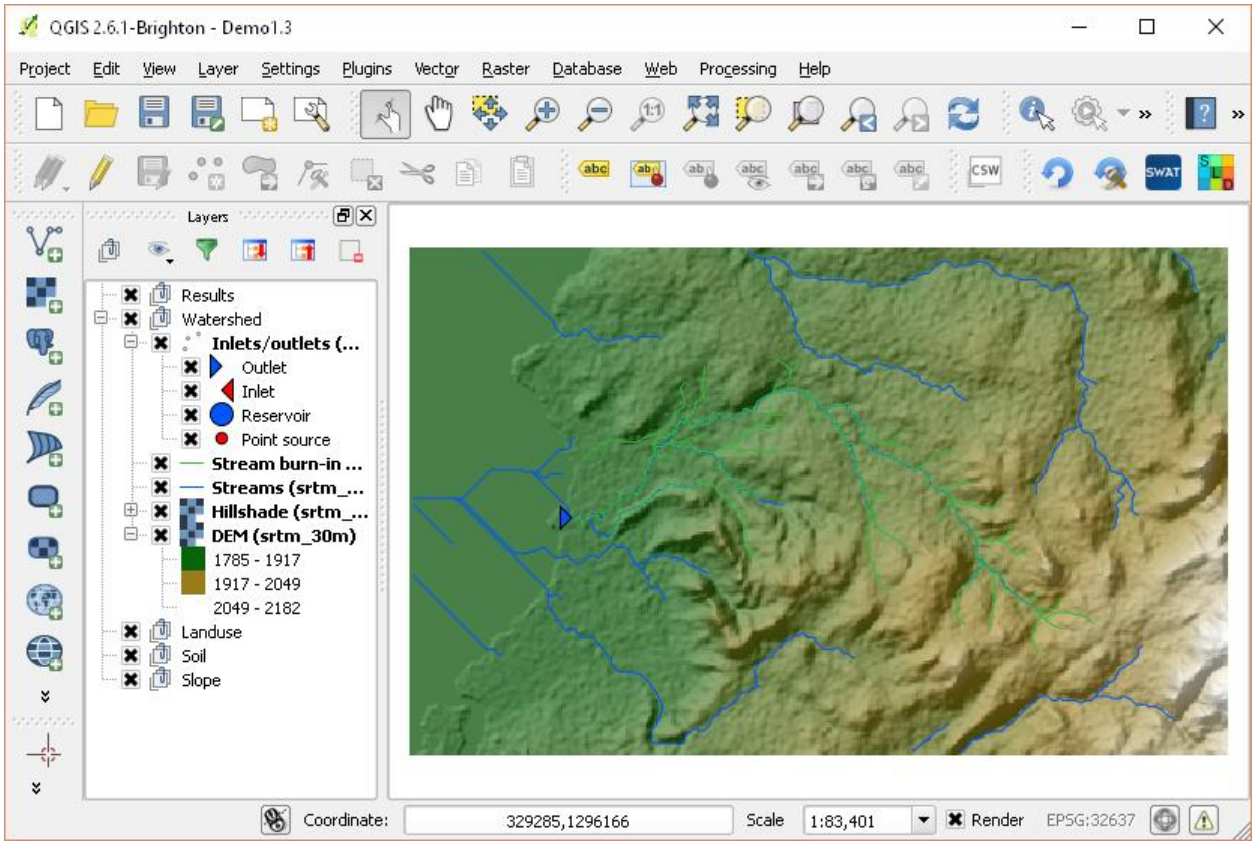


Figure 8: Step where Inlets/Outlets are added.

13. 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 (Figure 9). You only need to use this option if you want to only use a subset of outlets. The default is to use all the points in the outlets file or those drawn.

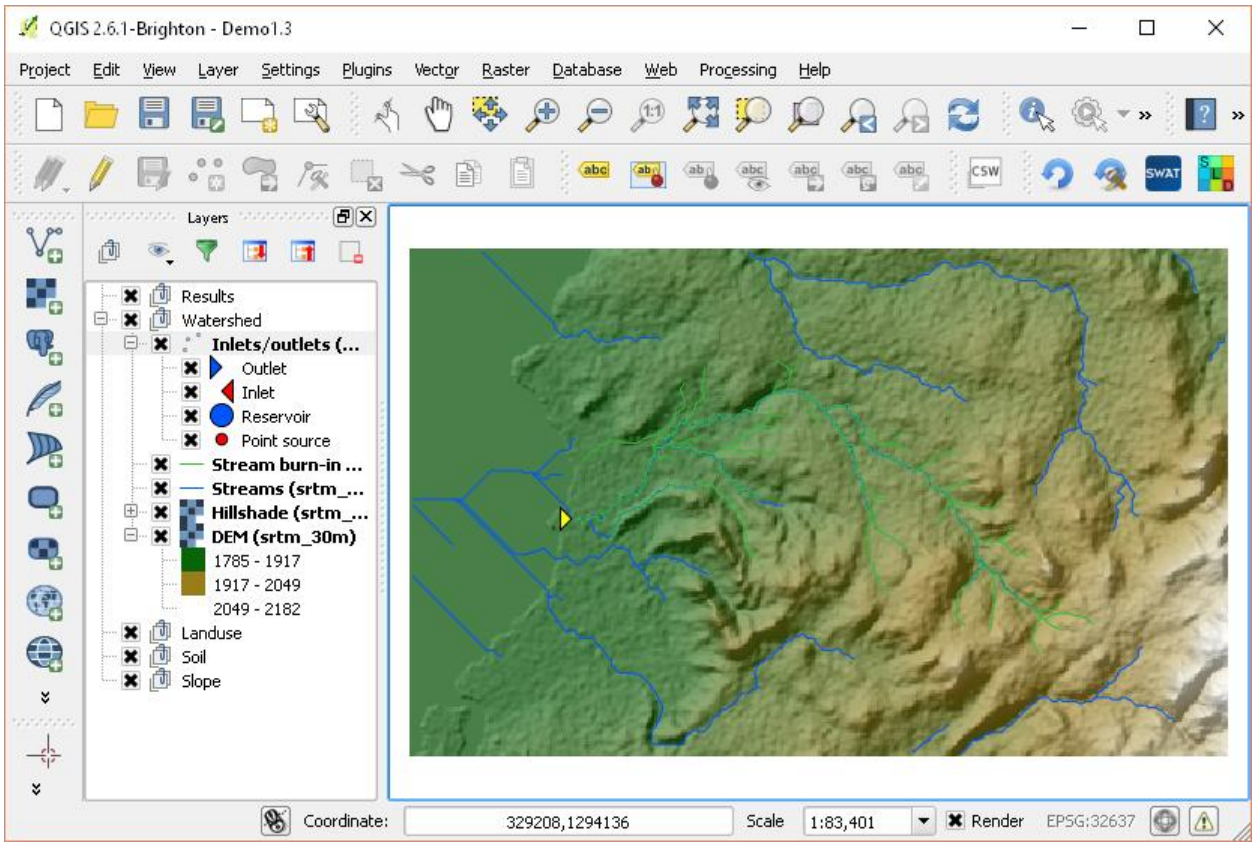


Figure 9: .Inlets/outlets selected

14. Clicking **Review snapped** shows the snapped inlets/outlets, i.e. those within the defined threshold distance. Clicking **Create Watershed** will create the watershed after a few minutes (Figure 10).

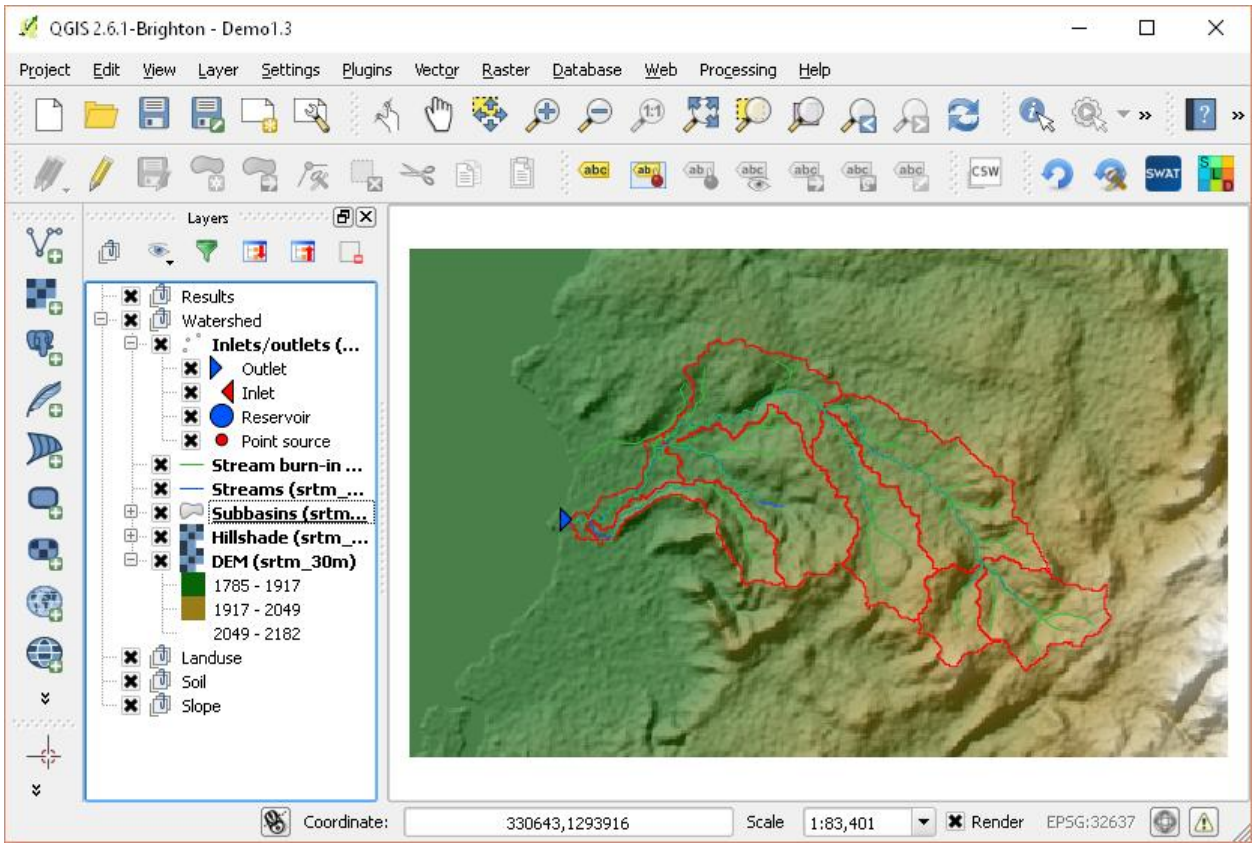


Figure 10: Delineated watersheds

15. In QSWAT users can select and merge subbasins. This is especially important in avoiding small subbasins. To merge subbasins, select the option **Select subbasins** under **Merge subbasins**. 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).

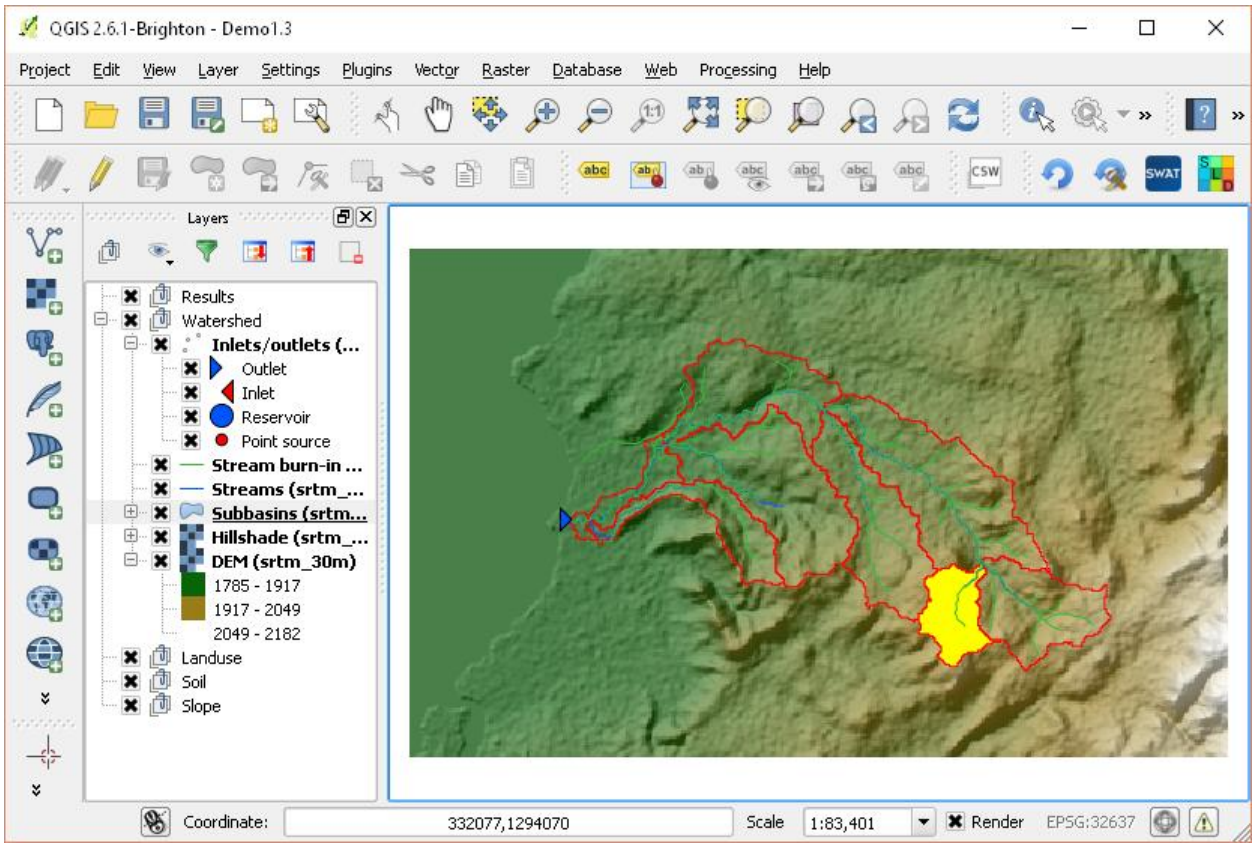


Figure 11: Step of merging subbasins

16. 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 put 25 in the box next to it. Click **Select**. It will show you subbasins that have an area less than 25% 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, inlet, reservoir, or point source. So in this example, the only small subbasin is that nearest the watershed outlet, which cannot be merged, and QSWAT will report this.

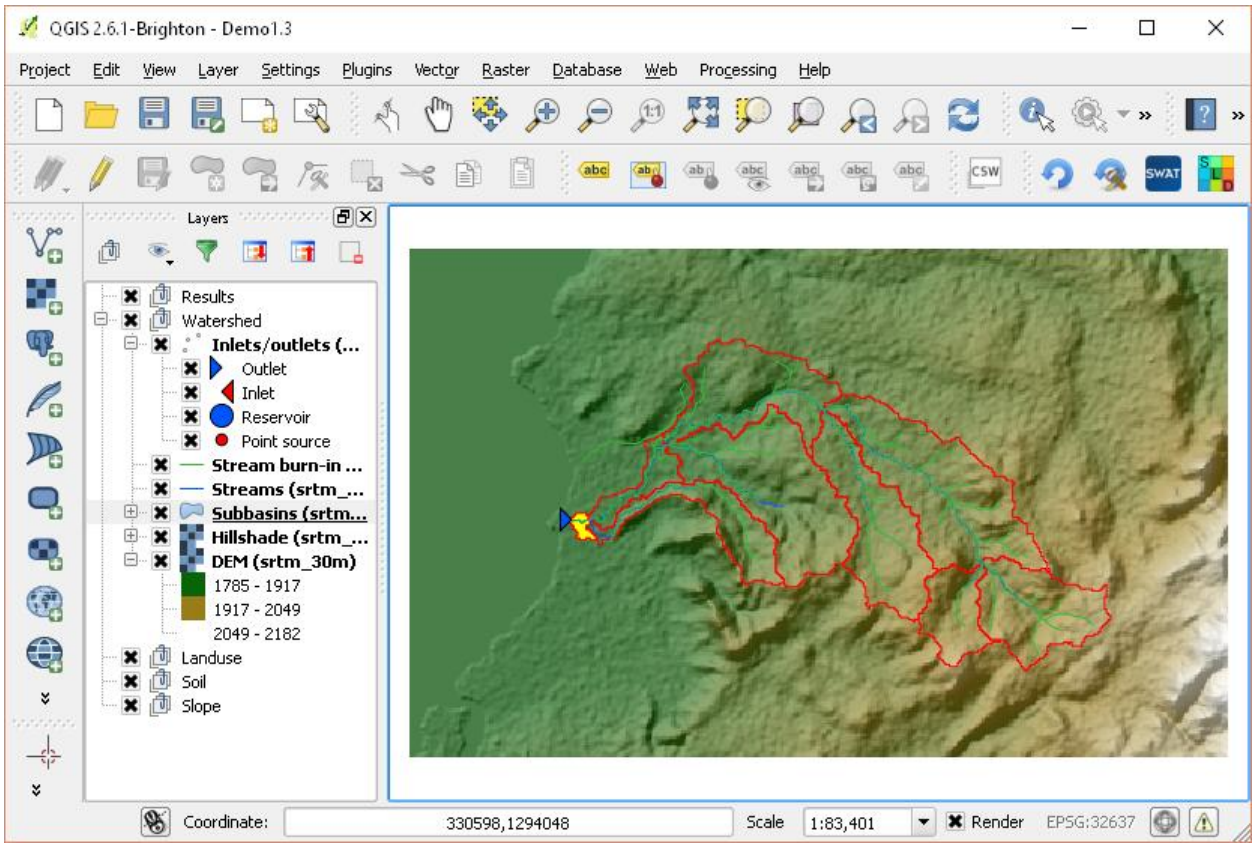


Figure 12: Subbasins that have an area less than 25% of the mean subbasin area merged to downstream subbasins

17. Point sources can be added to all the subbasins by checking the option **Add point source to each subbasin** (Figure 13). You can also add reservoirs to selected subbasins. You then need to click **Add**. Unlike points added earlier, these extra point sources or reservoirs do not create subbasin boundaries, but are added to existing subbasins, provided they do not already have point sources or reservoirs. Adding point sources to all subbasins is useful if you have several to add. It is quite safe since only those you provide input data for will have any effect on the SWAT simulation.

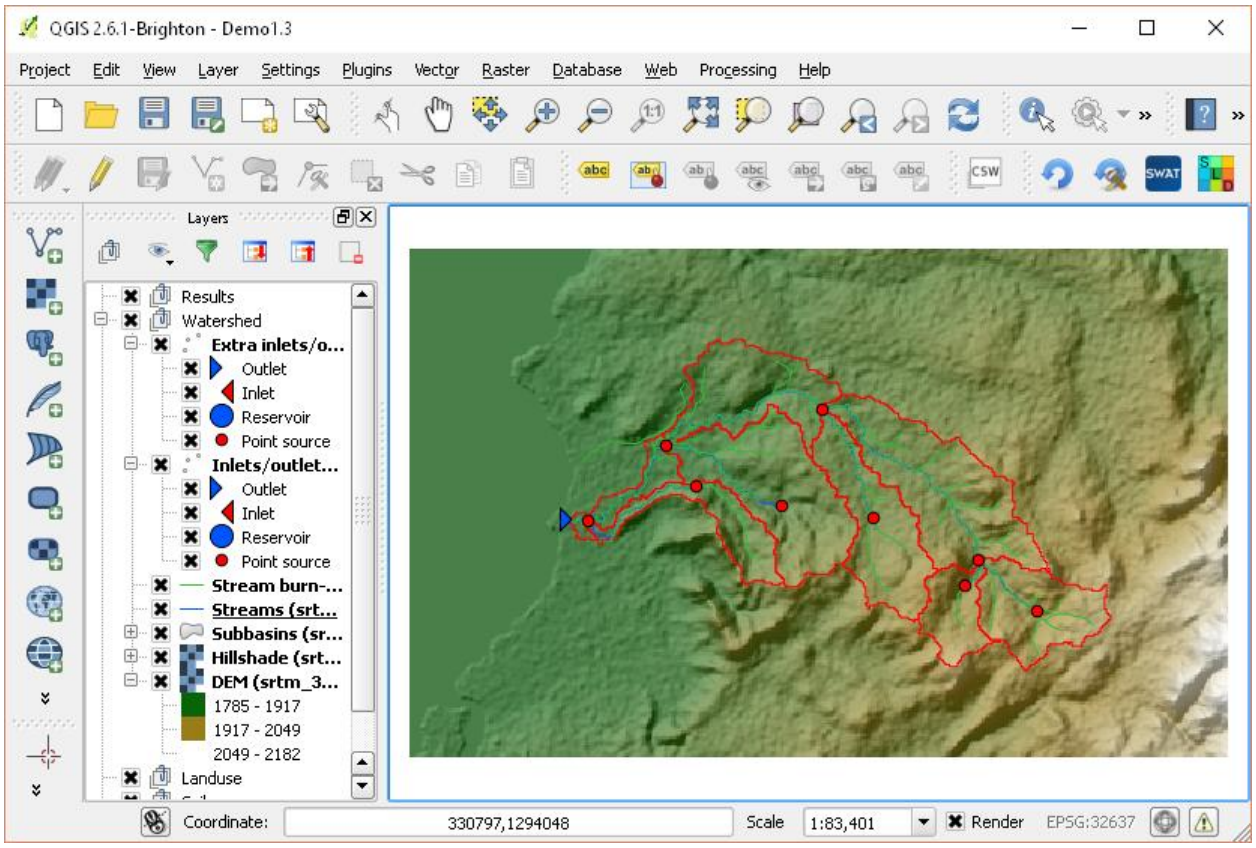


Figure 13: Point sources added to each subbasin

18. You need to click **OK**. This causes the subbasins to be numbered and calculates and stores information about the subbasins and streams. This step ends watershed delineation, and enables the **Create HRUs** option (Figure 14).

Figure 14: Watershed delineation is completed, and Create HRUs is enabled

5.2. Create HRUs

19. 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.

20. To do this we click **Create HRUs**, select *ExampleDataset\Landuse\roblandusenew\hdr.adf* as the **Landuse map**, and select *ExampleDataset\ Soil\mowr_soil90\hdr.adf* as the **Soil map**.

21. 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.

22. For landuse, select the option *Use csv file* in the **Landuse table** pull-down menu. Make the same selection for **Soil table**. Make sure the **Soil data** option is set to *usersoil*.. The other soil data options are for watersheds in the USA for which you are using STATSGO, STATSGO2, or SSURGO soil maps. To use STATSGO2 or SSURGO soil maps you will need to download *SWAT_US_SSURGO_Soils.mdb* from http://swat.tamu.edu/media/63316/SWAT_US_SSURGO_Soils.zip and install it in *C:\SWAT\SWATEditor\Databases.* No soil lookup table is needed with STATSGO2 or SSURGO.

23. 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. To generate the **FullHRUs** shapefile, click in the check box.

24. 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.

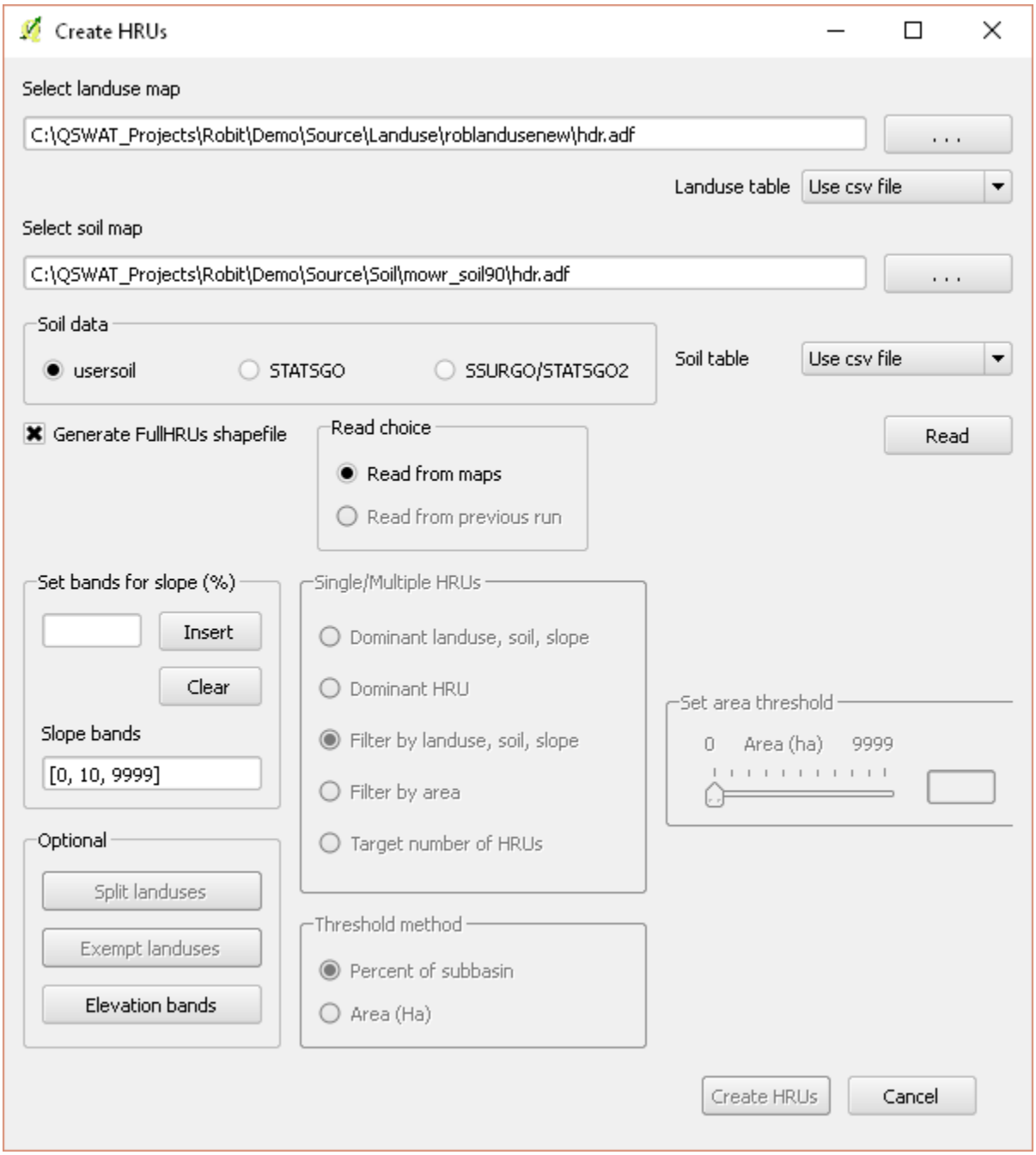


Figure 15: Reading the land use, soil and slope maps

25. In our case, we have chose to make landuse and soil lookup tables from .csv files, so when we click **Read** we are first asked to select the files. For landuse, navigate to *Robit_landuses.csv* in *ExampleDataset*. This has the appropriate structure for a landuse lookup table. Its contents are shown in Figure 16. 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 four-letter (unless it is HAY) SWAT landuse code, as found in one of the tables *crop* or *urban* in the reference database. More than one map value may map to the same landuse code, but map values may not be repeated.

```
LANDUSE_ID,SWAT_CODE
1,WATR
2,AGRL
3,PAST
4,FRST
```

Figure 16: Robit_landuses.csv

When we select the csv file, it is imported into the project database to make a new lookup table. Its 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. It will be selected in the **Landuse table** menu and used by default next time we run the project - we only need to import the csv file once.

26. The csv file for soils is *ExampleDataset\Robit_soils.csv* and in the same fashion will make a lookup table *Robit_soils*. It maps values from our soil map to soil names found in the *SNAM* field of the *usersoil* table we made earlier in the reference database (as described in Appendix I, page 66).

27. After reading the grids you will notice a number of changes to the QSWAT display (Figure 17):

- 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 legends for the landuse map *robelandusenew* and the soil map *mowr_soil90* include the landuse and soil categories from the lookup tables.
- A shapefile *FullHRUs* has been created and added. This allows users to see where in each subbasin the potential Hydrological Response Units (HRUs) are physically located. If, for example, we zoom in on subbasin 2, in the Legend panel select (left button) FullHRUs, open its attribute table (right button), set the mouse to *Select Feature(s)* (QGIS toolbar) and click in the map canvas on the box below the number 2, then we get a view like Figure 18. We see that this potential HRU is composed of several parts, has the landuse AGRL, the soil LVx, and the slope band *0-10*. Its area of 172 ha is 50.1% of the subbasin. *Close* the Attribute Table Editor. Generating the FullHRUs shapefile is optional, as it can

take some time. A count of the potential HRUs (51 in this example) is displayed whether this file is created or not.

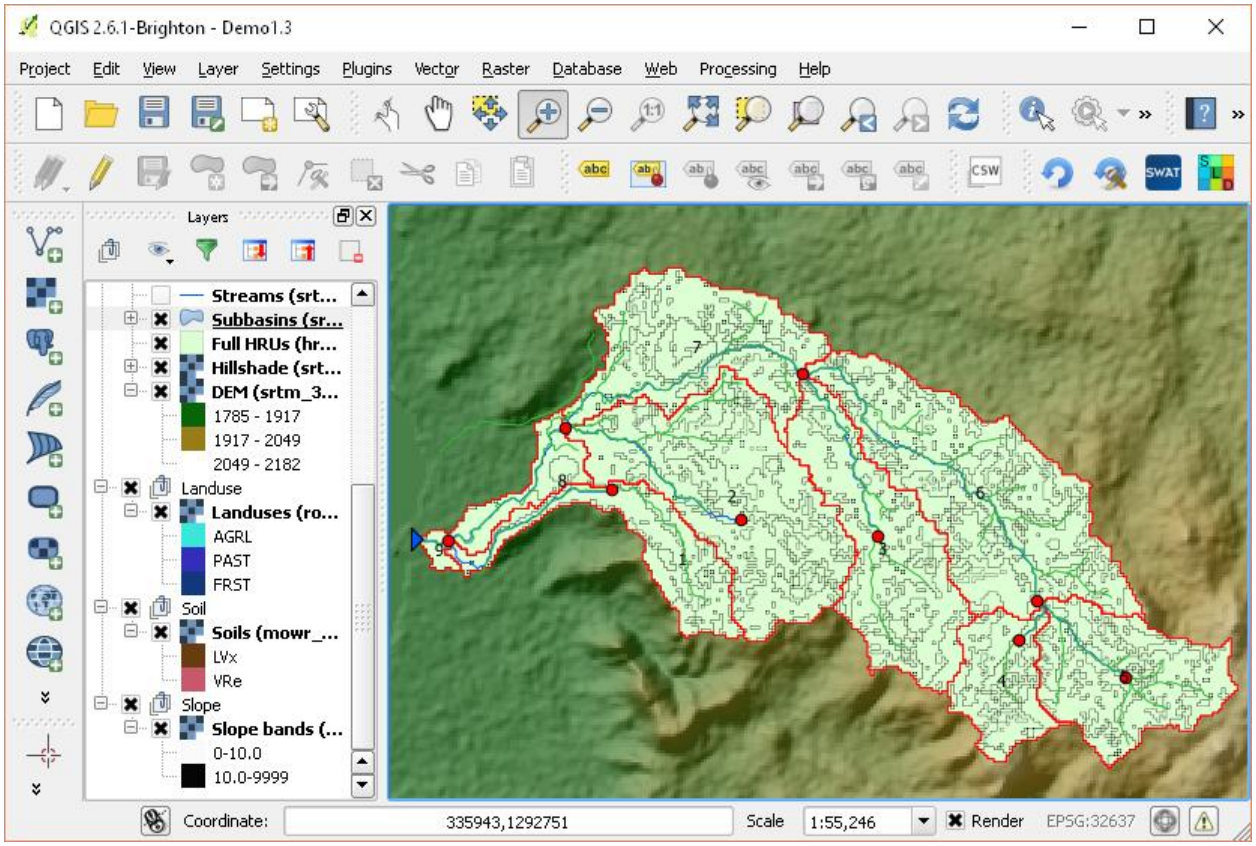


Figure 17: After reading grids

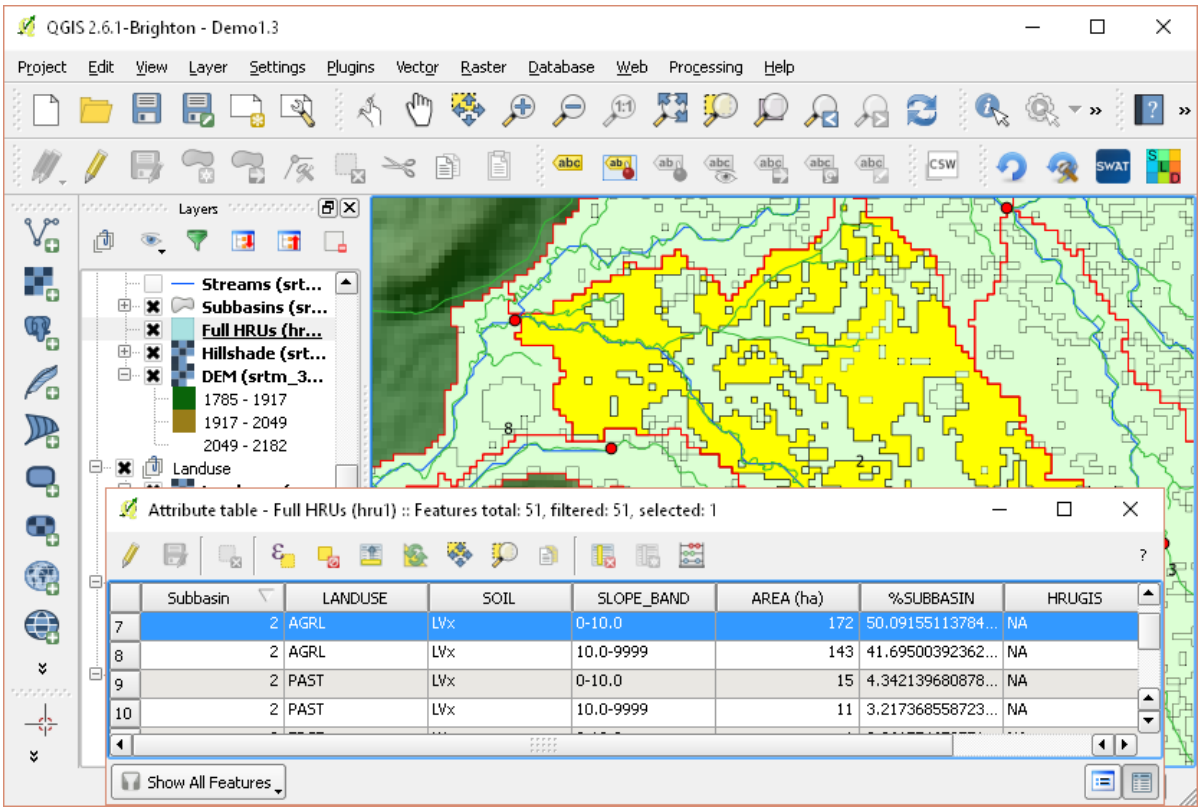
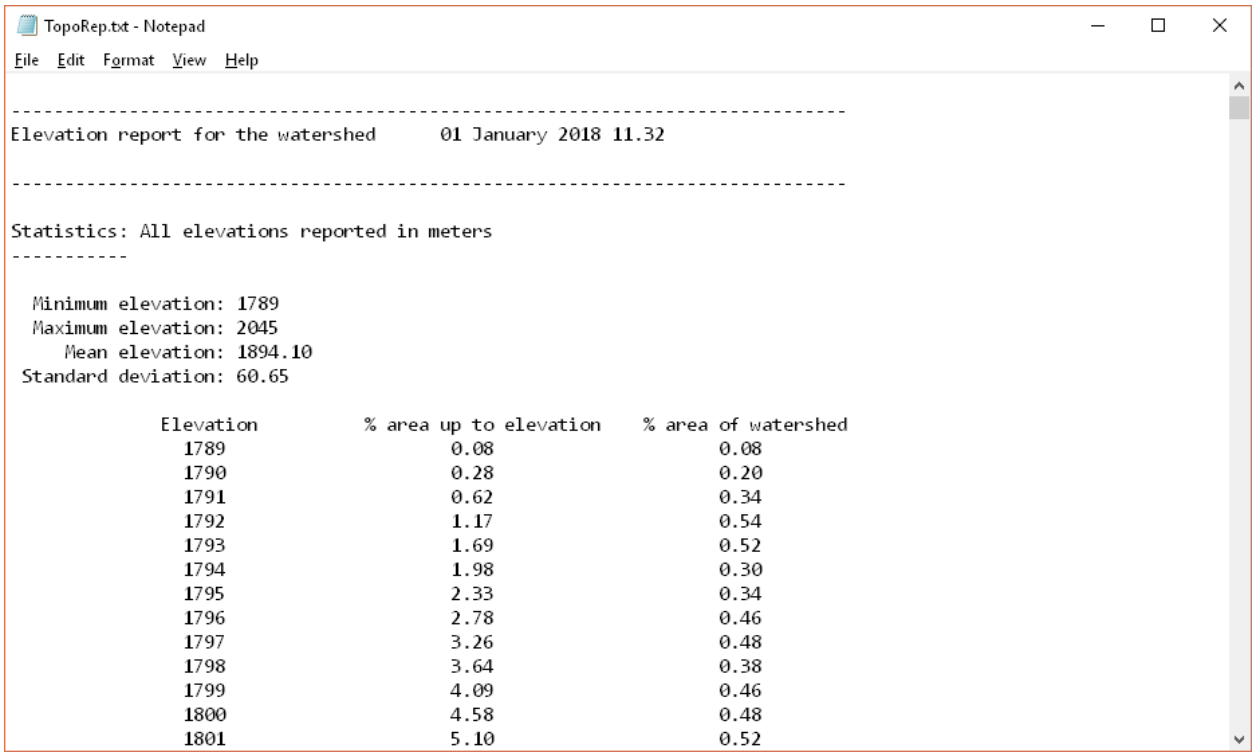


Figure 18: Viewing a particular HRU

28. 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 19). The landuse and soil report lists the landuse, soil and slope-band areas for each subbasin (Figure 20).

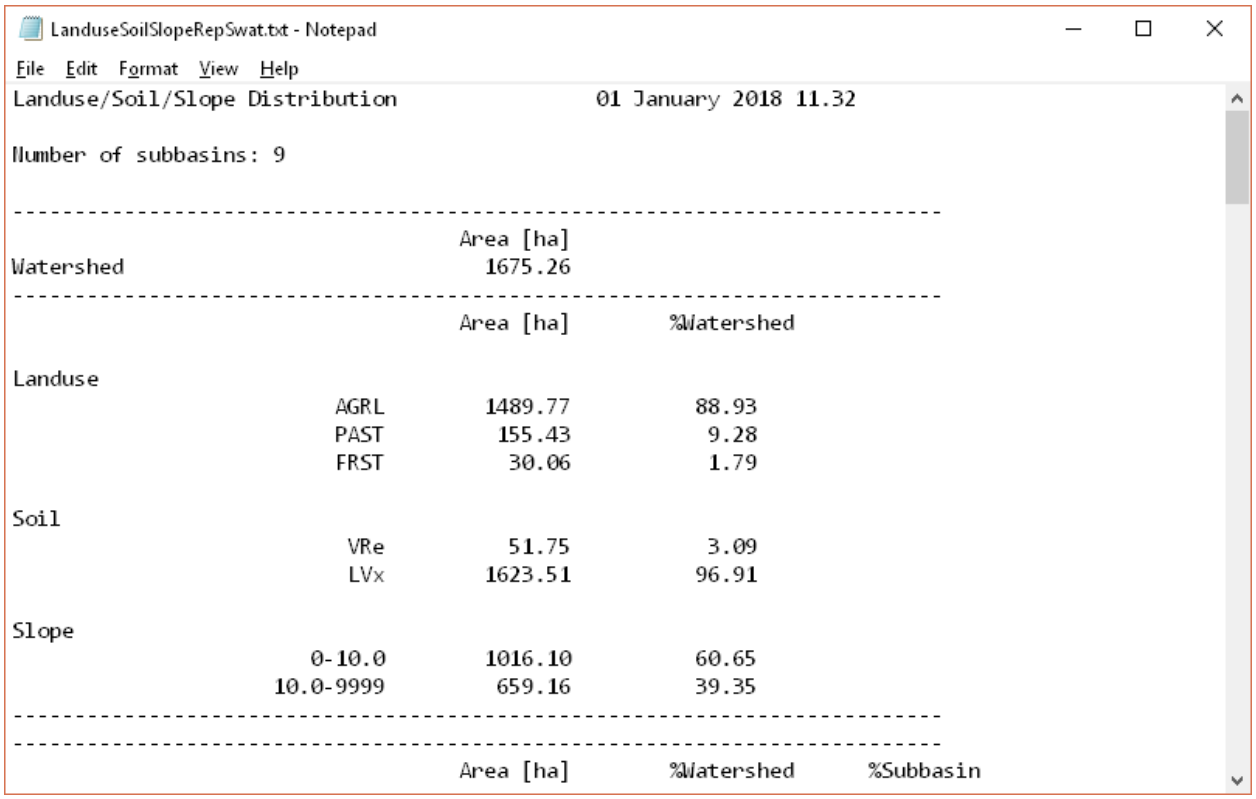


TopoRep.txt - Notepad

File Edit Format View Help

```
--------------------------------------------------------------------------------
Elevation report for the watershed      01 January 2018 11.32

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

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
                1800                     4.58                     0.48
                1801                     5.10                     0.52
```

Figure 19: Elevation report



LanduseSoilSlopeRepSwat.txt - Notepad

File Edit Format View Help

```
Landuse/Soil/Slope Distribution                01 January 2018 11.32

Number of subbasins: 9

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

Landuse
                          AGRL        1489.77          88.93
                          PAST         155.43           9.28
                          FRST          30.06           1.79

Soil
                           VRe          51.75           3.09
                           LVx        1623.51          96.91

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

Figure 20: Landuse and soil report

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

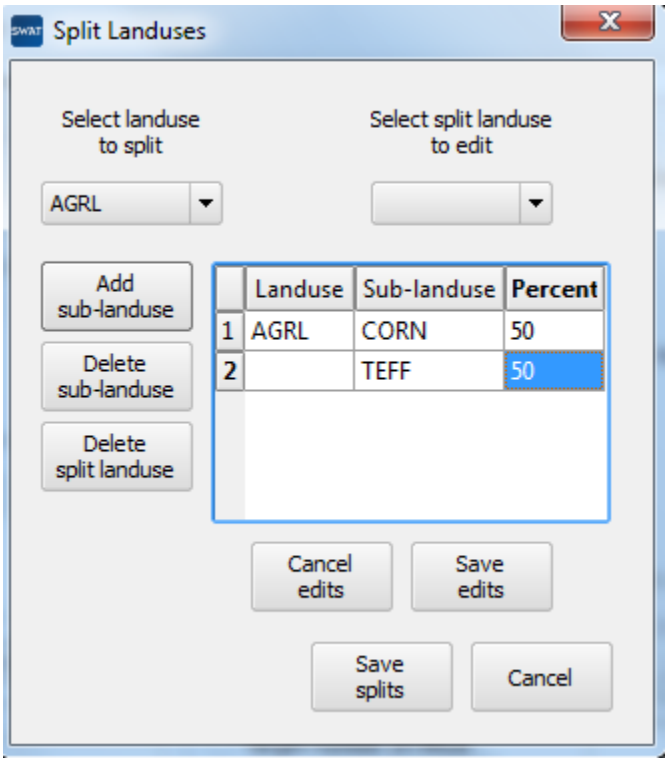


Figure 21: Splitting a landuse

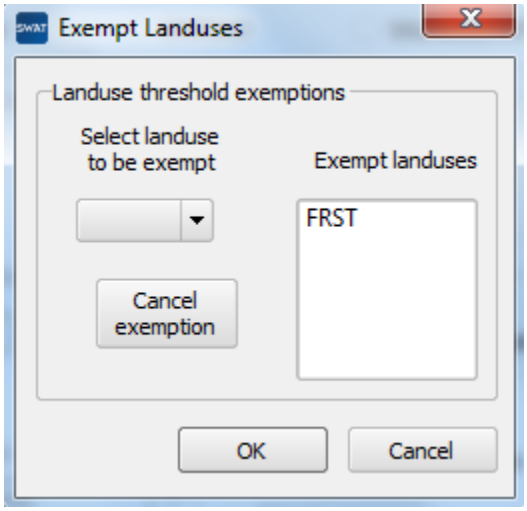


Figure 22: Exempting a landuse

30. 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 subbasin (i.e. option of **Single HRU**). **Dominant landuse, soil, slope** chooses the landuse

with the biggest area in the subbasin, the soil with the biggest area in the subbasin, and the slope range with the biggest area in the subbasin and uses them for the whole subbasin. **Dominant HRU** selects the largest of the potential HRUs in each subbasin and makes its landuse, soil and slope range the ones chosen for the whole subbasin.

The options with **Filter by landuse, soil, slope; Filter by area;** and **Target number of HRUs** create **Multiple HRUs**. For the **Multiple HRUs** option, users can exclude HRUs that are insignificant by considering **percentage thresholds** or **area thresholds**. The option of **Filter by 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. The option of **Filter by area** will ignore HRUs that are below a specified threshold amount (either percent of the subbasin, or area). Finally the option of **Target number of HRUs** limits the number of HRUs to a user preferred amount (i.e. between the lower limit of one HRU per subbasin and the upper one of retaining all potential HRUs).

For this exercise, use the method of **Filter by land use, soil and slope** and choose the **Threshold method** of **Percent of subbasin**. For landuse, the value of 81 as the maximum we can choose indicates that there is a subbasin where the maximum value for a landuse is 81%. If we chose a higher value than 81% we would be trying to ignore all the landuse categories in that subbasin. Hence 81% is the minimum across the subbasins of the maximum landuse percentage in each subbasin. 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 52%. 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 (54) is reported at the top of the map canvas.

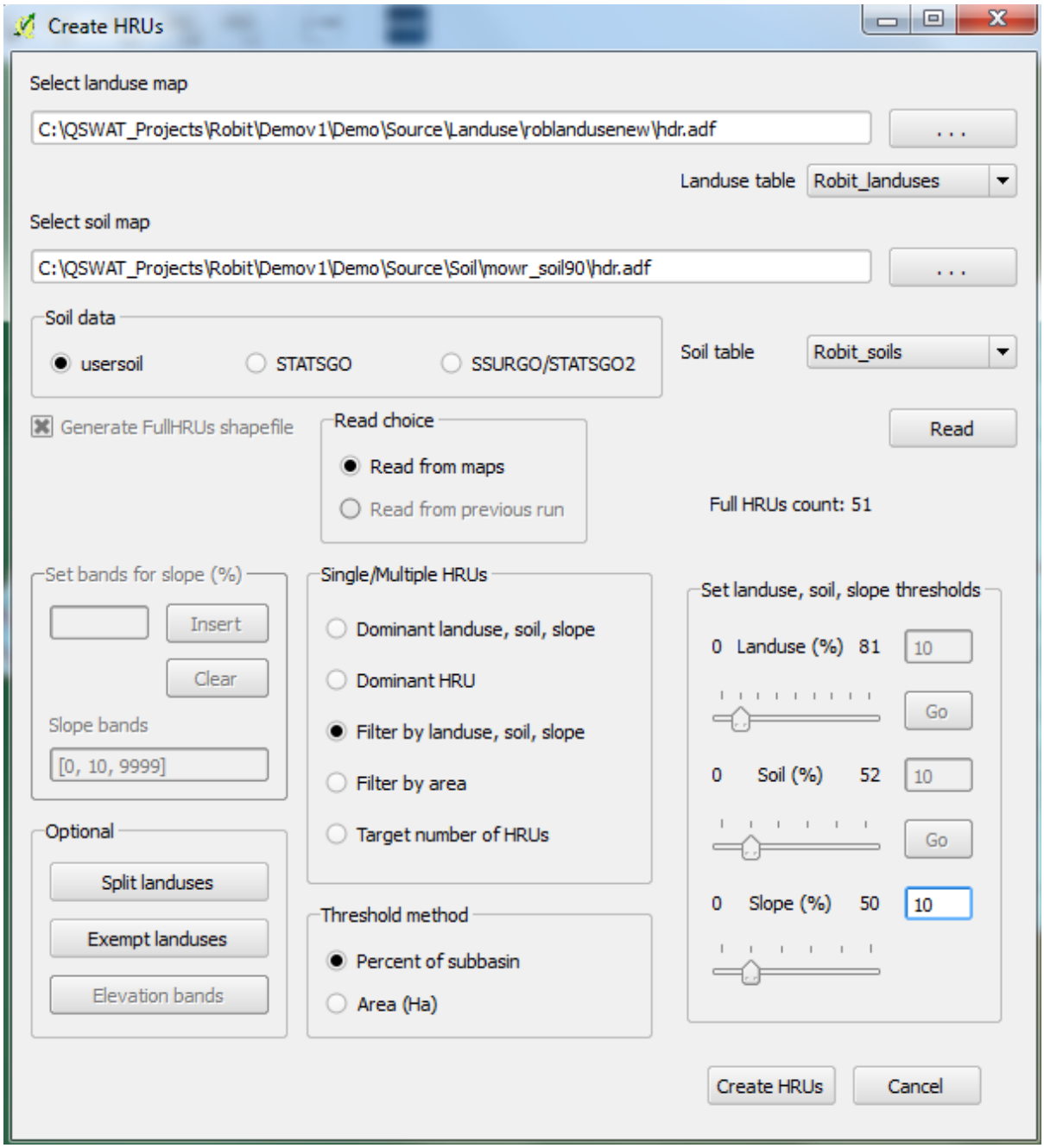


Figure 23: Creating multiple HRUs by Filter by land use, soil and slope

31. **Create HRUs** is now reported as done and the third step is enabled (Figure 24). If you requested a *Full HRUs* file you will notice that an *Actual HRUs* file has now been added. This contains the potential HRUs that were retained as actual HRUs (and so typically has some holes representing those that were not retained). This file also shows the numbering of HRUs as well

as subbasins. If you open its attribute table you will see in the last column, called *HRUGIS*, values like 000050001, which indicates that this row is for HRU 1 in subbasin 5. Input files for this HRU (found in the *TxtInOut* folder) use the HRUGIS value as their base file name. HRUs that were split, those for landuse AGRL, have two HRUGIS values, since they will have two sets of HRU input files, one for CORN and one for TEFF. 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, check **Read from previous run** and click **Read** to reload the map data, change the thresholds and/or the landuses to be split or exempted, and rerun the **Create HRUs** step.

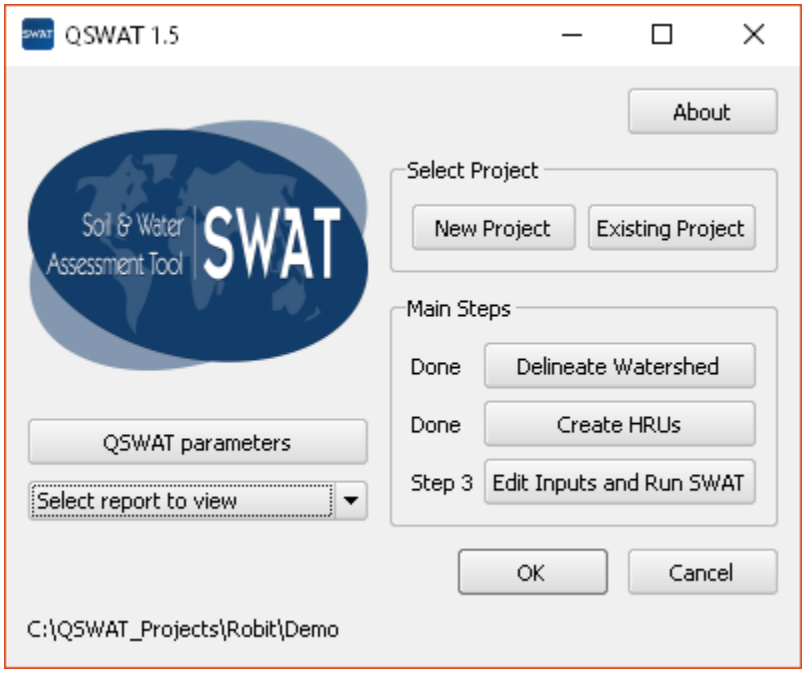
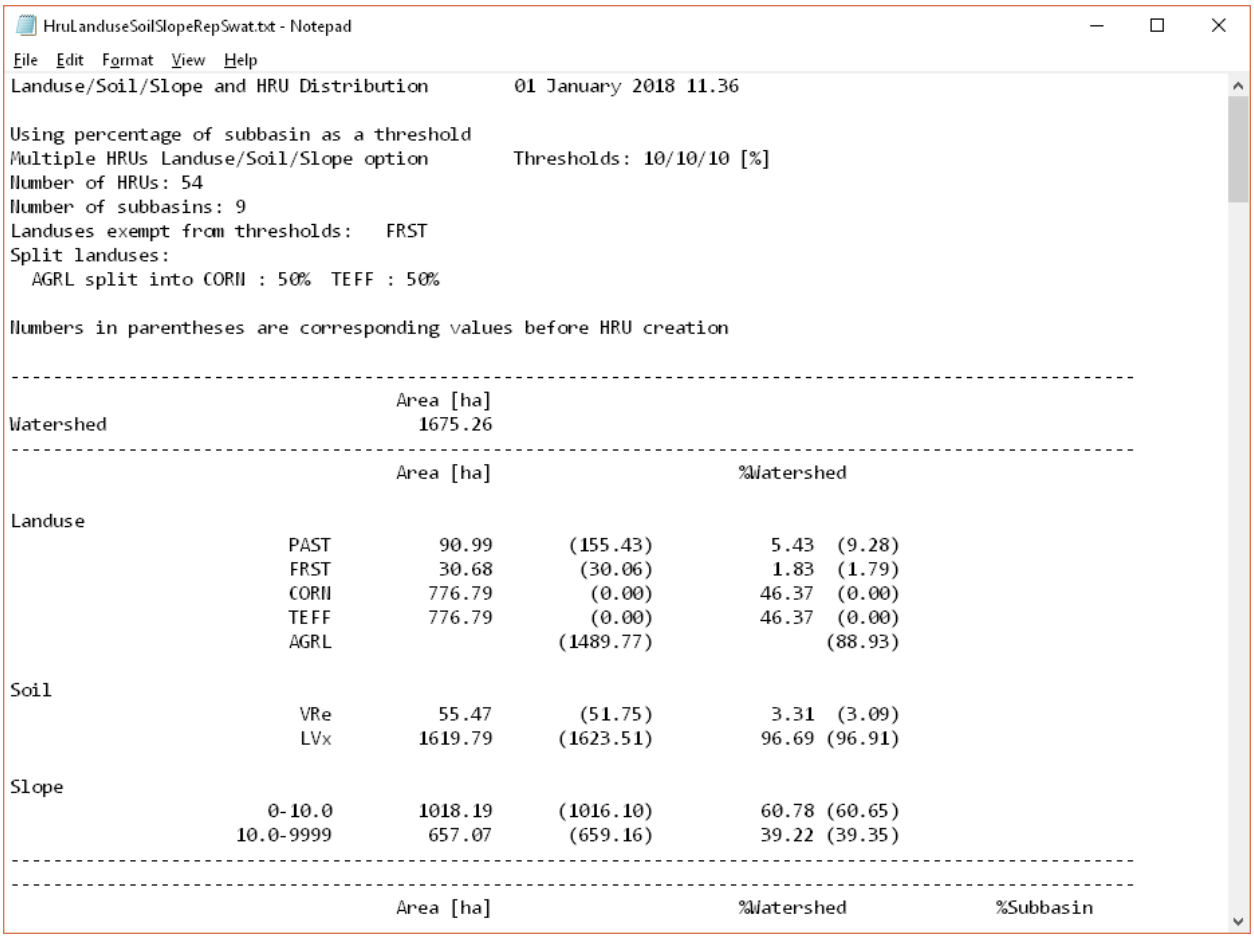


Figure 24: About to do step 3



HruLanduseSoilSlopeRepSwat.txt - Notepad

File Edit Format View Help

```
Landuse/Soil/Slope and HRU Distribution        01 January 2018 11.36

Using percentage of subbasin as a threshold
Multiple HRUs Landuse/Soil/Slope option        Thresholds: 10/10/10 [%]
Number of HRUs: 54
Number of subbasins: 9
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

Landuse
                        PAST        90.99       (155.43)          5.43  (9.28)
                        FRST        30.68        (30.06)          1.83  (1.79)
                        CORN       776.79         (0.00)         46.37  (0.00)
                        TEFF       776.79         (0.00)         46.37  (0.00)
                        AGRL                    (1489.77)               (88.93)

Soil
                          VRe       55.47        (51.75)          3.31  (3.09)
                          LVx     1619.79      (1623.51)         96.69 (96.91)

Slope
                       0-10.0     1018.19      (1016.10)         60.78 (60.65)
                     10.0-9999     657.07       (659.16)         39.22 (39.35)
-------------------------------------------------------------------------------------------------------------
-------------------------------------------------------------------------------------------------------------
                                Area [ha]                    %Watershed            %Subbasin
```

Figure 25: HRUs report

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

5.3. SWAT Setup and Run

33. At this point almost everything is ready to write the SWAT input files and run SWAT. The SWAT Editor assumes that dates are written in the **mm/dd/yyyy** format. You may need to change to this system on your machine, by opening the Control Panel, selecting **Region** and setting **English (United States)** as the format. On later versions of Windows you find *Settings*, select *Time & Language*, *Change date and time formats*, and select for *Short date* either *M/d/yy* or one of the similar patterns that have the month/day/year pattern, differing only in the default number of digits. Otherwise you may get messages about *invalid date*. Click **Edit Inputs and Run SWAT**. This starts the SWAT Editor (Figure 26).

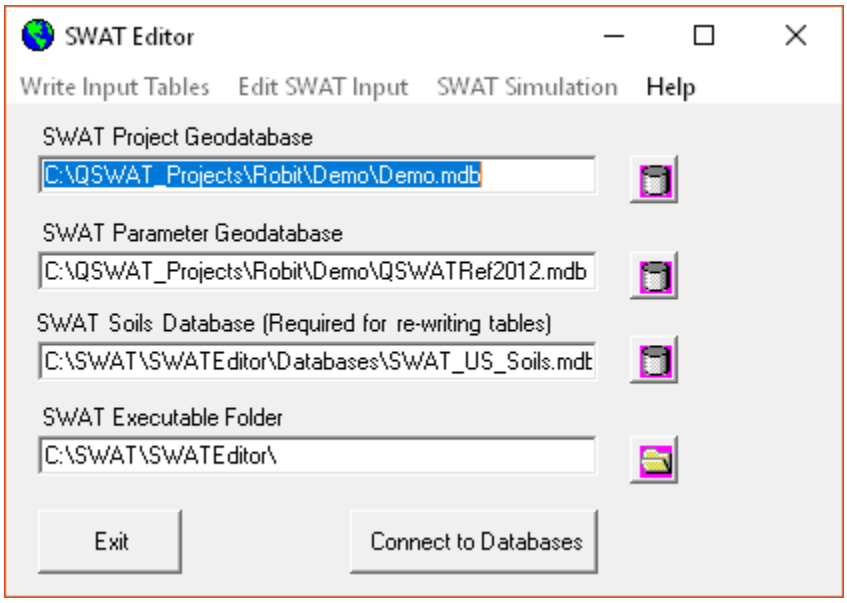


Figure 26: Connecting the Demo project to SWAT databases

34. The local SWAT project database and reference database (referred to as a *Parameter Geodatabase)* have been selected in the first two fields. An American soils database is selected in the third, and the SWAT Editor folder is selected in the third. These can be changed if necessary, but should be OK already. Click **Connect to Databases** to connect to the project database and to the reference.

35. Now the **Write Input Tables** form will be active. Click the option **Weather Stations.**

36. There are two kinds of possible weather information used by SWAT, weather generator data and actual weather data. Weather generator data is data about the typical weather in a locality, and its variability. It is input to a weather generator tool that will generate pseudo random data, and this is used when actual weather data is missing, either for particular dates, or for particular categories of weather, such as solar radiation. (SWAT insists on a weather generator being defined for each subbasin even if there is no missing actual data.) Weather generator data is provided by a table of virtual *weather generator stations*, each with its own location and data. The SWAT Editor assigns to each subbasin the weather generator station nearest to its centroid. For our example we have a weather generator table **WGEN_Robit** that you should have imported into the reference database earlier. Clicking the **Weather Data Generator** tab, and choose **WGEN_Robit**.

37. Actual weather data can be input in 5 different categories: precipitation, temperature,

solar radiation, relative humidity and wind speed. In each category for which data is available, the user supplies two kinds of files. The first is a weather station file, containing the names and locations of weather stations. For each station there will be a data file containing data. For example, we have for rainfall a file *pcpRobStation.txt* that contains one precipitation station (a rainfall gauge) called *robpcp*. This name informs the SWAT Editor that the precipitation data for this gauge will be found (in the same folder) in a file *robpcp.txt.* The file names used are chosen by the user, with the sole proviso that the data file names must match the station names used in the stations file. The SWAT Editor needs only to be provided with the stations files, since it can then find the data files. As with weather generator station files, there may be many stations in each file, and again each subbasin is assigned its nearest station in each category.

38. We have actual data in all five categories, stored in the folder *ExampleDataset\ClimateRobit*. The station files are *pcpRobStation.txt, tmpRobStation.txt, solRobStation.txt, rhumRobStation.txt,* and *windRobStation.txt* for precipitation, temperature, solar radiation, relative humidity, and wind speed respectively. For each tab for rainfall, temperature, etc. check the option for gages rather than simulation and select the appropriate stations file. Click **OK.** The climate data files (which will be accessed by the SWAT Editor using station information as explained above) are: *robpcp.txt, robtmp.txt, robsol.txt, robrhum.txt,* and *robwind.txt*. Click **Cancel** to close the Weather Data Definition form.

39. To write the remaining input tables and files select **Write Input Tables → Write SWAT Input Tables**. Click **Select All** in the list box (Figure 27) and click the **Create Tables** button. There is a program for estimating the heat units needed for plant growth, using climate data, but it is currently available only for the USA. Since this project is outside the USA, choose the option **No**. The heat units value appears in the management (*.mgt) files, and you can later edit these values using the SWAT Editor. Also answer **No** to writing point source, reservoir and inlet information. Writing the database tables can take a few minutes, especially if your simulation covers several years, but it is fast in this case. A form announces that the tables have been built, where you click **OK** to acknowledge. If all is well the form will show that all database tables and files have been written successfully (Figure 27). Click **Cancel**.

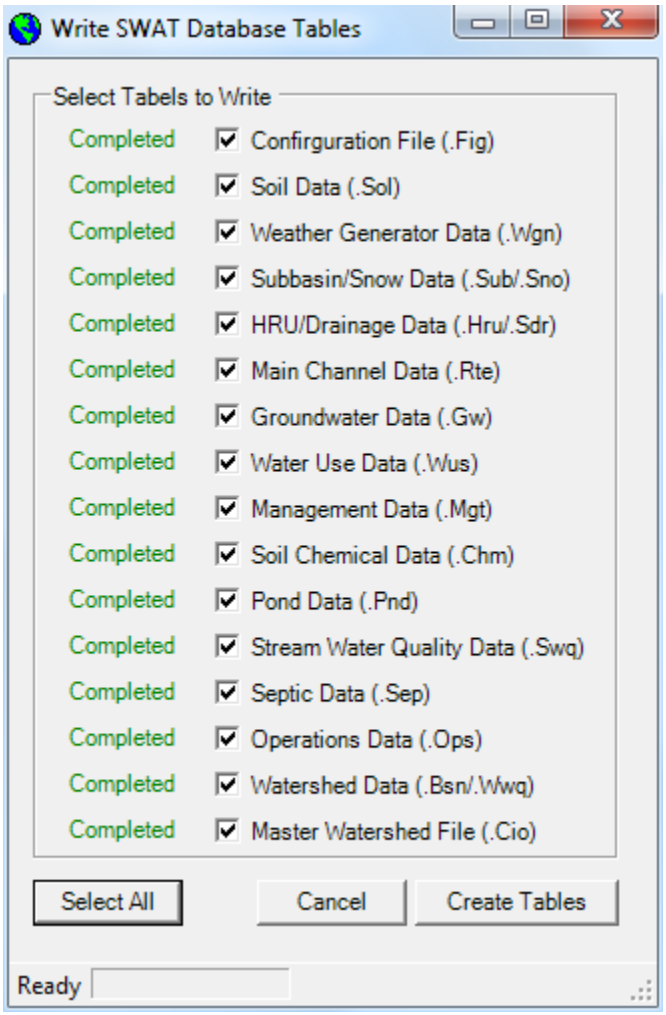


Figure 27: Finished writing the SWAT database tables and input files

40. Now the user can edit different SWAT databases and simulation conditions (e.g. methods to estimate potential evapotranspiration, stream flow routing, etc). This is also the step where users can implement different management scenarios.

41. To run SWAT click **SWAT Simulation → Run SWAT**. Since we provided actual climate data (for 24 years from the start of 1900), the SWAT Editor has calculated the minimum and maximum dates that we can use. Set the time periods to run SWAT to 01/01/1990 to 12/31/2000 (remember to use the m/d/y format!), chose **Monthly** time step, and **NYSKIP** (initial number of years to skip of 3), check the **Print Water Quality Output** option, and leave the **Output File Variables** set to **All**. Select a release version of SWAT.exe, 64-bit if you have a 64-bit machine, else 32-bit. Finally click **Setup SWAT Run** (Figure 28). This step activates **Run SWAT**.

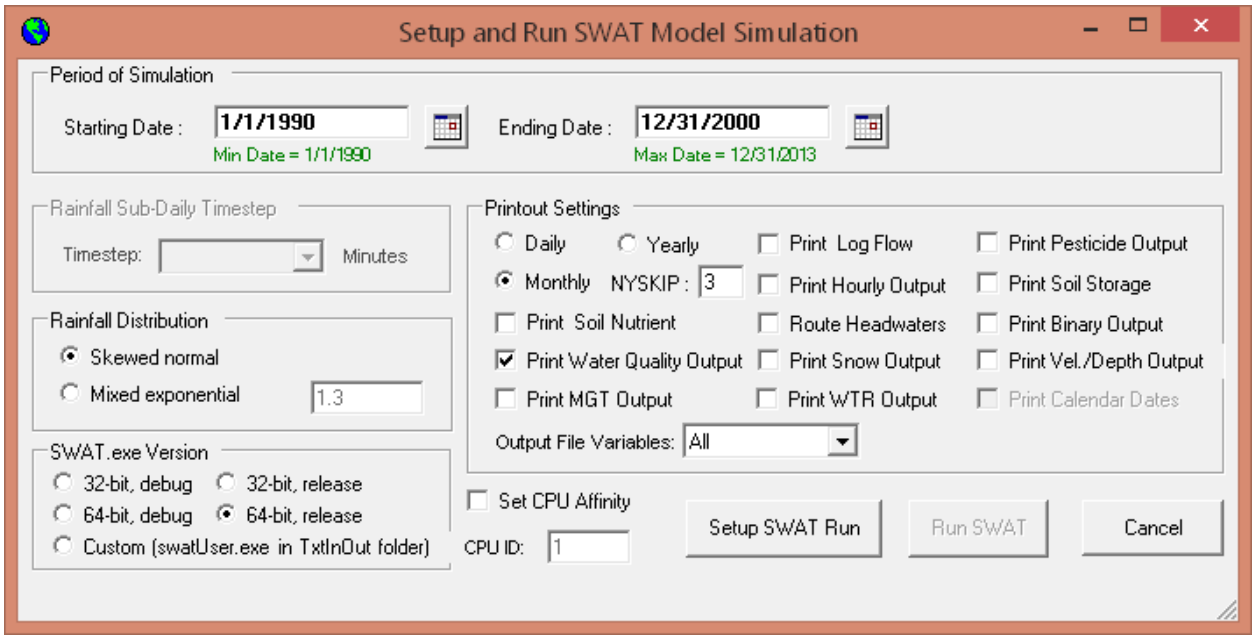


Figure 28: Setting up SWAT Run

42. Click **Run SWAT** to launch the SWAT executable in a command window (Figure 29).

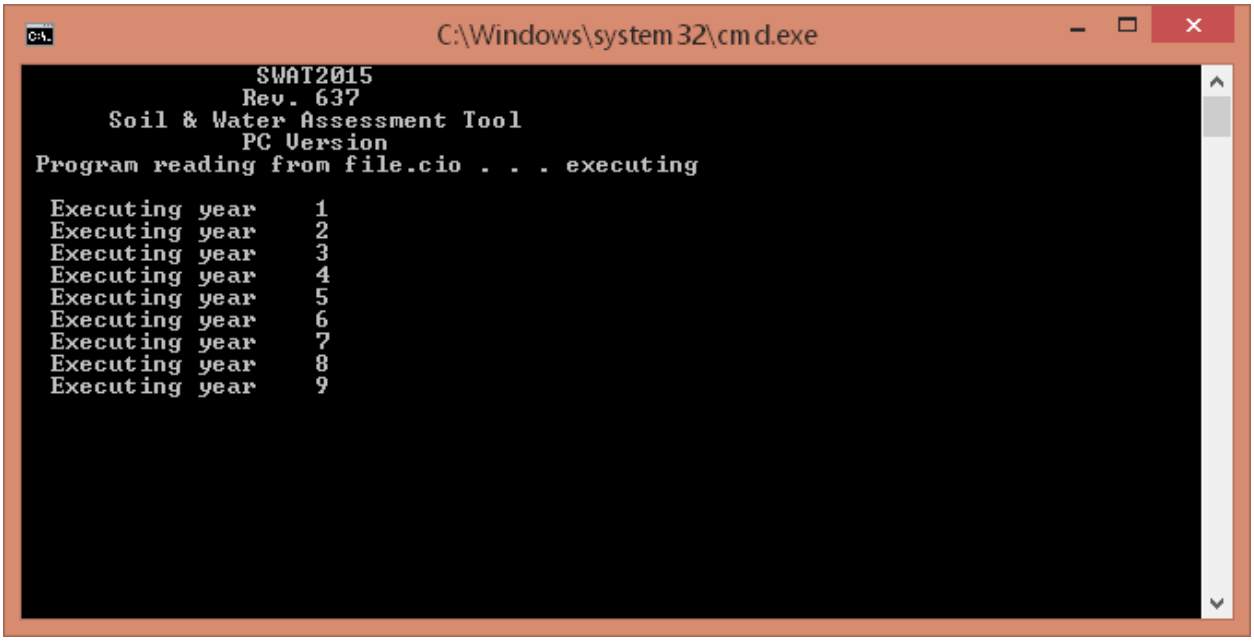


Figure 29: Running SWAT

43. If the run is successful the command window will close and a message box will say that SWAT was run successfully. Click **OK**.

44. If the run is not successful the command window will stay open with a message telling you to **Hit Enter to exit**. The release versions of the SWAT executable run faster than the debug versions, but don't provide information on errors, so if you have an error hit enter to close the command window and run SWAT again but with a debug version of SWAT.exe selected. The error message may suggest which SWAT input file needs to be checked, or you may be able to get more information from examining the SWAT source code, but probably you will need to report the problem to the SWAT team. Go to the SWAT support page http://swat.tamu.edu/support/ for advice. Click **Cancel** to close the SWAT simulation window.

45. If you want to use QSWAT visualisation you need to use **Read SWAT Output** in the **SWAT Simulation** menu. Select output.rch, output.sub, output.hru, output.sed, and (if you checked **Print Water Quality Output** in the **Run SWAT** form) output.wql, and click **Import Files to Database.**

46. The SWAT simulations can be saved using the **Save Simulation** button under **Save SWAT Simulation**. This copies C:*QSWAT_Projects\Robit\Demo\Scenarios\Default* to C:*QSWAT_Projects\Robit\Demo\Scenarios\Sim1* if the user chooses to save as *Sim1*. If *Sim1* already exists it will ask the user to confirm it should be overwritten. Click **Cancel** to close the SWAT Output form.

47. **Set Default Simulation** option under **SWAT Simulation** tab helps to restore the model parameters and its output back to a preferred previous SWAT simulation (i.e. not necessarily only to the default simulation).
48. **Manual Calibration helper** option under **SWAT Simulation** is used to calibrate SWAT parameters manually. It can also be used to implement calibrated model parameters for SWAT simulation when automatic model calibration is done outside the QSWAT environment (e.g. using SWAT-CUP).
49. Click **Exit** to close the SWAT Editor.

5.4. Output Visualisation with QSWAT

Once SWAT has been run and the output imported, so that there is an output database *SWATOutput.mdb* in the *Scenarios\Default\TablesOut* folder of the project, then a button *Visualise* becomes available in the main QSWAT form (Figure 30). Visualisation allows outputs from SWAT to be visualised graphically by subbasin, reach, or HRU.

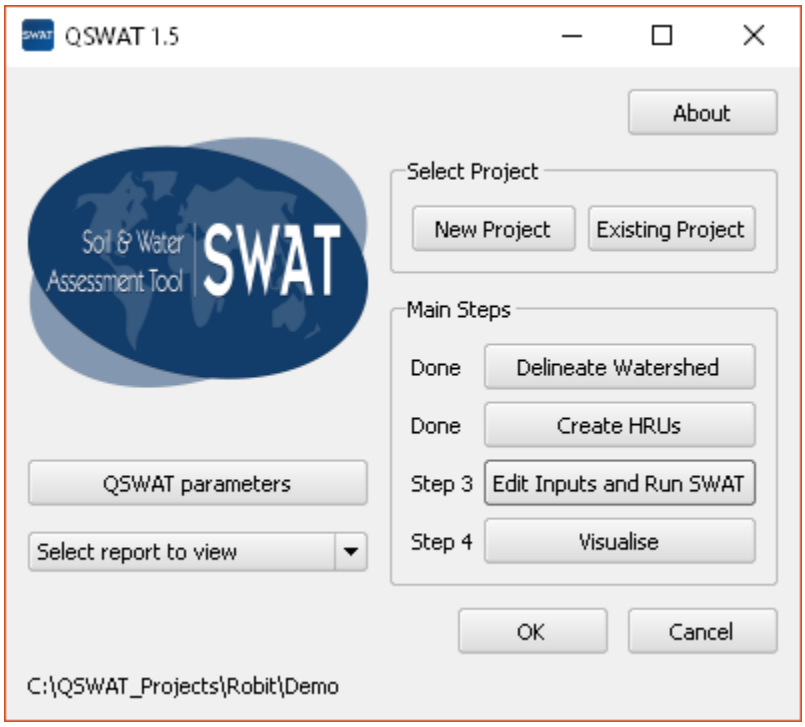


Figure 30: Main QSWAT form showing Visualise button

5.4.1. Preparing to visualise

In order to visualise results you have first, of course, to (successfully) run SWAT. Then you have to save your results in the output database. To do this, in the SWAT Editor, use **SWAT Simulation → Read SWAT Output** to bring up the **SWAT Output** form. Then check the boxes for the outputs you want to save in the database. QSWAT can display results from output.rch, output.sub, output.hru, output.sed, and output.wql (which only has data if the **Print Water Quality Output** was checked in the SWAT Editor **Run SWAT** form). Then click the **Import Files to Database** button to create (or update) the database. It is a good idea to use the **Save Simulation** button to save the simulation, having provided a suitable name. Then if you change anything and run SWAT again, the results from the saved run will still be available. Saving the previous simulation also saves copies of the watershed, streams and actual HRUs shapefiles, which are needed for visualisation, and which may be changed if you rerun watershed delineation or HRU creation. Click **Cancel** to exit from the **SWAT Output** form, and **Exit** from the SWAT Editor.

5.4.2. Running visualise

Click **Visualise** on the main QSWAT form to bring up the **Visualise Results** form (Figure 31). Here the **Choose scenario** pull-down menu shows the simulations, or scenarios, that are available: the current **Default** one, and in this example another called S**im1** that has been saved.

Select the one that you wish to display. Then use the **Choose SWAT output table** to see what tables are available: you will see all or some of *hru, rch, sed, sub,* and *wql* (the list depends on the output files you imported into the SWATOutput database). Select one of these.

5.4.3. 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.

5.4.4. 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 subbasin, reach, or HRU and displayed as a map. For animation there is an animated display of the value in each subbasin, reach, or HRU 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 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. subresults.shp) and stored in the **TablesOut** folder 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, rivs.shp, and subs.shp in this folder. Do not choose any of these as your file name: they are copies of the actual HRUs, stream reaches and watershed shapefiles generated by QSWAT, copied into this folder and used as a basis for the results shapefile (and for animation). This is done because these shapefiles may be changed by subsequent runs of QSWAT and not have the same HRUs, stream reaches or subbasins as this run had.

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.

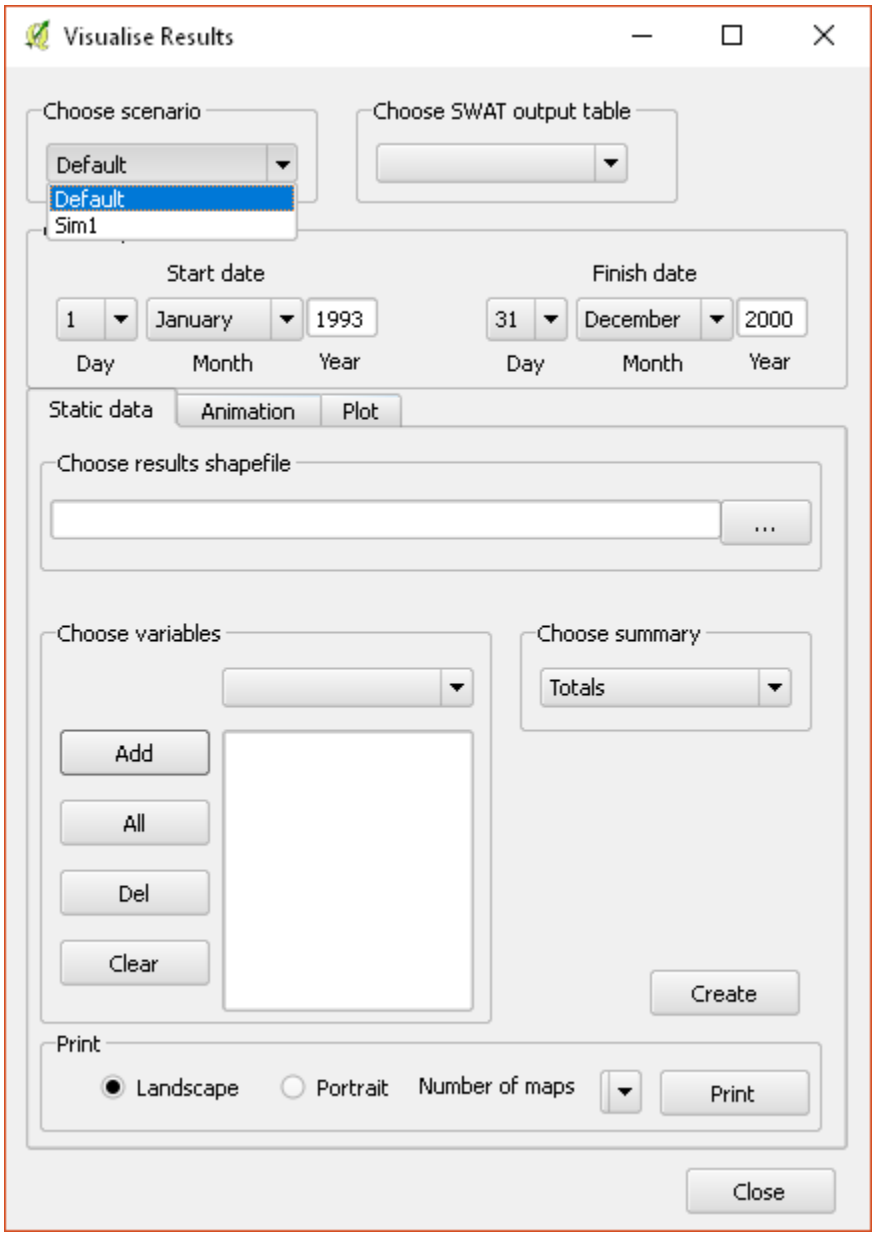


Figure 31: Visualise results form

For each variable, for each subbasin, the output data has a value for each day, month, or year of your simulation (according to what printout frequency you chose for the SWAT run). 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 subbasin, reach, 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 subbasin, reach, or HRU.
- **Minima**: the summary is the minimum value for the variable for the subbasin, reach, or HRU.

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.

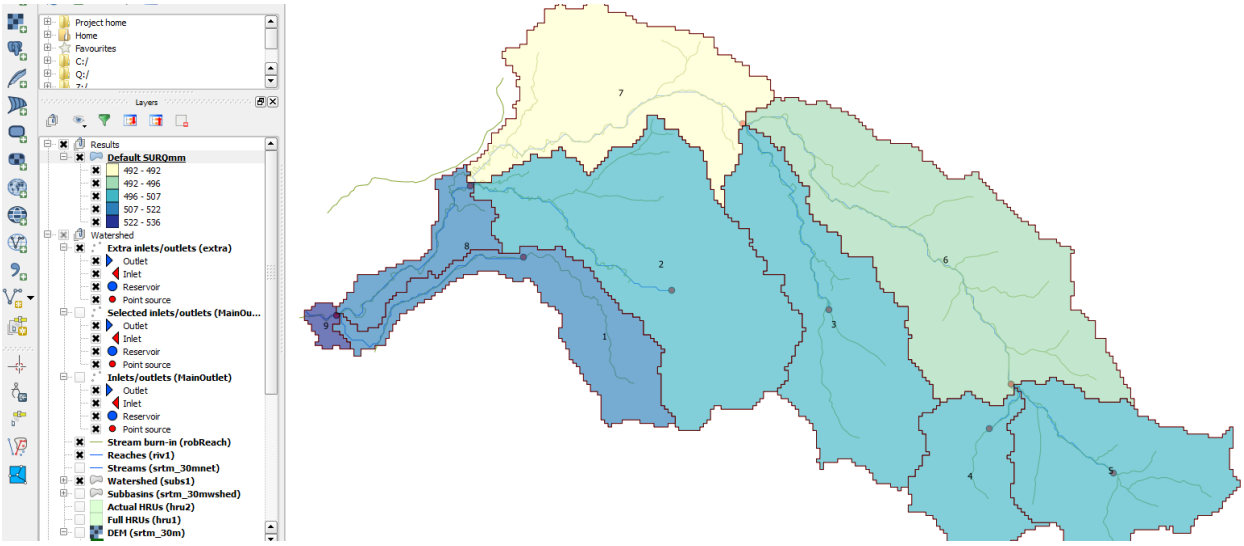


Figure 32: Displaying results

When you have loaded the variables you want included into the variables box, selected the one to display (SURQmm for this example), and chosen the summary (annual means), click the **Create** button. You may be asked to confirm if you are overwriting an existing shapefile. Then the results file will be displayed as shown in Figure 32. The legend shows that we are displaying **SURQmm** annual 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 **Style** 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**.

5.4.5 Printing

You may want to create a printout of your SWAT results, and QSWAT makes use of the QGIS Print Composer to help you do this. There are a number of predefined templates to create a composition of 1, 2, 3, 4 or 6 results maps laid out in either landscape or portrait mode.

The templates are not expected to provide final compositions. They are rather intended to provide a basic composition for you to edit using the Print Composer. Such editing might include adjusting the size and position of the map boxes, adjusting the scale of the maps, changing the text in the text boxes, adding new text boxes or images, removing existing elements.

As well as containing 1 to 6 maps, each with its legend, each of the templates provides

- a page title, initially containing "SWAT results: <project_name>"
- a North arrow
- a scale bar
- a text box containing "Author/Company"
- a date text box containing "Created on: <date>" where the date is in yyyy-mm-dd format

Figure 33 shows the result of creating a 3 map portrait print composition and immediately exporting it. In practice one would at the very least replace 'Company/Author' with something appropriate (or remove it) and perhaps lower the legends for the first and third maps to avoid the clash of their texts with the corresponding maps.

To create this composition we

- created the SURQmm map as described earlier
- switched to the Sim1 scenario, the sub table, and created the SWmm annual means map
- switched to the rch table and made the SEDCONCmg_kg annual means map. As it is based on the rch table it uses streams as the basis for the display. Line widths correspond to stream widths, and colours to variable values. To show this map clearly in the canvas you need to remove the visibility of other results maps, but (as explained below) this will not affect the print composition.
- selected *Portrait* and the number 3 in the *Print* group in the visualisation form's static tab, and clicked **Print.**

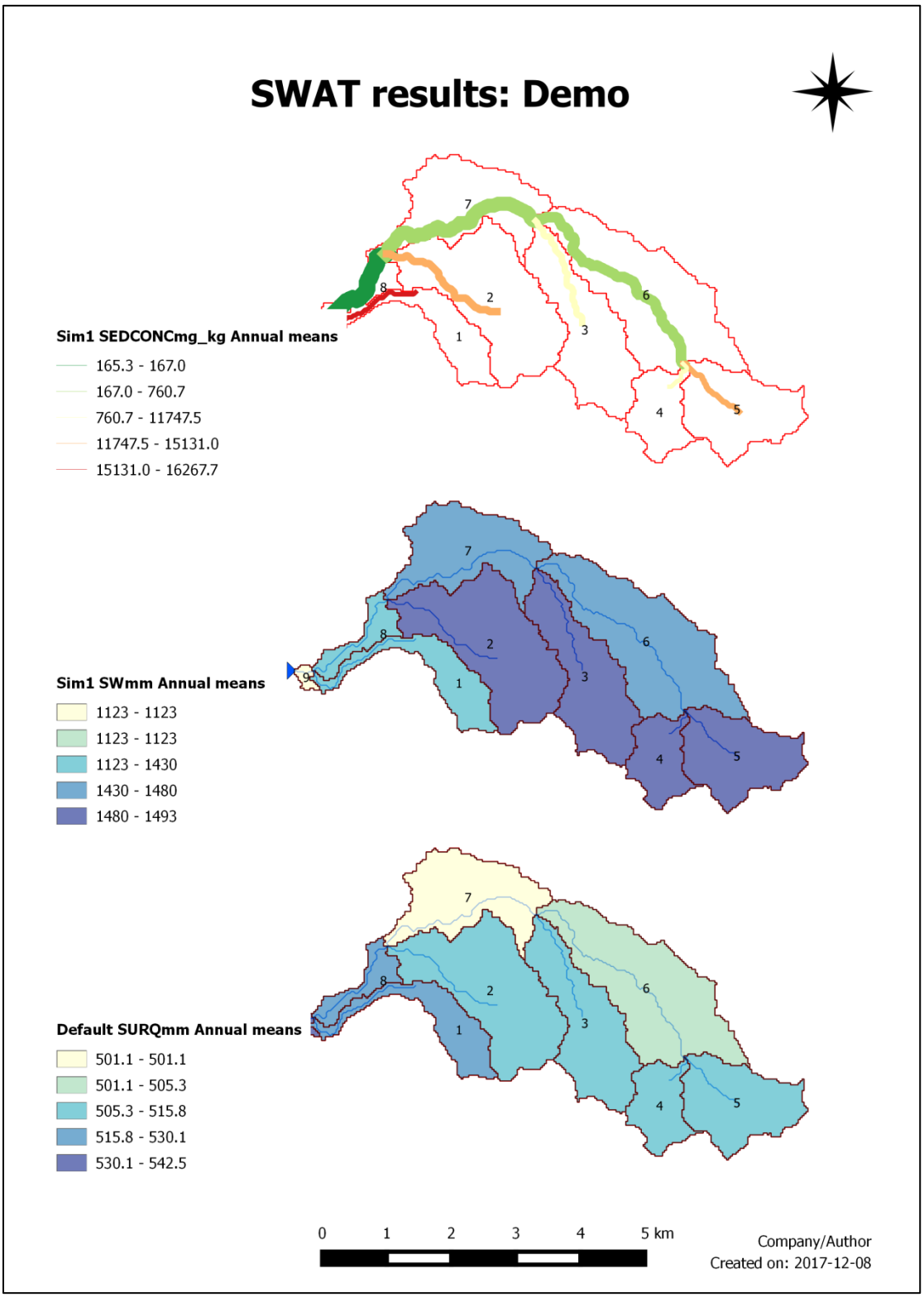


Figure 33: Print composition

It is important to note that if two maps use the same scenario and table, they would by default use the same results file and the second would replace the first as a results layer in the Layers panel. To avoid this, if you want both in the print composition, you have to change the name of the results shapefile for the second one. This was not necessary here since for the second map we used the Sim1 scenario.

The rules use for creating print compositions of *n* maps are as follows:

- The first *n* maps in the *Results* group in the layers panel will be used. They are taken one at a time, whether or not they are currently marked as visible. They are displayed in the order in which they occur in the *Results* group, stepping from top left to bottom right, and across before down, in the composition.
- Layers in the *Watershed* group are included in the background if they are currently marked as visible. It is normal to have the **Watershed** layer visible, so you can see the subbasin boundaries, and common to have the **Reaches** layer also visible. (Results layers have their transparency by default set at 35% to enable these watershed layers to be visible as well.)
- The print composer is created with the map layers locked (otherwise they would automatically change to whatever the map canvas currently contains) but with all the composition items unlocked, so they can be moved, resized, have their text changed, etc.)
- The scale bar is attached to the first (top left) map. If you change that map's scale the scale bar will change automatically, but will not reflect the scales of the others unless you change them similarly.

To avoid the maps in the print composition being smaller than necessary, before you click **Print**, select the FullHRUs entry in the Layers panel, (doesn't matter if it is visible or not) and click the 'Zoom to layer' button at the top. Reshape your map canvas to make your watershed fill it as much as possible. If you don't have a FullHRUs shapefile, an alternative technique is to set the *Scale* using the drop down menu to a value fairly close to the current one, such as 1:50,000. Then adjust the window size and shape, aiming to make the boundary of the map canvas as close as possible to your watershed boundary. Annoyingly, this tends to change the scale, but because you chose one in the menu it is easy to reset, and you should with a few adjustments achieve a good fit.

This document cannot describe all of the features of the QGIS Print Composer. There are documents (e.g. https://docs.qgis.org/2.8/en/docs/user_manual/print_composer/print_composer.html) and tutorials (e.g. www.qgistutorials.com/en/docs/making_a_map.html) on the web.

But here are a few hints for some common tasks (all done in the print composer):

- The first lines of legends are often longer than you want. To change their font, select the legend in the main window, then open **Fonts** (on the right), and the relevant item is 'Subgroup font'. To edit the text, select it in the panel under **Legend items**, then use the

button for text editing (pencil and paper symbol). To reduce the spacing between legend items open **Spacing** and change **Subgroup** space (to 1mm perhaps)

- To change the size of maps, select the map in the main window and then edit the Scale entry under **Main properties**. The scale is the ratio of actual distances to map distances, so you reduce it to make the map larger. Use the same number for all the maps and the scale bar will stay correct (it changes automatically with the first map).
- It is possible to use HTML in labels. The tutorial above has an example.
- Use the Items panel (top right) to lock what you have fixed, to avoid accidentally changing it again.

5.4.5.1 Printing other maps

It may be that you would like to include in a print composition other maps, such as the landuses or soils. All you need to do is include such a layer in the *Results* group.

But the map you have may well extend outside the watershed. We illustrate how to deal with this by adding a landuse map to our print composition.

The first task is to clip the landuse map to the watershed boundary. First we will need to know the nodata value for this map - it is always a good idea when using QGIS tools for transforming maps to know and use the nodata value. Right click on the **Landuses** entry in the layers panel, select *Properties*, and under *Layer info* in the *General* tab you will see the nodata value is 255. (This is unusual - it is usually negative.)

Select (left click) another layer in the *Results* group (which will place the new map above that layer), Then use *Raster → Extraction → Clipper* to bring up the *Cliipper* form. Find the *Landuses* layer in the drop-down menu for the *Input file (raster)*, browse to (for example) the *Source\crop* folder and use a name such as *landclip* for the *Output file*, check *No data value* and set it to 255, select *Mask layer* as the *Clipping mode*, and select the *Watershed (subs1)* layer to clip with. Make sure *Load into canvas when finished* is checked (Figure 34). Click *OK*.

It is not unknown for clipping with the watershed shapefile to fail with a message that there is a fault in the shapefile. An alternative is to use the Saga raster calculator, which seems to be reliable for this purpose. First load the raster *Source\srtm_demw.tif*, which is the raster version of the subbasins map. Then use *Processing → Toolbox* to see the available tools, find *SAGA→ Grid - Calculus → Raster calculator.* In the raster calculator form choose the **Landuses** layer as the main input layer, and use the *Additional layers* button to open the *Multiple selection* form. Check *srtm_30mw* and click OK. Enter the formula *gt(b, (0-1)) * a*, click the *Result* button, choose *Save to file*, and browse to, say, *Source\crop* and use *landclip* as the file name. Click *Save*, and then *Run* the calculator. (The idea behind the formula is as follows: *a* is the landuse file we want to clip. *b* is the subbasins raster *b* which has valid values from 0 up within the watershed boundary and nodata outside it. The *gt* expression will return 1, so that *a* values are

preserved, where *b* is valid, Outside the watershed boundary, *b* gives nodata, and the calculator there gives the result *a* nodata.)

The result (from clipping or from using the raster calculator) will not be very satisfactory. The name will be **landclip** (if made with the clipper) or **Result** (if made with the raster calculator) Right click on the layer, and use *Rename* to change it to, for example, **Landuse**.

The legend will only include numeric map values and the default colouring will be black and white or grey scale. You can fix this by copying the style of the original **Landuses** layer. Right click on that layer, select *Copy Style*, right click on the **Landuse** layer and select *Paste Style*.

The colours used for the original landuses map were chosen randomly, and you may want to change them. Right click on the **Landuse** layer and select *Properties* and then the *Style* tab. Double right click on entries in the colour map to make a colour selection tool available. Choose another colour, and click *Apply* to see the result in the map canvas. You may also wish to set the layer's transparency so that the subbasin boundaries and streams are visible: use the *Transparency* tab in the *Properties* form, and set *Global transparency* to, say, 20%. Click Apply to see the result. Click *OK* to finish.

If necessary, drag the **Landuse** layer to the position you want in the *Results* group

Figure 35 shows a 4-map landscape print composition with the landuse map.

It is finally worth mentioning that it can be that parts of results shapefiles can lose all colour, becoming white and transparent, when they are at the upper or lower limit of the range of values. This sometimes happens when the project is restarted. If this happens you can fix it by using *Properties* and then the *Style* tab to show the colouring scheme of the affected layer, click *Classify* to recalculate the bands, and then click *Apply* and *OK*.

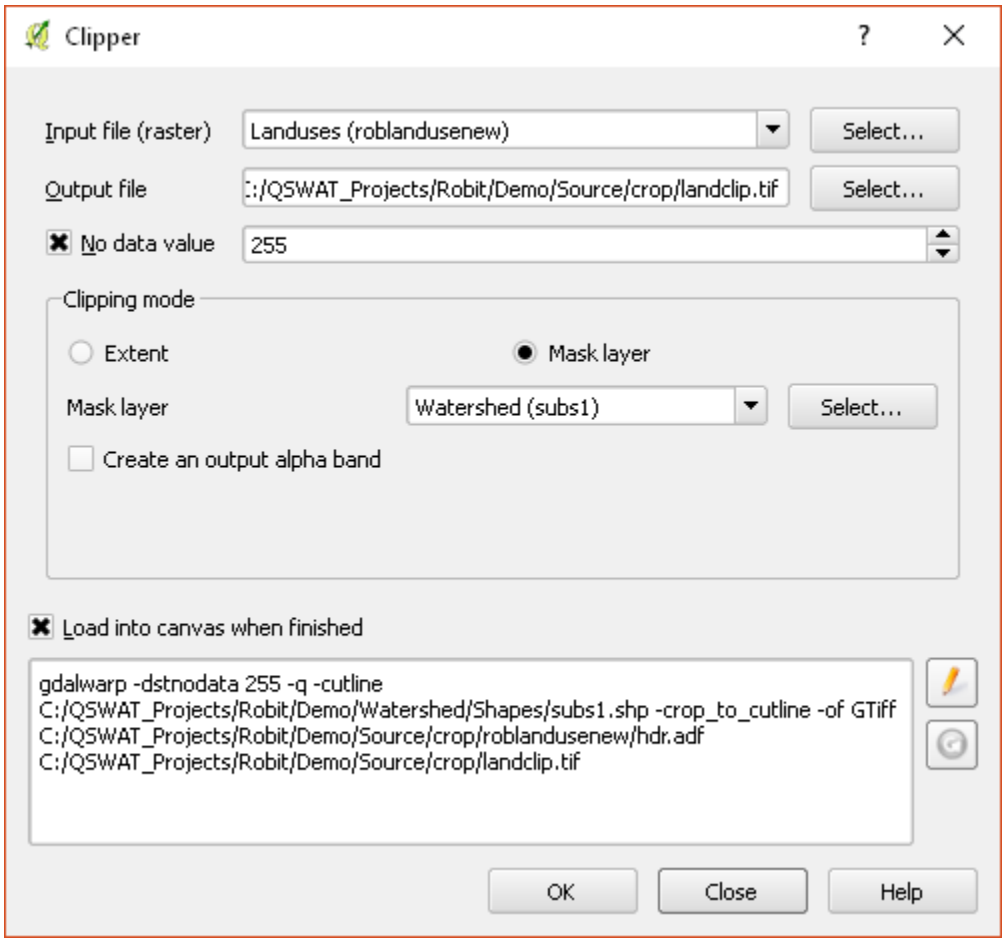


Figure 34: Clipping landuse map

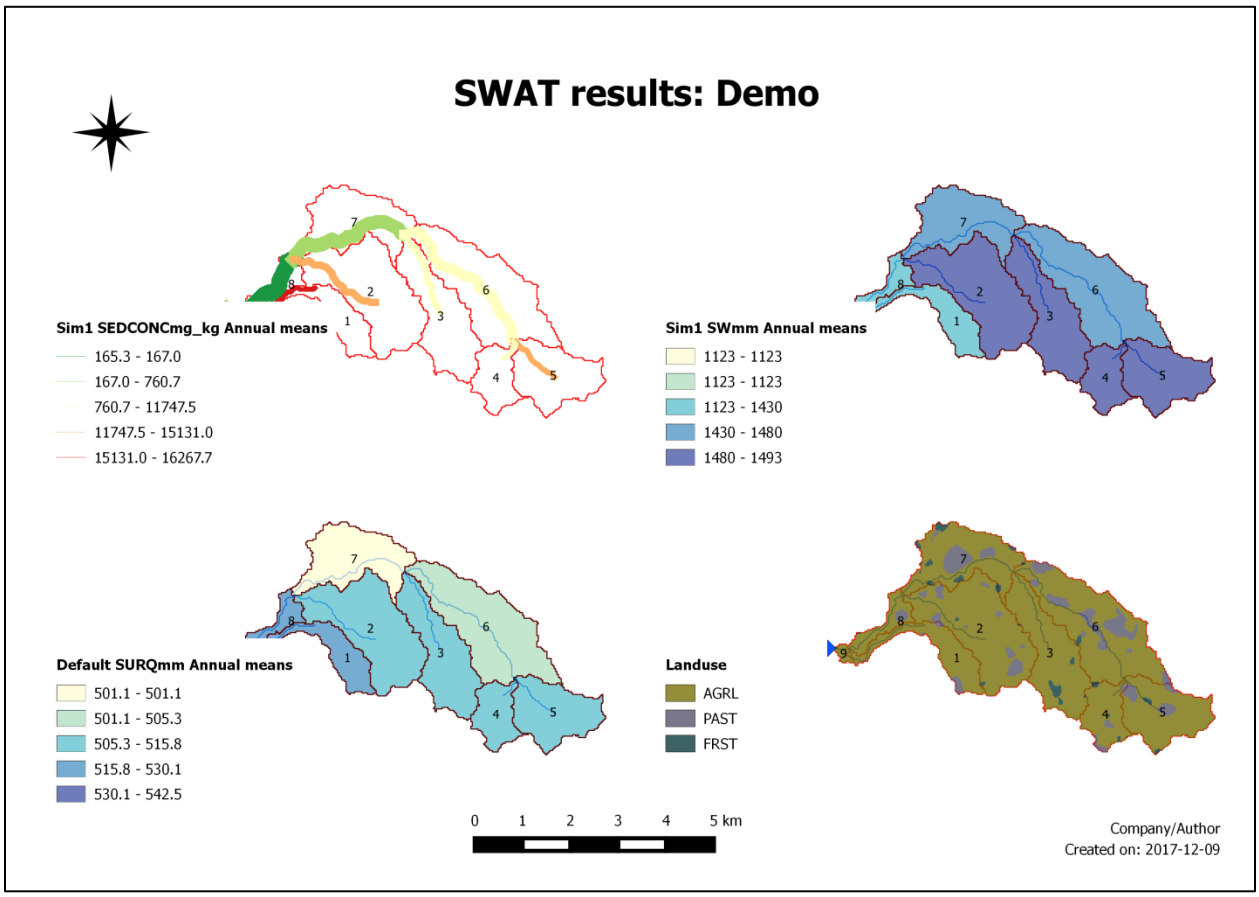


Figure 35: Print composition with landuse map

5.4.5.2 Multiple maps

There is a convenient method for viewing multiple results (or other maps) at the same time, to enable visual comparison. This is based on the optional plugin *Dockable MirrorMap*. First install this plugin using the *Plugins* → *Manage and Install Plugins* menu. This adds a button to the toolbar, and each time you click it a new mirror window appears. The '+' button for each such window allows one or more layers to be added (whether or not that layer is currently marked as visible). Select a layer in the *Layers* panel and then click the '+' button. You can add a label to the window by clicking on the spanner button at its lower right. Figure 36 shows the result of adding 3 mirror maps and adding our results layers. For the SEDCONC map (upper right) we also added the **Watershed** layer to provide the subbasin boundaries.

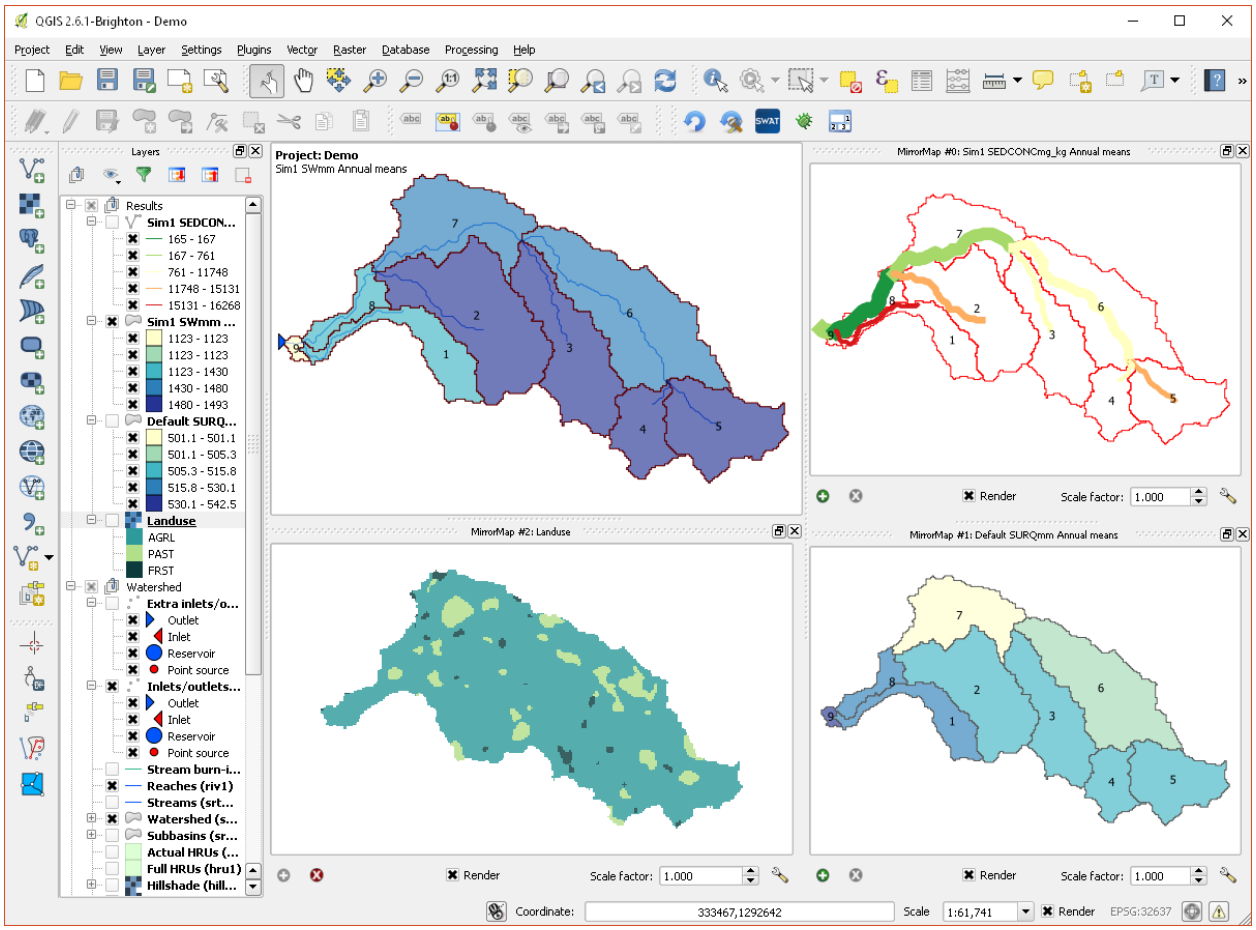


Figure 36: Using dockable mirror map plugin

5.4.6 Animation

To animate, select the **Animation** tab (Figure 37).

There are two modes of animation: using the map canvas and using a print composer. Map canvas animation shows the animation for one output variable in the map canvas. Print composer animation can show (by creating a video) the animation of 1, 2, 3, 4 or 6 output variables in synchrony.

5.4.6.1 Map canvas animation

Make sure you have the scenario and output table you want, and that the dates are for the period you want to animate. You probably do not want, for example, to animate at a monthly time step for more than a year or two.

Initially, select **New** in the *Set up animation* group. The **Variable** pull-down menu contains all the variables from the chosen output table: select one of these. A new animation layer is created in the map canvas, showing the initial values of this variable in the simulation (Figure 38). Here we are animating the PETmm variable from the *sub* table in the *Sim1* simulation. Note that the range of possible values runs from 89 to 171 (the minimum and maximum values for PETmm for the whole simulation period). You can view the actual initial values for each subbasin by examining the attribute table (right click on the **Sim1 PETmm** entry in the **Animations** group 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 reporting period chosen for the simulation. 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 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 the *Animation* folder 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). Hover the mouse over the entry in the **Layers** panel to see its name.

It is also possible to make a video of the animation. To do this, at the position when you want to start the recording, click the **Start recording** button. This will turn red, to show recording is in progress. Then run, or resume running, the application. Wait until this finishes, or pause it if you wish, and click **Stop recording**. This will create an animated gif, called *<variable>Video.gif* (in this case *PETmmVideo.gif*) in the *TablesOut* folder of the scenario. You can watch the video by clicking the **Play recording** button.

If you get a memory error, with a message suggesting you try GIMP, see the instructions on page 55.

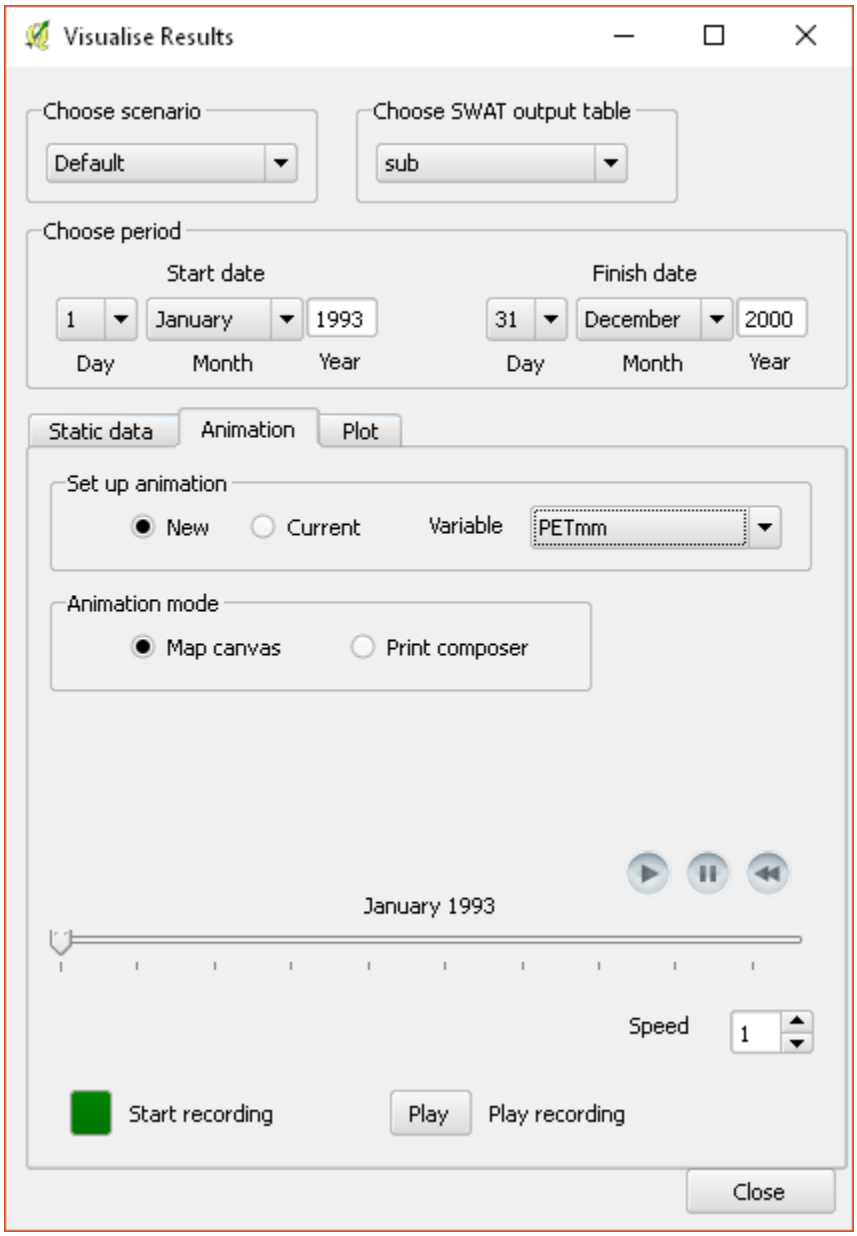


Figure 37: Map canvas animation

The animated gif is made from a collection of *.png* images created in a folder *Gif* in the default scenario's *TablesOut* folder. The video is created, and these image files deleted, by stopping recording, so if you want to do anything with these files you need to do so after the animation (for example when it reaches the end or when you pause it) but before you click **Stop recording.**

The speed of the video is determined by the **Speed** spinbox setting when the video is created: it will not reflect any changes in speed made while it was recording. It may play back faster than it

was recorded, so you may need to recreate it with an adjusted speed setting if it runs too fast.

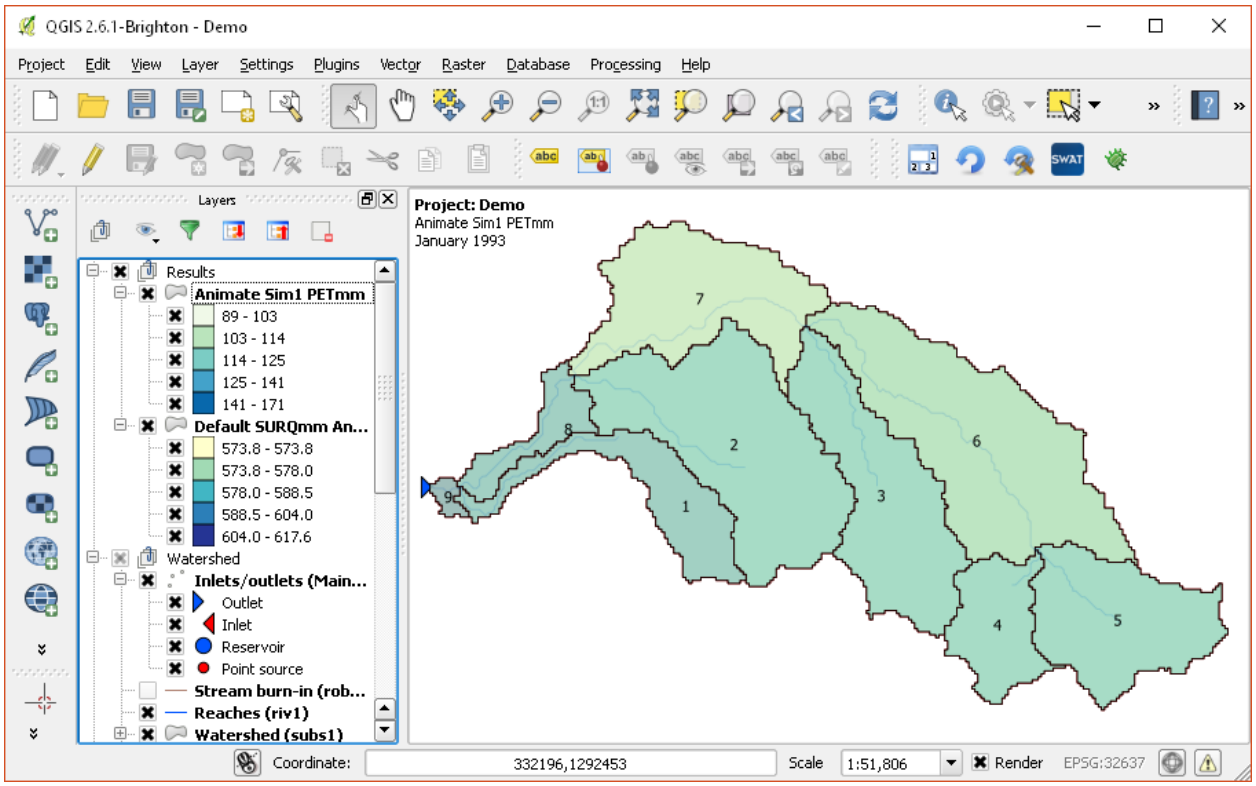


Figure 38: Start of animation

When you click the **Play** button your computer's default application for displaying gif files is used

Note that the animation video file name starts with the name of the variable being animated. So you may safely make a number of animations of different variables. But if, for example, you wanted to animate the same variable over different time periods you would need to copy or rename the first animation file before making the second.

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**.

5.4.6.2 Print composer animation

Print composer animation adds two possibilities to map canvas animation:

- The animation is based on the print composer, and so adds more possibilities like extra labels, scale bar, north arrow, etc.

- It allows the synchronous animation of 1, 2, 3, 4, or 6 variables.

It consequently needs a little more setting up. On the other hand, there is no equivalent to the slider or left/right keyboard arrows for running the animation after its creation.

In the previous section we used the **New** choice in the *Set up animation* box. If we keep the **New** setting and select another variable (possibly after changing the scenario and/or table) we will get another animation layer. Figure 39 shows the situation after we changed the table to *rch* and selected SEDCONCmg_kg as the variable. If we had changed the choice to **Current** before selecting the variable, the SEDCONCmg_kg layer would have replaced the PETmm one. The current animation layer is the currently selected (or only) layer in the *Animations* group. (If there is no such current layer it is as if **New** were selected.)

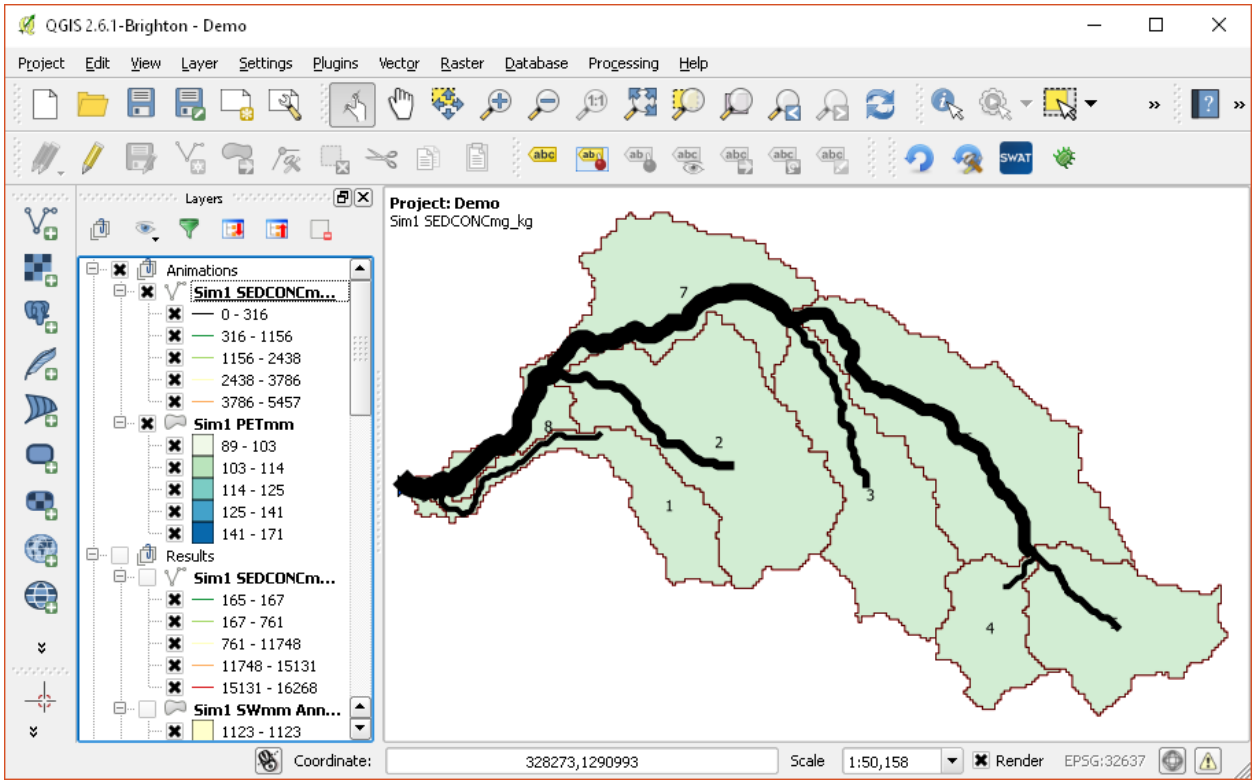


Figure 39: Second animation layer added

We could now run the animation in the map canvas, but it will not be very satisfactory as one layer lies over the other. It would be even less satisfactory if both layers were based on subbasins, or on streams, or if there were more layers. So instead of choosing **Map canvas** as the *Animation mode*, we select **Print composer** (Figure 40).

This brings up a choice between **Landscape** and **Portrait** layouts, and selects 2 as the number of maps, since we currently have two animation layers. We can add more such layers if we wish, and increase this number, or reduce it to 1. If the count is set to *n*, then the top *n* layers in the

Animations group will be used (regardless of their current visibility). In addition, as with static print compositions, visible layers in the *Watershed* group are included.

Figure 40: Setting up a print composer animation

Change the layout to **Portrait,** move the slider by hand to near the end of the period, say to June 2000 (to reduce the number of images we will be creating) and click **Start recording**. An information form is shown explaining that the print composer is about the start and that you can

modify it as you wish, but then must save it as a template file, which must be called *AnimateTemplate.qpt* and placed in the *TablesOut* folder of the *Default* scenario. Click *OK*. The print composer appears (Figure 41).

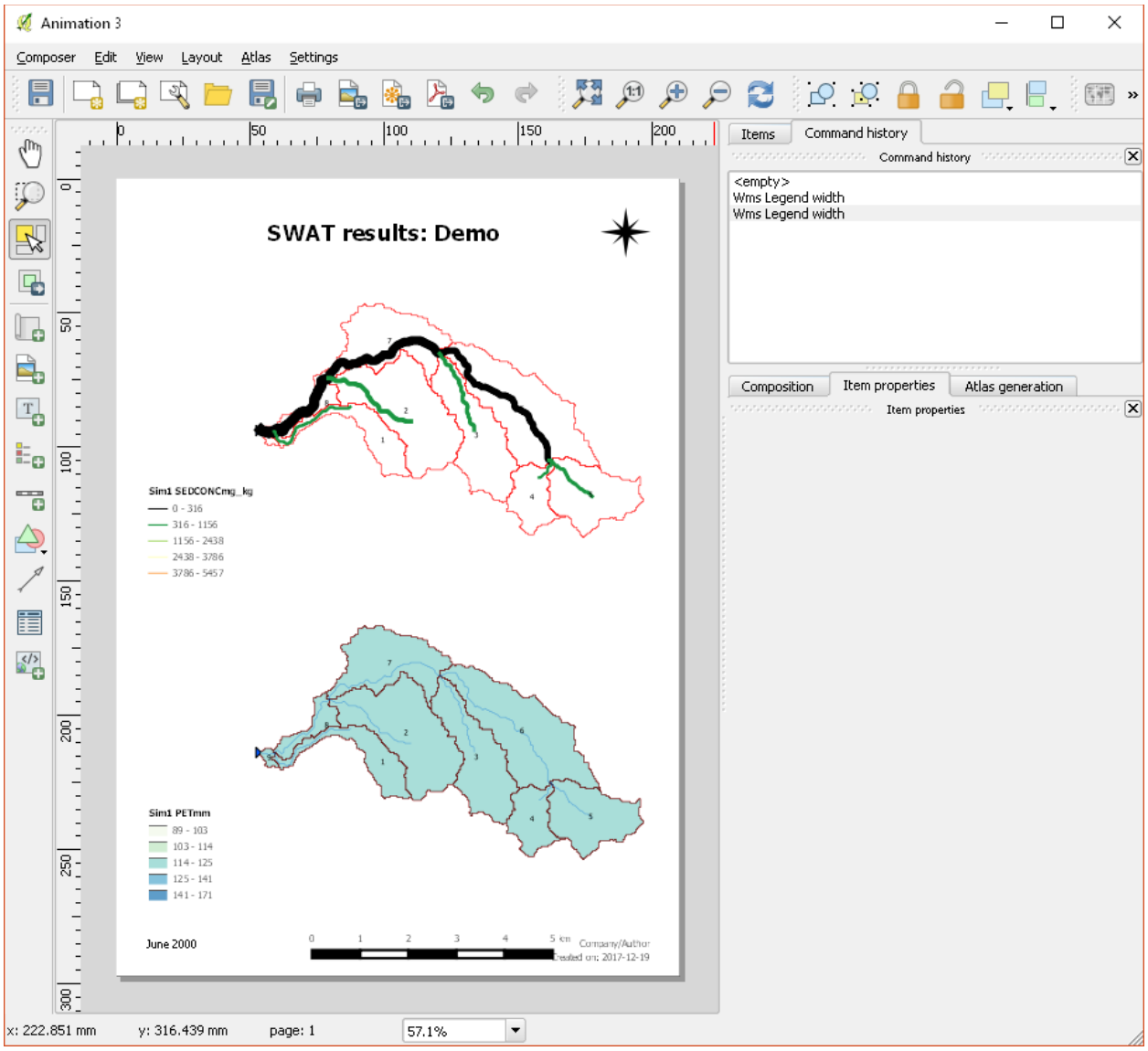


Figure 41: Animation print composer

You can now make what changes you would like: changing the positions or scales of the maps (keep the same scale for both!), removing or adding labels, changing the 'Company/Author' text, and so on. When you are ready, use the *Composer → Save as Template* menu item to save as *C:\QSWAT_Projects\Robit\Demo\Scenarios\Default\TablesOut\AnimateTemplate.qpt* as previously instructed. You will have to confirm saving as the file already exists - it was used to set up the composition you have just edited. *Quit* the print composer. (If you made no changes you can quit from the composer without saving the template.)

The form shows that we are recording, since the recording button is red, so we run the animation by clicking the run button (the grey right-pointing arrow). When it reaches the end, click **Stop recording**. At each time step the print composer is run to create another image, and you will see its window flash on and off. Click **Play** to see the recording. The recording is called *Videon.gif* where *n* is an integer used to generate a new file name, and is stored in the *TablesOut* folder of the *Default* scenario. An information message gives the file name.

5.4.6.3 Gif creation failure

Creating the animated gif from the *.png* files seems to consume a lot of memory and can fail. If this happens, you will get an error message reminding you where the *.png* files are (in *C:\QSWAT_Projects\Robit\Demo\Scenarios\Default\TablesOut\Animation\Png*) and suggesting you use GIMP. You may of course use any program (or internet service) you wish, but GIMP is a well respected, open source and free program that can perform a wide range of image manipulations, and animated gif creation is very easy with it. You can download it from https://www.gimp.org/downloads/.

To create the animated gif, start GIMP. Use *File → Open as Layers* and navigate to the *Png* folder, where you will see files *still00001.png, still00002.png,* etc. Click on the first, and Shift-click on the last so all are selected, and click *Open*. The files are loaded as layers into a single image. Make sure the layer you can see is the last one (see the date at upper or lower left). If it is the first, reverse the order: use *Layer → Stack → Reverse Layer Order* to do this. Now we just save as an animated gif. Use *File → Export As* to open the *Export Image* form. Select a suitable folder and file name - make sure it ends in *.gif* - and click *Save*. A new *Export Image as GIF form* starts. Set the choices as shown in Figure 42. The points to note are

- Check *As animation*
- Uncheck *Loop forever*
- Set the delay to something bigger than the default 100 milliseconds: We have set it to 2000 (i.e. 2 seconds).

Click *Export*. Wait for it to finish: the current form closes and a new one appears.

When you use *File → Quit* to quit from GIMP, you may get a message that there are unsaved changes. This seems to be an error: *Discard Changes* can be clicked. You can then use an image display tool or a web browser to view the animation.

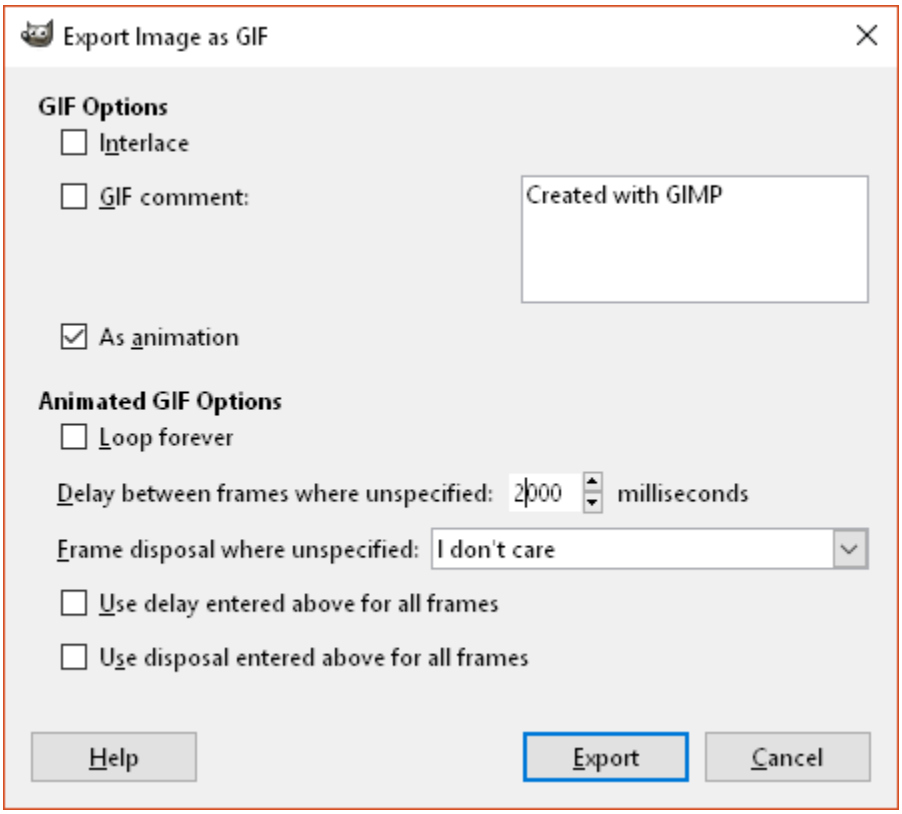


Figure 42: Creating animated gif using GIMP

5.4.7. 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 subbasins, reaches, 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.

5.4.7.1. 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 (Figure 43). 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 clicking **Add plot**. This

adds a row to the table, and already shows the chosen scenario and table. The **Sub** (subbasin) 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, subbasin, HRU 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.

Whether the HRU setting is available or is set to '-' depends on the table we are using. If it is *hru* (and we chose a multiple HRU option for the scenario) then we can choose an HRU from the current subbasin. Otherwise HRU is not used and is set to '-'.

Suppose we want to compare the main outlet flows from two scenarios, and for both the main outlet subbasin is number 9. Then *rch* is the appropriate table, and we select this and select 9 as the **Subbasin**. Finally we select *FLOW_OUTcms* 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 (almost) what we want. The subbasin number is cleared in the new row, to remind us that in a different scenario subbasin numbers might be different. We select the appropriate number, in this case 9 again.

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, Sub,* and *HRU,* and we can select a variable name from the file.

If you use the observed data file *observedFlow.csv* from *ExampleDataset* and click **Plot** you will get an error *Missing data for date 1998/1 for plot 2* (assuming the observed plot is second in your table). The reason is that this file only contains data from the start of 1993 to the end of 1997. Change the year of **Finish date** to 1997 and click **Plot** again.

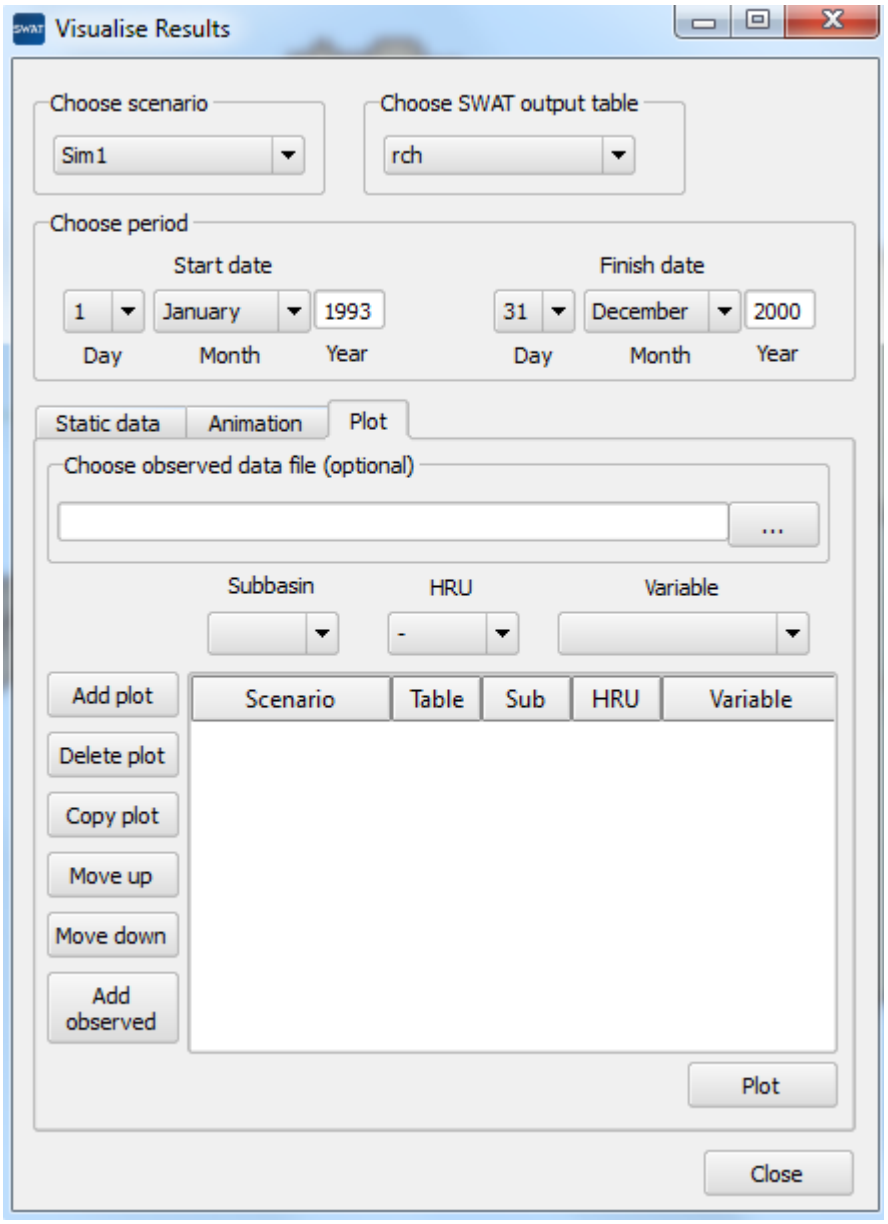


Figure 43: Plot screen

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.

Once the data is written to the *.csv* file, SWATGraph is automatically started to display the data graphically: see Figure 44. 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 44 shows a plot

of the flow out of subbasin 9 (the main outlet) against the observed data file *observedFlow.csv* supplied with the Robit example. Note that this file only has data from the beginning of 1993 to the end of 1997, so the finish date needs to be adjusted to 31 December 1997 before clicking **Plot.**

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 but one) and a number of editing options (rightmost button, the green tick). 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. Editing the graph disrupts the date markers on the horizontal axis: click the **Update** button to restore them.

The data from the *.csv* file is also displayed. The column headings, also shown in the graph legend, take the form **Scenario-Table-Number-Variable**, where **Number** is the subbasin number if the table is reach or subbasin, and the hru number if the table is hru. 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 *C:\SWAT\SWATEditor\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.

5.4.7.2. How compatible do different scenarios need to be?

Obviously, they need to use the same time interval (monthly, daily or yearly). 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 reach in each scenario, which gives figures for the whole watershed, it doesn't matter how the subbasins and hrus in each were selected, or how

the subbasins are numbered, as long as you select the main outlet subbasin number for each.

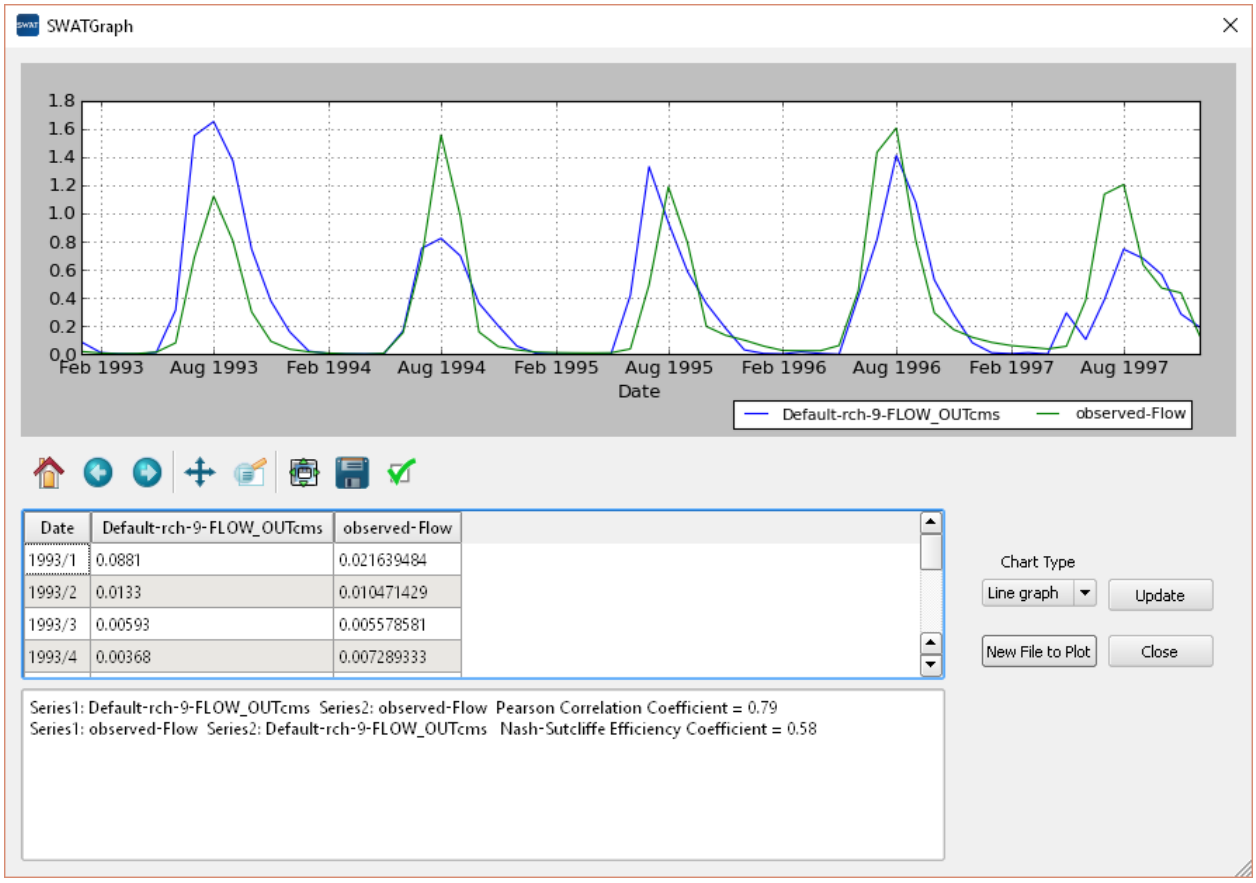


Figure 44: SWATGraph

5.4.7.3. 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, Subbasin and HRU 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.

5.5. SWAT Output Visualization with SWAT Error Checker

SWAT-Check is a tool that is used to summarize results from SWAT simulations. It can also provide warnings that can help to improve the modeling skills in SWAT. Currently, the program is integrated with the SWAT model and it can be found under the **SWAT Simulations** tab, and under **Read SWAT Output** drop-down list. It can also be installed separately. It can be downloaded from the SWAT website: http://swat.tamu.edu/software/swat-check/.

SWAT-Check evaluates different processes in the landscape. The following section describes the purposes of some of the tabs in the SWAT-Check program. Each tab has also brief note on the subject it is dealing with.

1) SWAT-Check evaluation can be executed by locating the *TxtInOut* folder of a particular simulation (Figure 45). Clicking **Run SWATCheck** button from the SWAT Output form directs to the *TxtInOut* folder in the *Default* simulation folder (the Default simulation always saves results from the most recent SWAT run). However, different simulations can be accessed by clicking the folder button in the right of the **Project Location** (Figure 45). SWAT-Check only reads files in the *TxtInOut* folder. Therefore, it is important to select the *TxtInOut* folder of a particular simulation that one is interested in (e.g. *C:\QSWAT_Projects\Robit\Demo\Scenarios\Sim1\TxtInOut*). The analysis will be executed by clicking the **Examine Model Output** button. Besides executing SWAT-Check, the **Setup** window provides a summary of the simulation details and watershed characteristics. Moreover, the **Setup** window provides messages and warnings that need to be addressed to improve the model simulation.

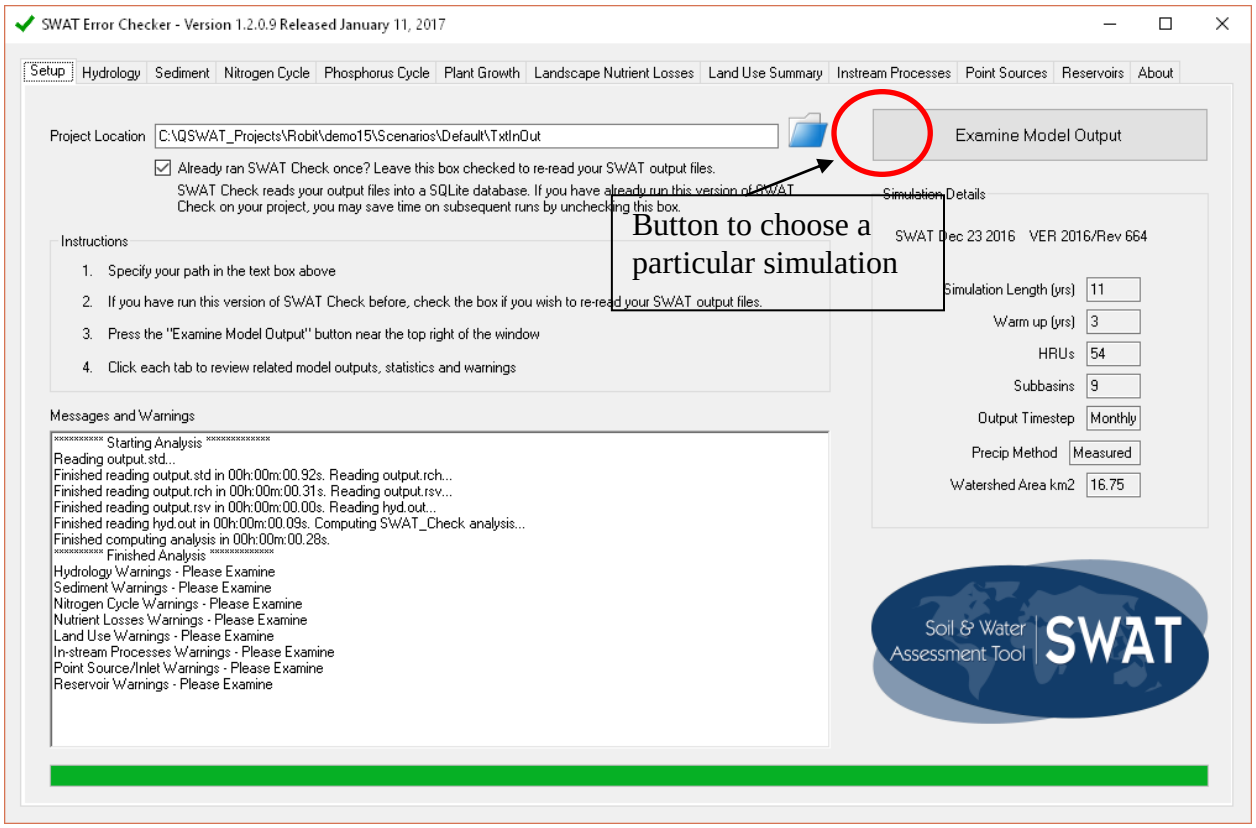


Figure 45: SWAT-Check Setup window showing how to run SWAT-Check for a particular simulation..

2) The **Hydrology** window in SWAT-Check summarizes the water balance both graphically and numerically. It can be accessed by clicking the **Hydrology** tab at the top of the SWAT-Check window. As hydrology is the basis for the landscape processes, particular attention should be given to modeling the hydrology correctly. Therefore, SWAT-Check will be a vital tool in providing a quick summary of the hydrological components and as such aids easy understanding of the system. For example, it provides the ratios of different water balance components. Users can, therefore, compare these values with published literature values, or measured estimates, and have an idea how to improve the model performance in subsequent model calibration processes. Moreover, it gives warnings and suggests methods to address them (Figure 46).

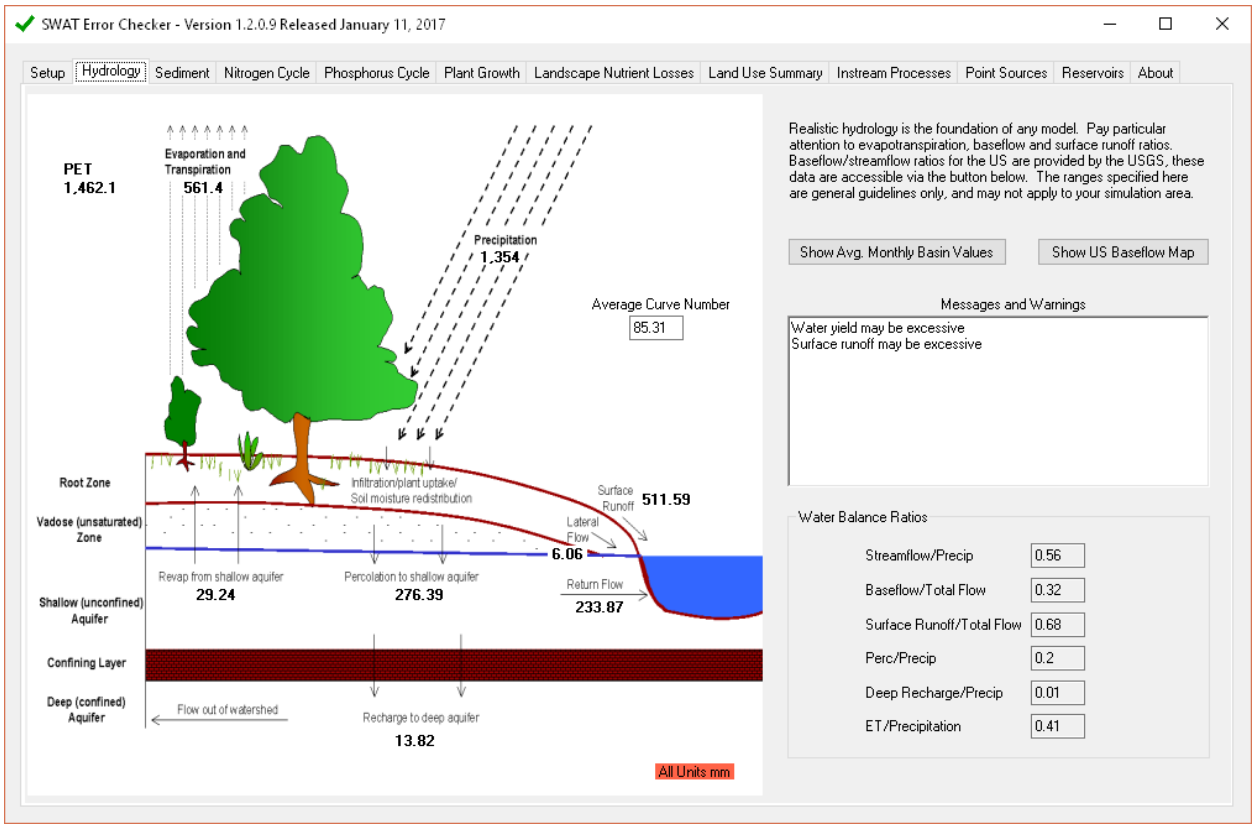


Figure 46: Graphical and numerical summary of hydrology related SWAT results from SWAT-Check.

3) The **Sediment** window summarizes graphically and numerically the sediment yield in the watershed – both in the upland and in-stream processes (Figure 47). It can be accessed by clicking the **Sediment** tab at the top of the SWAT-Check window. Sediment loss from the landscape is affected by many factors. Therefore, summaries of sediment related results from SWAT-Check can provide insights on how to improve the model performance in simulating sediment loss in the watershed. For example a quick overview of sediment overestimation could hint to inadequate biomass production, which could be further examined in the **land use summary** tab (Figure 48). Moreover, the **Sediment** tab provides warnings when results are outside ranges available in the literature. However, results are watershed specific and one needs to use these warnings as a basis for further re-examination by taking the actual landscape processes in the particular study watershed into context.

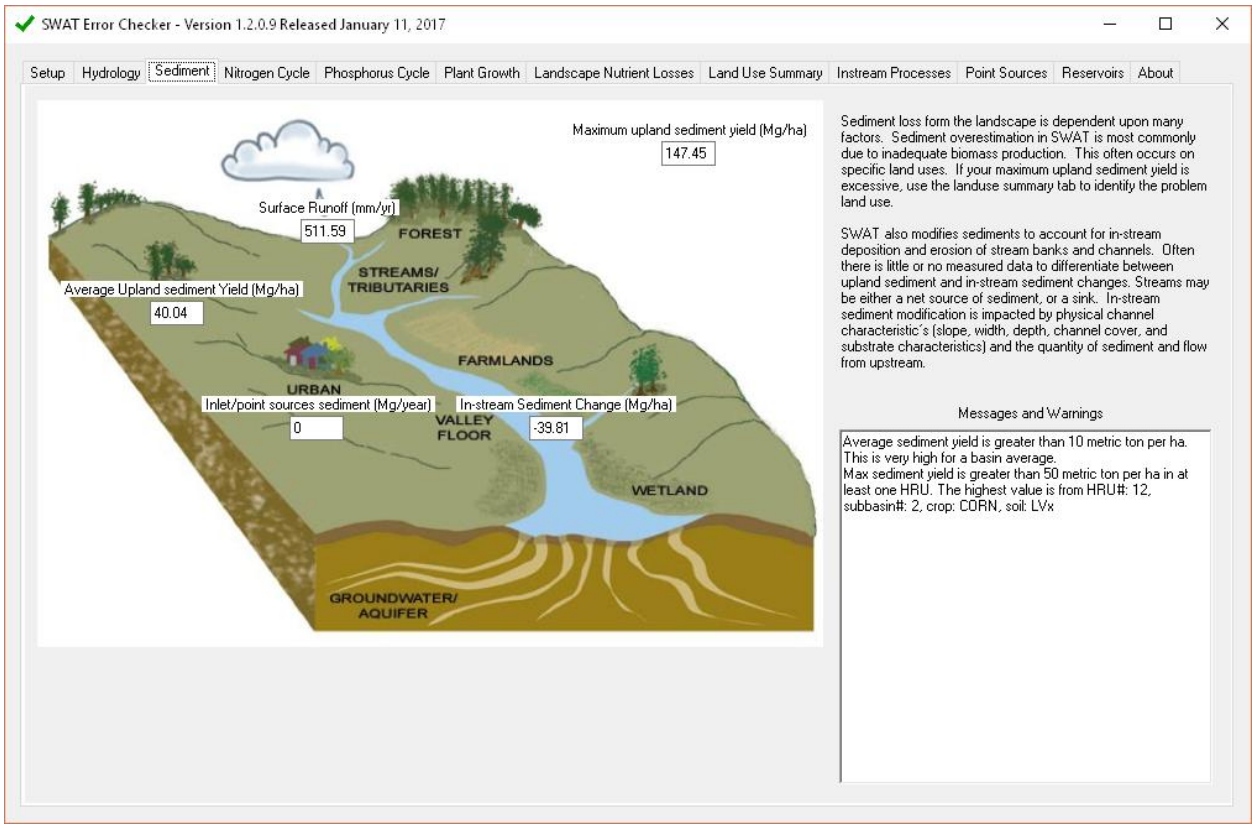


Figure 47: SWAT-Check sediment tab that summarizes sediment loss for different landscape processes

4) The **Land Use Summary window** summarizes the different landscape process components for each land use type in the watershed. These include hydrology, water quality, sediment transport and crop growth. The **Land Use Summary** report can be accessed by clicking the **Land Use Summary** tab at the top of the SWAT-Check window. In the case of model calibration, users focus on the watershed outlet, and they may overlook processes taking place in minor land use types. In such a case, the **Land Use Summary** report is very important in showing where things get wrong in the process of SWAT modeling as it summarizes results spatially land use by land use. Moreover, it provides warnings related to a particular land use type so that the users can take appropriate measures.

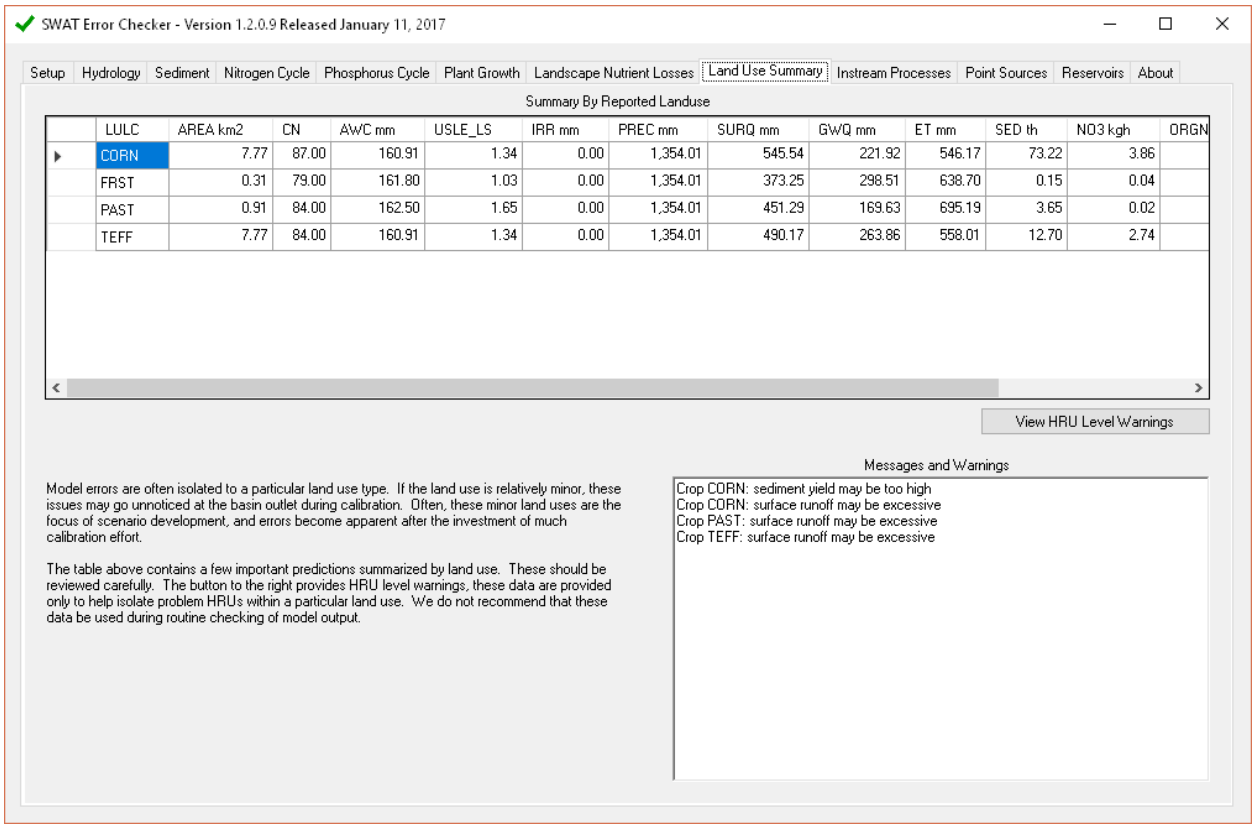


Figure 48: SWAT-Check Land Use Summary window that summarizes different landscape process components for each land use type in the watershed..

5) This manual mainly focuses on hydrology and sediment related outputs of the SWAT-Check. However, in studies related to water quality, besides the functions of SWAT-Check discussed above, **Nitrogen Cycle**, **Phosphorous Cycle**, and **Landscape Nutrient Loss** windows may be of vital importance. Results from the **Plant Growth** window of the SWAT-Check will provide an important guide for users who study agricultural crop yield, and/or biomass yield. Moreover, the **Plant Growth** window provides information about the various crop stresses (e.g. Nitrogen, Phosphorous, and Temperature). In cases where reservoirs are included in the management operations, the **Reservoirs** window could provide important information on reservoir processes. Therefore, all these functionalities in SWAT-Check can be accessed by clicking the respective SWAT-Check toolbars.

6 Appendix I

6.1 Adding tables into SWAT reference database

Tables need to be added to the local copy of the SWAT reference database. (*usersoil* and *WGEN_Robit*).

We need to use a new *usersoil* table in the SWAT reference database. A local copy of this database was made when the project was started, so that changes to it would not affect other projects, and would not be lost if we installed a new version of QSWAT. If you want to change this database and use it in several projects you could instead edit the master copy *C:\SWAT\SWATEditor\Databases\SWATRef2012mdb* before you create any projects, but remember to reinstate your changes if you install a new version of QSWAT.

The tables to be added to the reference database are *usersoil*, using *ExampleDataset\usersoil.xlsx*, and *WGEN_Robit*, using *ExampleDataset\WGEN_Robit.xlsx*. This section shows how tables can be imported from Excel files. Tables may also be imported by other means. In particular they may be imported from comma separated value files or copied directly from other Access databases.

We will use as our first example importing *WGEN_Robit.xlsx* into the copied reference database. *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. You can define a new table in the reference database, as we do here, or replace the dummy data in the existing table *WGEN_user*.

1. Open the copied reference database *C:\QSWAT_Projects\Robit\Demo\SWATRef2012.mdb* in Access. Select **External Data → Excel → Browse** to the *ExampleDataset* folder where the data to be imported is located, select *WGEN_Robit.xlsx* and click **Open**.

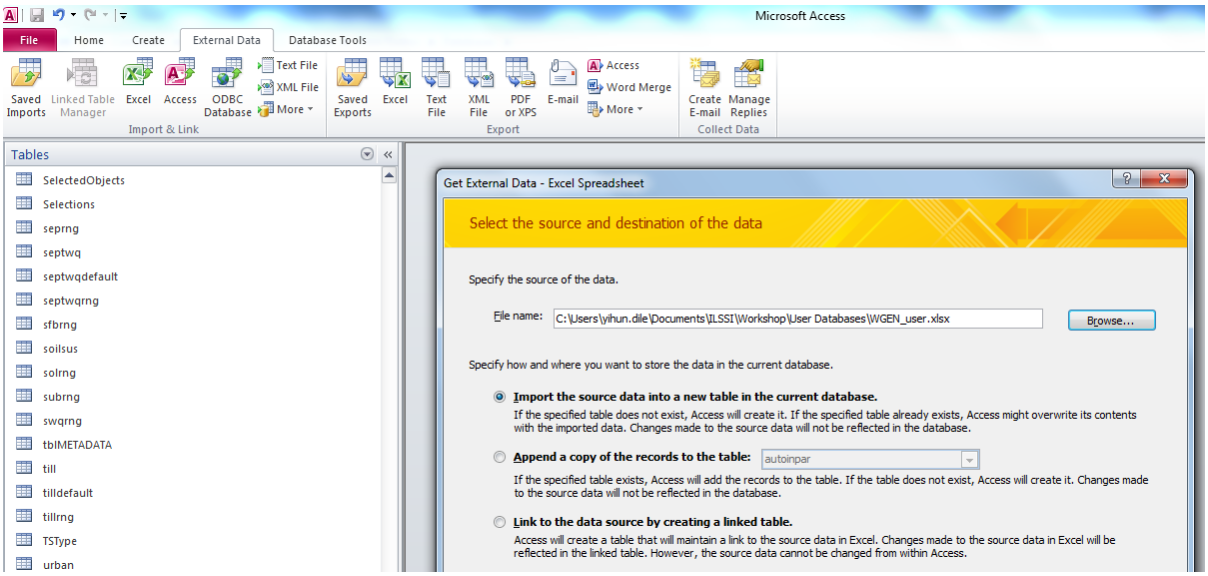


Figure 49: Importing data into SWAT database

2. Select the radio button – **Import the source data into table in the current database** and click **OK**.
3. Follow the wizard - click **Next** three times, and on the fourth step choose the radio button **No Primary Key** (Figure 50) before clicking **Next**.. The table to be appended should have the same format as the table *WGEN_US_FirstOrder* in the database.

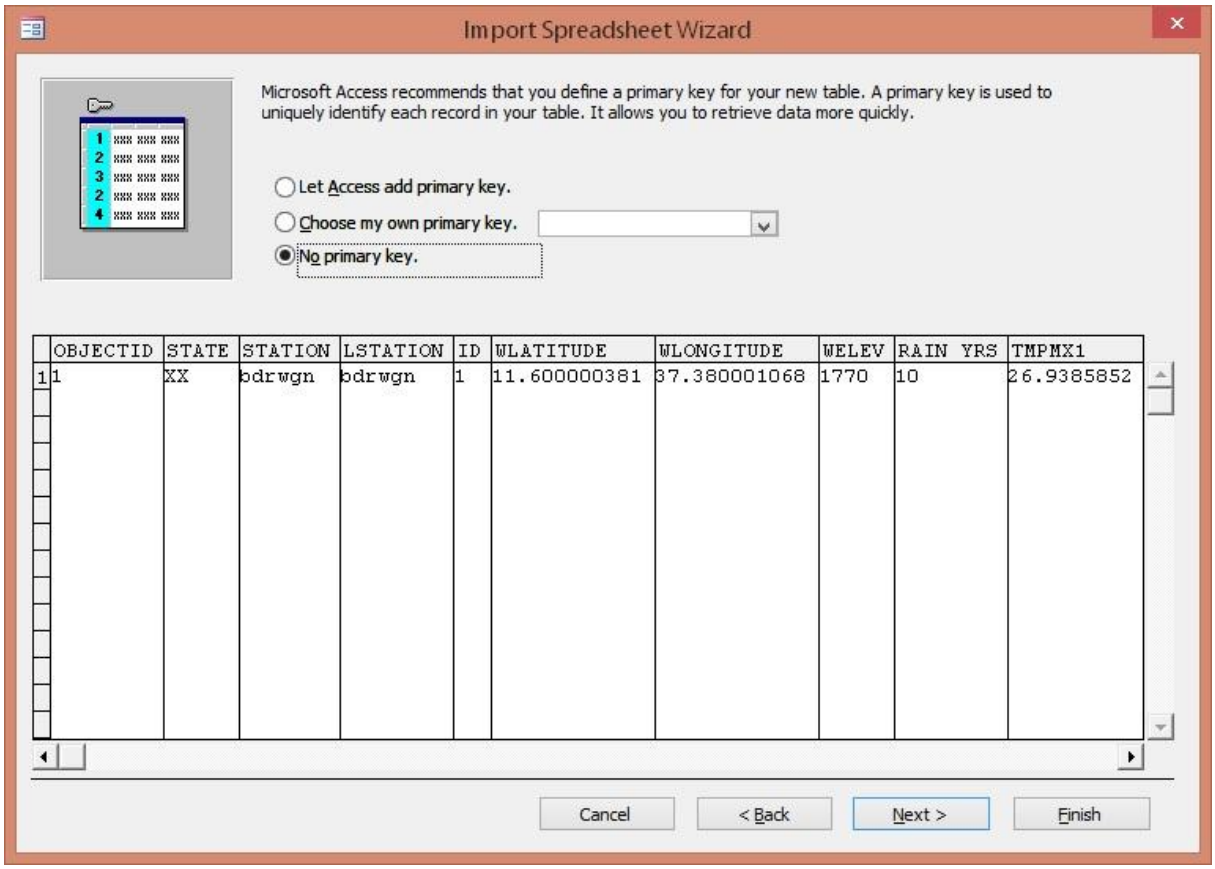


Figure 50: Formatting the table similar to the table WGEN_US_FirstOrder

Name the table with a name starting *WGEN* (e.g. *WGEN_Robit*, Figure 51)

Creating the *usersoil* table in the reference database from *ExampleDataset\usersoil.xlsx* is very similar, except that since there is already a table *usersoil* you will be asked to confirm that you want to overwrite it. The table must be called *usersoil*.

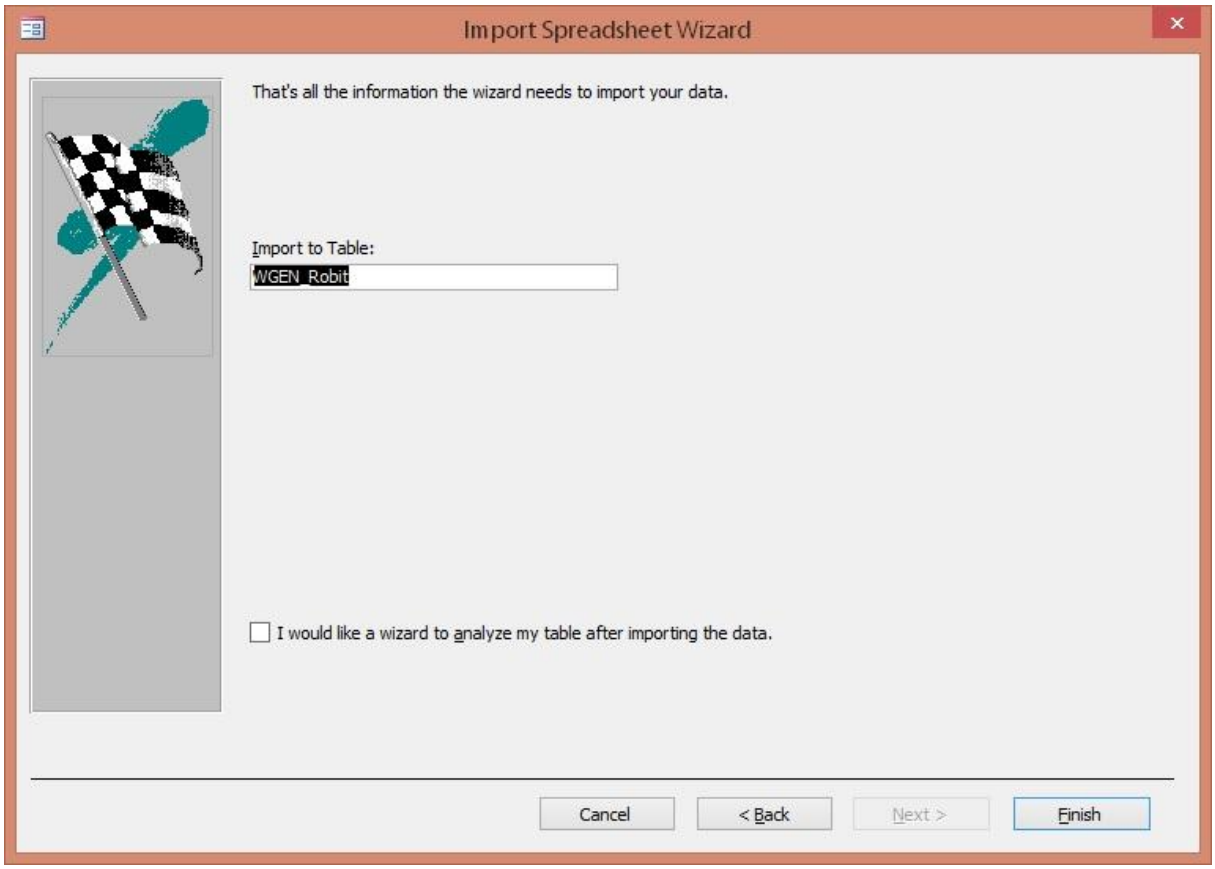


Figure 51: Naming the table

6.2 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 *ExampleDataset\ClimateRobit*, e.g. *pcpRobStation.txt*.

2. The actual climate data should have the same file name as the station NAME in the station data (e.g. robpcp). 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 are included as -99. The data format for the precipitation, solar radiation, wind speed and relative humidity is the same. Refer to the example dataset for precipitation in the folder *ExampleDataset\ClimateRobit*, which is. *robpcp.txt*. The only exception is for minimum temperature and maximum temperature, in which case the minimum temperature and maximum temperature are separated by a comma. Refer to the example dataset for temperature *ExampleDataset\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, QSWAT will use simulated data using the weather generator. The algorithm used by QSWAT 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.

6. Weather generation

When measured data is not available, SWAT uses data simulated by a weather generator program which uses parameters supplied in weather generator files. The SWAT Editor uses a weather generator table, which can contain the parameters for a number of weather stations, each represented by one line in the table. An example weather generator table (*WGEN_user)* is supplied with SWAT Editor, in the reference database QSWATRef2012.mdb (*C:\SWAT\SWATEditor\Databases)*. You can make your own such table, add it to the SWAT reference database with a name starting with the string *WGEN*. Use **Write InputTables → Weather Stations**, select the **Weather Generator Data** tab, and then find *WGEN_user* in the pull-down **Locations Table** menu. It will then appear in this menu. The SWAT Editor will then allocate to each subbasin the nearest weather generation station from the selected table.

There is a weather generation table covering the world available from the SWAT website: http://swat.tamu.edu/media/99082/cfsr_world.zip. Install as explained in the pdf file included in the zip archive, and you will see *WGEN_CFSR_World* as an option for the **Locations Table**.

7. Using global data from the SWAT web site

The SWAT web site offers global weather data (http://globalweather.tamu.edu/). In the website, you can provide the appropriate Lat and Long values for the bounding box of the study area. If your search with your bounding box provides you no (or very limited) station data, you might try a larger search area. You can acquire climate data that ranges from 1/1/1979 to 7/31/2014. It is good to start at midnight (12.00AM), and also select all five kinds of data. You can receive the climate data both in SWAT files format and CSV files in a zipped folder. It is sufficient to select just the SWAT files. The zipped folder will include *pcp.txt, rh.txt, solar.txt, tmp.txt, and wind.txt*. These files give details of the weather stations – name, latitude, longitude and elevation. Use **Write InputTables → Weather Stations**, select the **Rainfall Data** tab, make sure the timestep is **Daily**, and **Raingages** is checked, navigate to *pcp.txt* as the **Locations Table**. Then select the **Temperature Data** tab, check **Climate Stations** and navigate to *tmp.txt*. In similar fashion choose *rh.txt, solar.txt,* and *wind.txt* in the **Relative Humidity, Solar Radiation,** and **Wind Speed** tabs. Select a WGEN table as described above in the **Weather Generator** tab, and click

OK. The SWAT Editor assigns weather stations or gages to subbasins according to proximity and writes the necessary tables and files. It announces when processing is complete, and you click **OK**.

6.3 Preparing landuse and soil look-up tables

For each landuse or soil map the user can prepare a table like *Robit_landuses* or *Robit_soils* and put it into either *C:\SWAT\SWATEditor\Databases\QSWATProj2012.mdb* or the project database *Demo.mdb* in the *Demo* folder (*C:\QSWAT_Projects\Robit\Demo*). In the first case it will be copied into every new project database, but the user must be careful to keep it and replace it whenever QSWAT is reinstalled. In the second case it will only be used on the particular project. The project database is created by copying *QSWATProj2012.mdb* to the *New Project.* Therefore, before *Step 1*, the user should to add any extra tables that are needed for a particular project. But note that it is also possible to have the lookup data for landuse and soil stored in csv files and to use the *Use csv file* option the first time the project is run, as described on page 22.

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 *usersoil, crop,* and *urban* in the reference 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 (page 73).

In the case of a landuse map, the 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 *QSWATProj2012.mdb*. So it must contain at least the columns LANDUSE_ID (type Long 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 and all the ones used in your map must be found in a table *crop* (or a table *urban* if the SWAT_CODE starts with a 'U') found in the SWAT reference database.

In the case of a soil map, the table should have the string *soil* 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 *QSWATProj2012.mdb*. So it must contain at least the columns SOIL_ID (type Long Integer) and SNAM (type Text). The SOIL_ID corresponds to the values in the soil grid. All the SNAM strings you use must be found in a table *usersoil* found in the SWAT reference database.

7 Appendix II – preparing global DEM data for QSWAT

7.1 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 (Section 7.6) and about dealing with large DEMs (Section 7.7) that should be of general interest.

7.2 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*.

7.2.1. 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.

7.2.2. 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.

7.3. Clipping the grids

Start QGIS and load the first grid (using *Layer → Add Raster Layer*, or the corresponding button on the left of the QGIS window). Select *Raster → Extraction → Clipper* to bring up the *Clipper* form (Figure 52). Set the grid to the *Input file*, type <grid_name>_clip_ll, where <grid_name> is some suitable name, as the *Output file*, check *No data value*, and enter the appropriate value, which is -32768 for 90m data and -32767 for 30m data[2]. You can find the appropriate value by right clicking on the entry in the *Layers* panel, selecting *Properties*, and selecting the *General* tab. The no data value is included in the *Layer info* box.

Use *Extent* as the *Clipping mode*, and 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. x here means longitude and y is latitude. Make a note of the values you use as you may need them again. The extent will be shown as a rectangle on the map canvas, so you can check it is where you intend it to be. Check *Load into canvas when finished* and click *OK*. You may get a warning about going outside the raster, which you can ignore if this is what you expected. Don't close the *Clipper* form, so that you can keep the no data value and extent. Right click on the original, unclipped grid in the *Layers* panel, select *Remove* and confirm. You will be left with the clipped portion. Load each of the other grids, and clip in the same way. Remember to change the output file name each time or you will overwrite a previous clip. Close the *Clipper* form when you have clipped all your grids and checked that the pieces abut to form a rectangle as expected.

2 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.

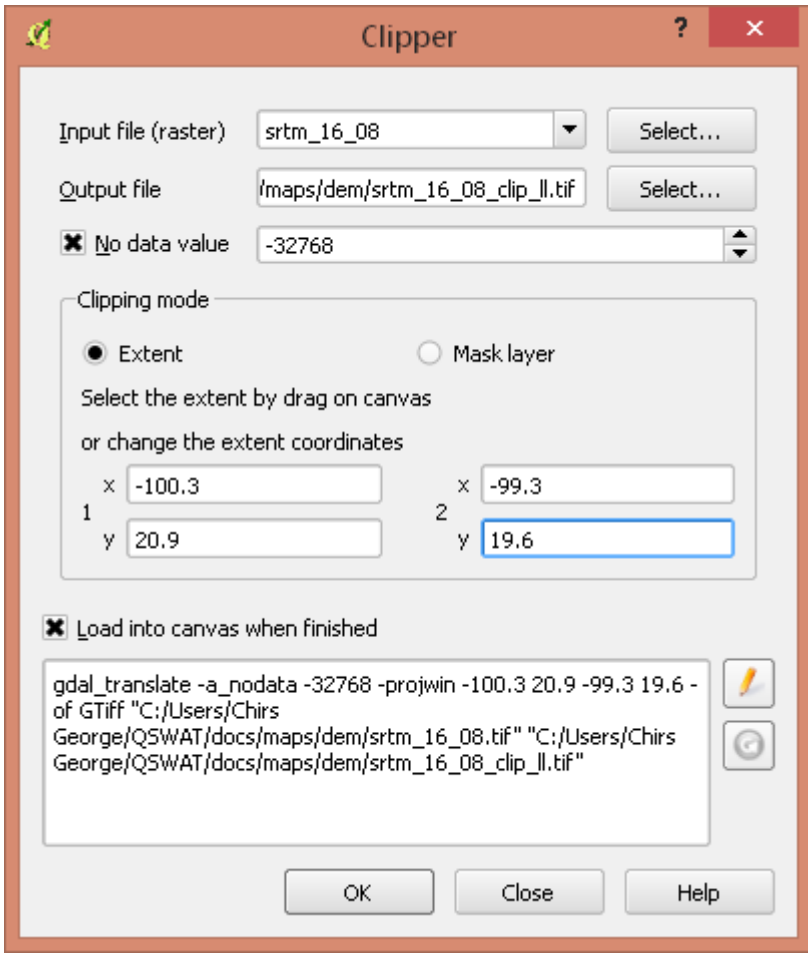


Figure 52: Clipper form

7.4. Merging the grids

You should have all the clipped grids loaded. Use *Raster → Miscellaneous → Merge* to open the *Merge* form (Figure 53). Click the *Select* button next to *Input files* and in the next form navigate to the map folder. Hold *Ctrl* and click on each of the files ending in _clip_ll.tif. Click *Open*. You will be back in the *Merge* form. Click *Select* by *Output file*, which should return you to the map folder, else navigate to it, choose a suitable name, perhaps ending in dem_ll to remind you what it is, and click *Save*. Back in the *Merge* form, check *No data value* and enter the value you used earlier, e.g. -32768 for 90m, -32767 for 30m. If necessary, check *Load into canvas when finished*, and click *OK*. Check the next two messages don't tell you of errors, and click *OK* to close each of them.

Check your merged DEM is OK by removing the visibility of the other layers. If all is well you can *Close* the *Merge* form.

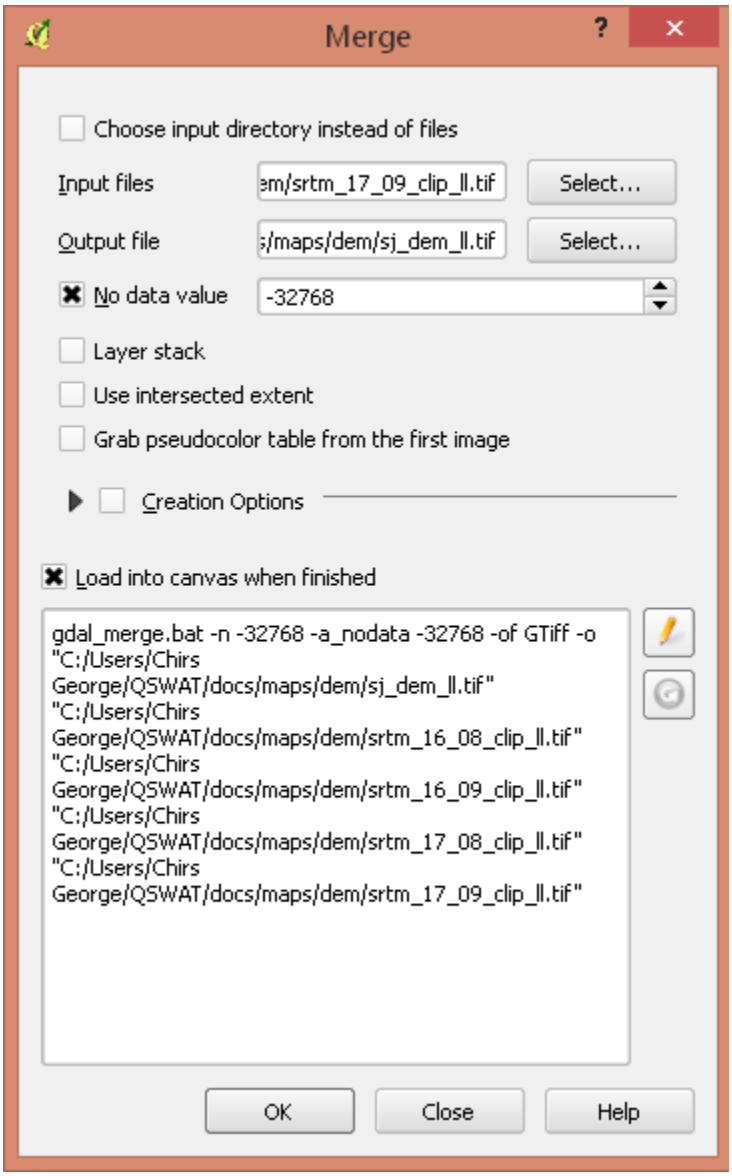


Figure 53: Merge form

7.5. 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 55). Select the merged grid as the *Input file* from the pull-down menu. Click *Select* next to *Output file*, navigate if necessary to the map folder, and type in a suitable name, perhaps ending in dem_utm to remind you what it is.

Check the *Target SRS* box and click the *Select button* beside it to bring up the *Select the target SRS* form (Figure 54). For zone 14 in the northern hemisphere, for example, 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. Note that it has an EPSG number of 32614, while the corresponding zone in the southern hemisphere has an EPSG number of 32714. So if you know the zone and the north or south hemisphere you can calculate the EPSG number. QGIS does not support finding the CRS by EPSG number, however. On the other hand, once you have used it it will appear in the recently used list, making it much quicker next time. Make sure the right zone appears in the *Selected CRS* box, and click *OK*.

Check *No Data values* and put in the same no data value, -32768 for 90m, -32767 for 30m. Uncheck *Load into canvas when finished*: the projection is different from the current lat-long projection and should be viewed in a new QGIS window.

TauDEM recommends that DEMs have square cells. Gdalwarp, the tool that we are using to do the reprojection, has a parameter to generate an output grid with a particular resolution, but the QGIS form does not offer this option. We can achieve it, however, by editing the command in the bottom window of the *Warp* form. To do this, click on the *Edit* button (the yellow pencil symbol) and include the string *-tr 90 90*. This is the parameter for 90 metre square cells. If you were using the 30 metre SRTM data you would use the string *-tr 30 30.* You can put this parameter string before the *-of* parameter, for example. Don't put it at the end as the final parameter is the output file name. See Figure 55. Do not click the *Edit* button or it will remove the extra parameter; just click OK to perform the reprojection. You should see a message saying that processing is complete (click *OK)*, and a second one showing the output is created, finishing

with 0...10...20...30...40...50...60...70...80...90...100 – done. Click *OK* to close this.

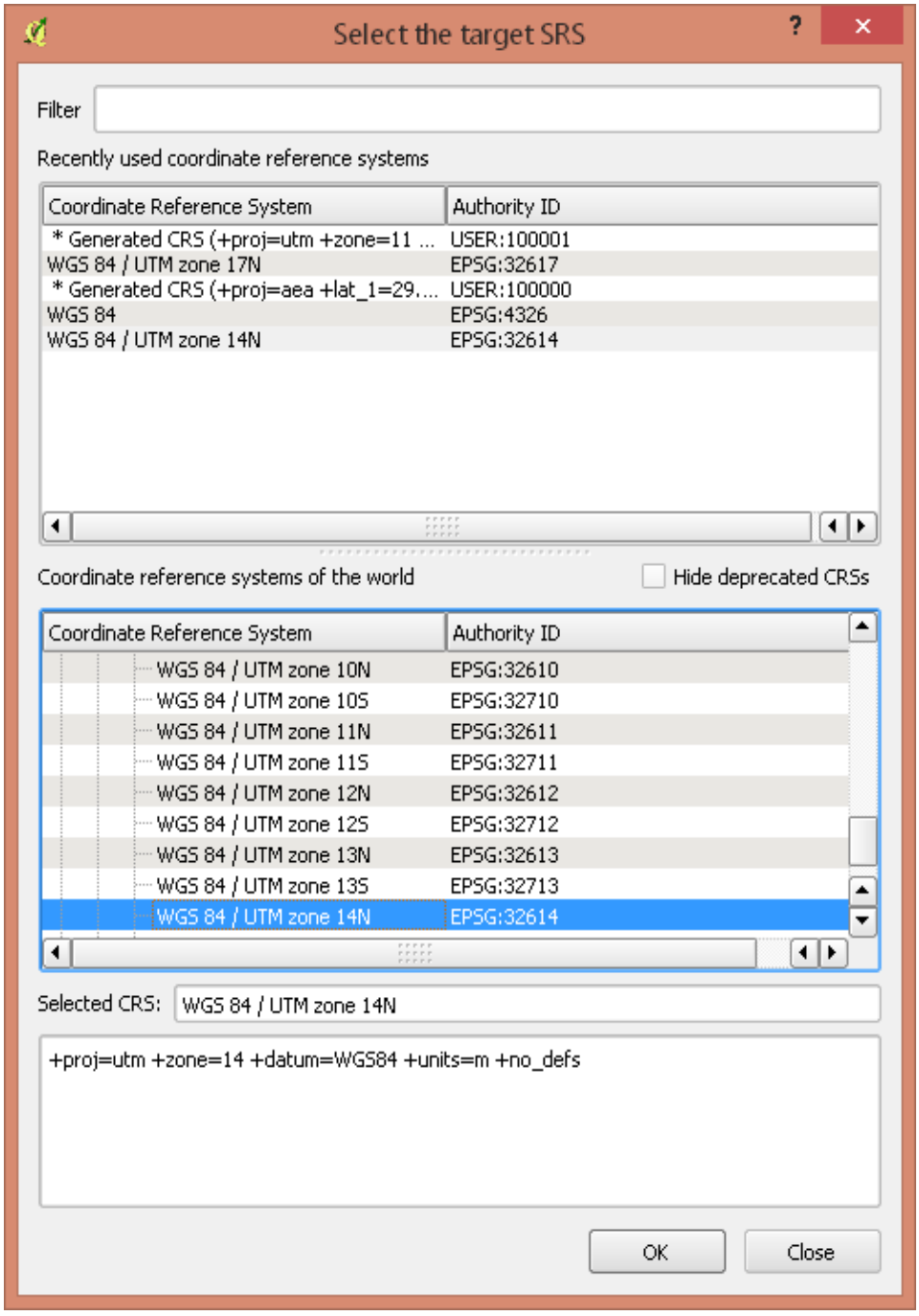


Figure 54: Select the target SRS form

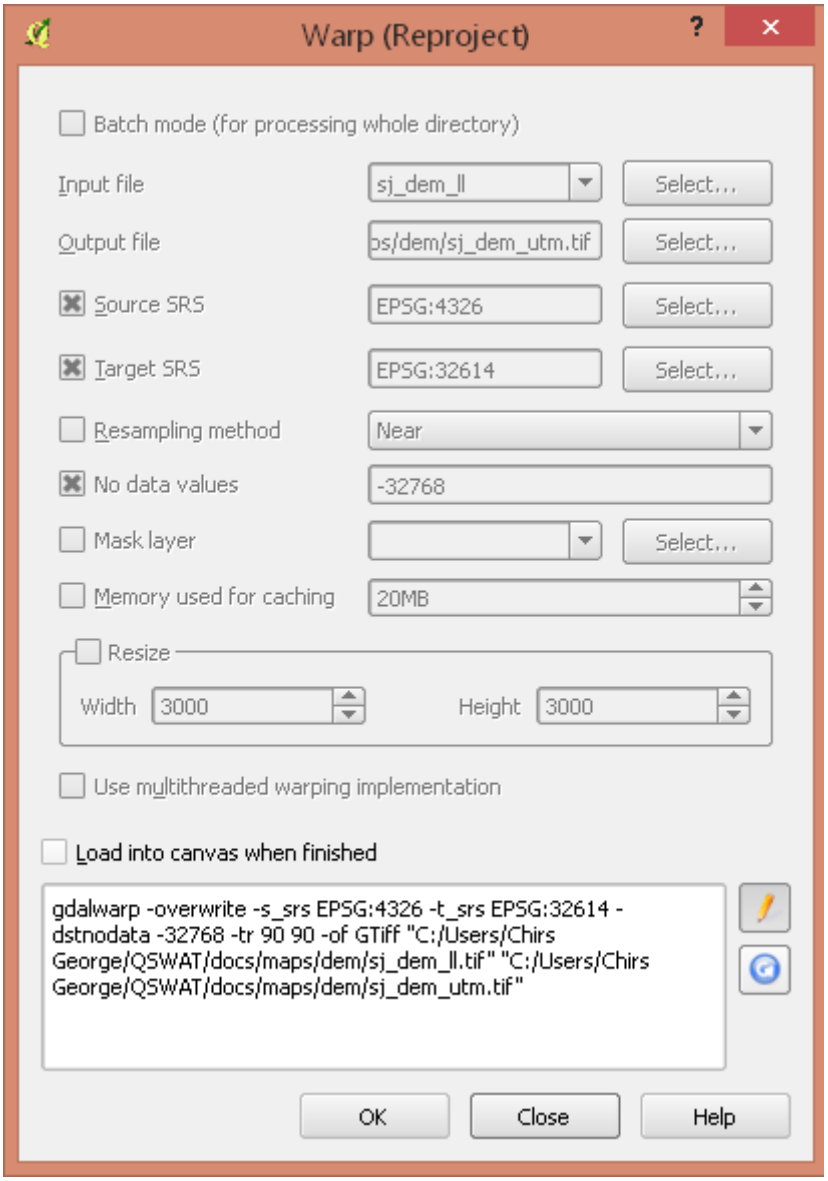


Figure 55: Warp (Reproject) form

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)[3]. If all looks well close the previous instance of QGIS. Choose *Discard* when asked if you want to save the project.

7.6. 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 7.7 for more information 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

3 If you see degrees instead of metres as the coordinates you should open *Project → Project Properties* and select the *CRS* tab. If *Enable 'on the fly' CRS transformation* is checked you are strongly advised to uncheck it. If it is checked then you will not be informed about loading layers with different projections, which can lead to errors in QSWAT.

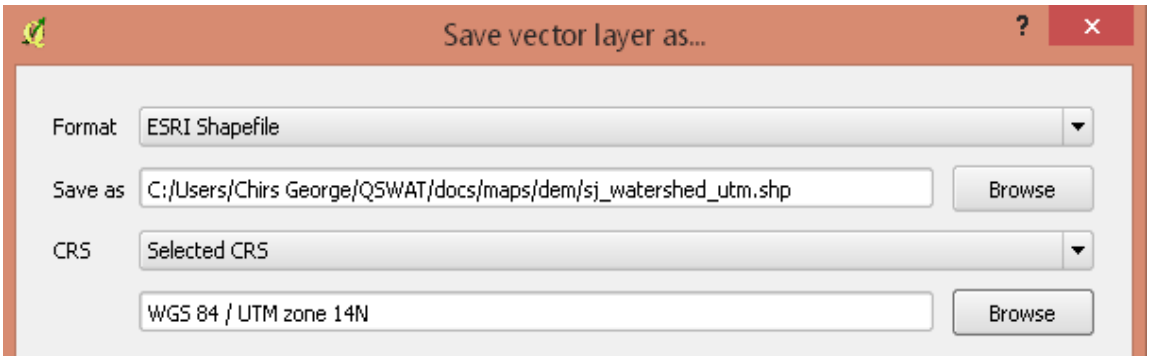


Figure 56: Reprojecting a shapefile

some other projection right click on the entry in the *Layers* panel, select *Save as...*, and in the next form change the CRS from *Layer CRS* to *Selected CRS*, use *Browse* to find the projection, enter a new file name and click *OK* (Figure 56).

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 57) Check the shape lies within the DEM, or we made a mistake somewhere!

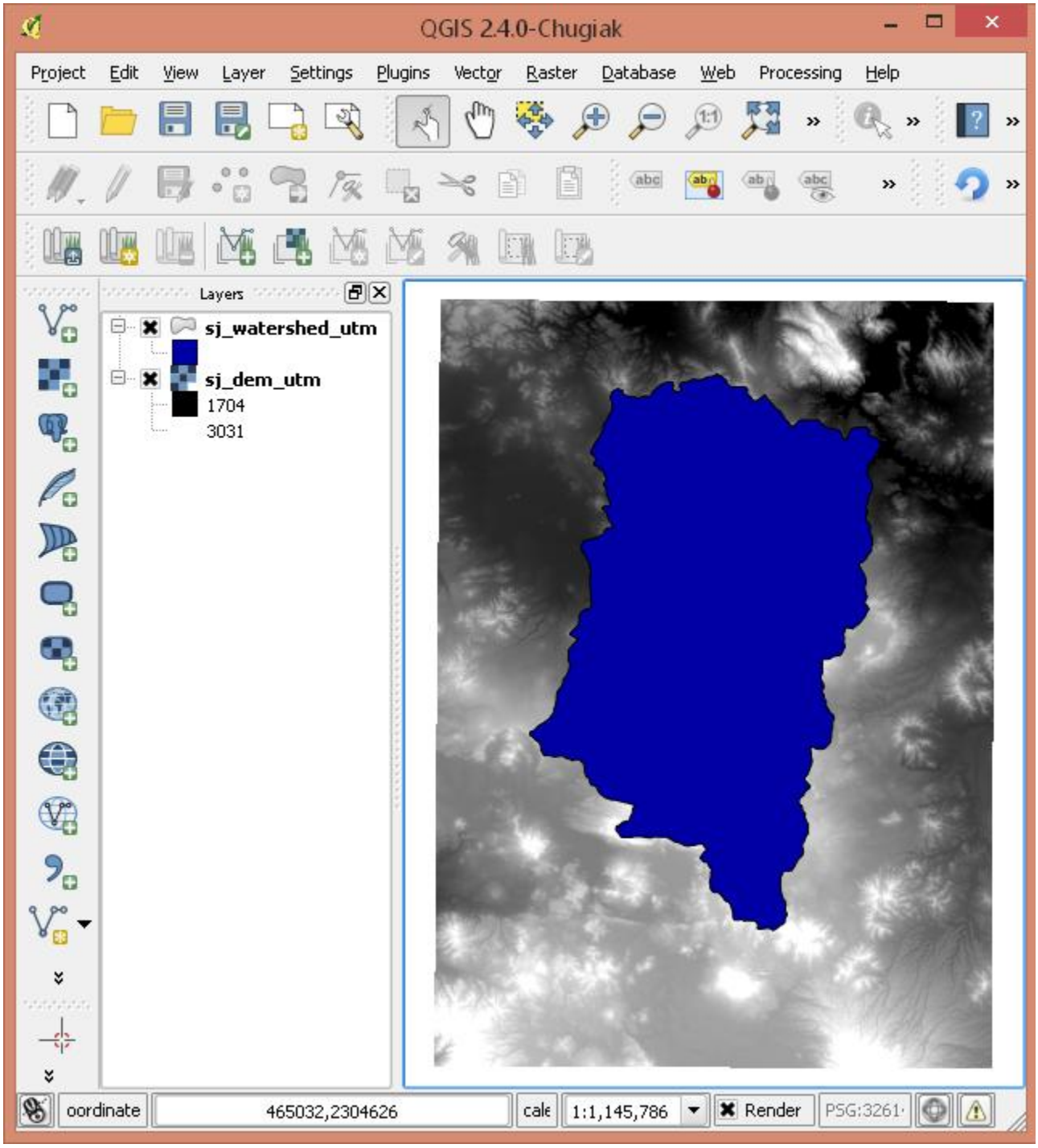


Figure 57: Ready to clip DEM with watershed boundary shape

Then use *Raster* → *Extraction* → *Clipper* to open the *Clipper* form (Figure 58). If not already

selected, select the dem_utm from the pull-down menu as the input, Click the output *Select* button, navigate to the map folder, and use perhaps sj_dem_clip_utm as the file name. Set the no data value to that of the DEM, -32768 for the 90m, -32767 for the 30m. Set the *Clipping mode* to *Mask layer* and make sure the boundary shapefile is selected Click *OK* to reproject, and again to clear the completion message.

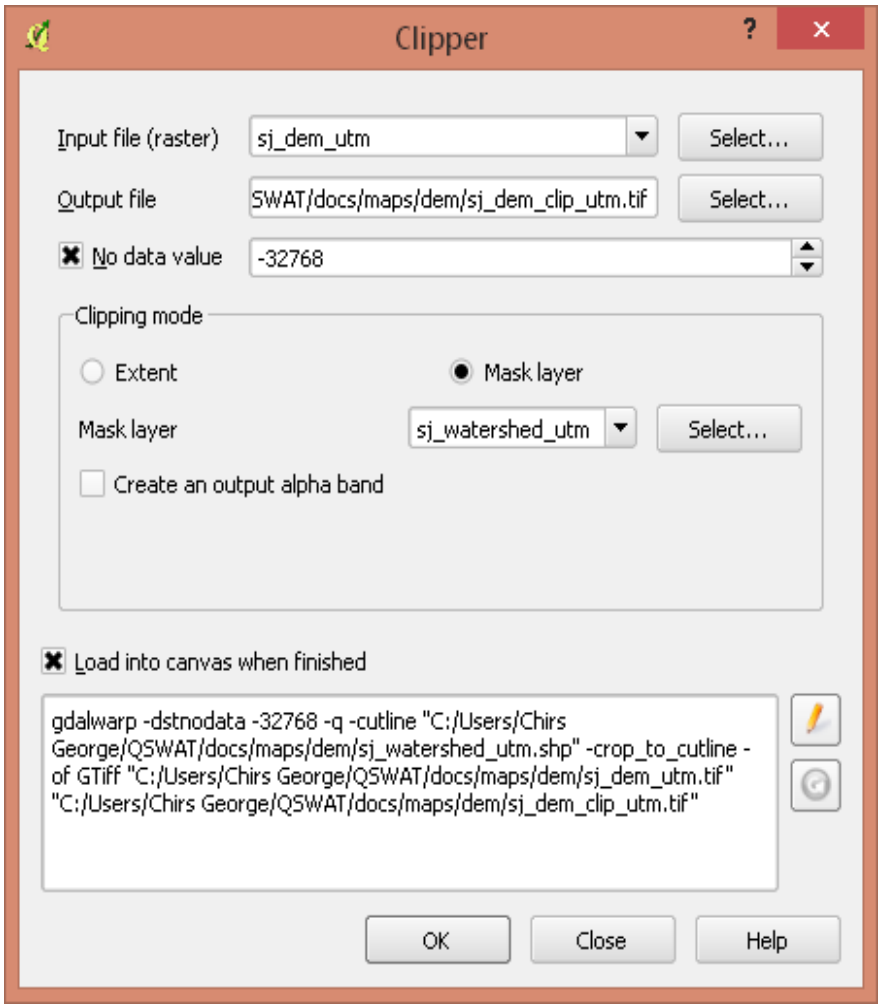


Figure 58: Clipping the DEM by mask

Close the clipper form and remove the dem_utm to check that we have a DEM clipped to the shape.

7.7. 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 (page 89) on 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.

8 Appendix III: Using a Predefined Watershed and Stream Network

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

On this tab you can select the shapefiles containing the watershed and its subbasins (the *Watershed shapefile*) and the stream network (the *Streams 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 *Sources* subfolder of that project, with names ending *wshed.shp* and *net.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, but you will later be able to add reservoirs and/or point sources to existing subbasins.

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

Necessary/Optional	**Name**	**Type**
Necessary	PolygonID	Integer
Optional	Area	Double
Optional	Subbasin	Integer

The *streams shapefile* must be a line shapefile (each line representing a stream 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

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

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[4]. 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 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 watershed 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 form a consecutive ascending sequence starting from 1, 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 streams shapefile: each subbasin must have exactly one stream reach, and each stream reach (unless it is zero length[5]) must have exactly one subbasin. These two fields provide the connection between the subbasins and the stream 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. Files previously generated by TauDEM will meet all

4 Changes to these files are only made to the copies of the originals in the *Source* folder of the project.
5 Zero length reaches are used to enable triple stream junctions.

these conditions, and their values may be safely reused.

There is also an assumption made about the stream reach shapefile. Each stream reach is a single line, represented as a sequence of points. The first point in the sequence 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.

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

Field	**Calculation**
Area	The area of the subbasin in square metres.
Subbasin	For subbasins which are not empty and not upstream of an inlet, a unique number from the contiguous sequence 1 .. n, where n is the number of such subbasins. 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 and stream network is chosen then the length of the stream reach in each subbasin will be used for the maximum flow length in that subbasin.

9 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 the windows version of 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 Table 1 in 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 folder *C:\SWAT\SWATEditor\Taudem5Bin* and rename *msmpi.dll* to,say, *msmpidll* (which is better than removal as you can restore it later if necessary).

9.1 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.

9.2 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* folder 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 Table 1 in Appendix II for some example times for large DEMs on different machines with typical settings of the number of processes.

9.3 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 (Figure 59). 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!

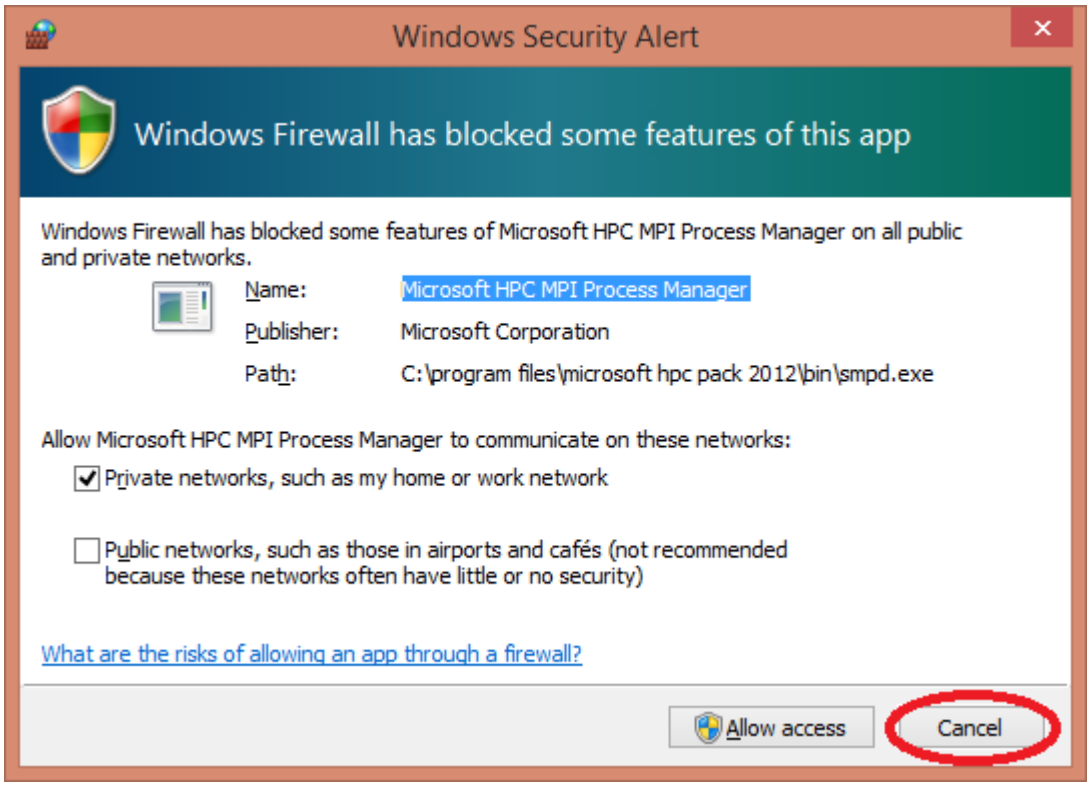


Figure 59: Cancelling network access for MPI