Setting up a SWAT Model with QSWAT

Prepared by
Venkatesh Merwade and Jibin Joseph
Lyles School of Civil Engineering, Purdue University
vmerwade@purdue.edu

October 2021

The objective of this module is to learn some of the basic functionalities of QSWAT, and to use it to create a SWAT model for a watershed. SWAT is a popular model among many watershed hydrologists who are interested in studying the impact of agricultural activities and land use management on the overall watershed health including streamflow and water quality. This exercise is developed by using data for Cedar Creek Watershed in northeast Indiana, but the process can be replicated for any other watershed in the United States by using a digital elevation model (DEM), landuse data and soil data.

Learning Outcomes

Create a QSWAT project
Delineate sub-basins and hydrologic response units in QSWAT
Create input files for SWAT using QSWAT
Set-up and run SWAT simulation
Visualize SWAT Results

Computer Requirements

You must have a computer with latest windows operating system, and the following programs installed:

1. QGIS 3.1x
2. SWAT Editor 2012
3. QSWAT3 v1.1
4. SWAT Check 1.2.0.10

For your personal computers, all these related programs, including their installation instructions, can be found at the following link: *https://swat.tamu.edu/software*.

Data Requirements and Description

This tutorial requires the following datasets for the study area:
(1) DEM (NED 1 arc second ~ 30 m data)
(2) Landuse Map (NLCD 2011 data)
(3) Soil Data (STATSGO2)
(4) Outlet point

You can get the data for Cedar Creek from the following link: ftp://ftp.ecn.purdue.edu/vmerwade/download/data/qswat.zip. Copy and unzip the data in your project folder.

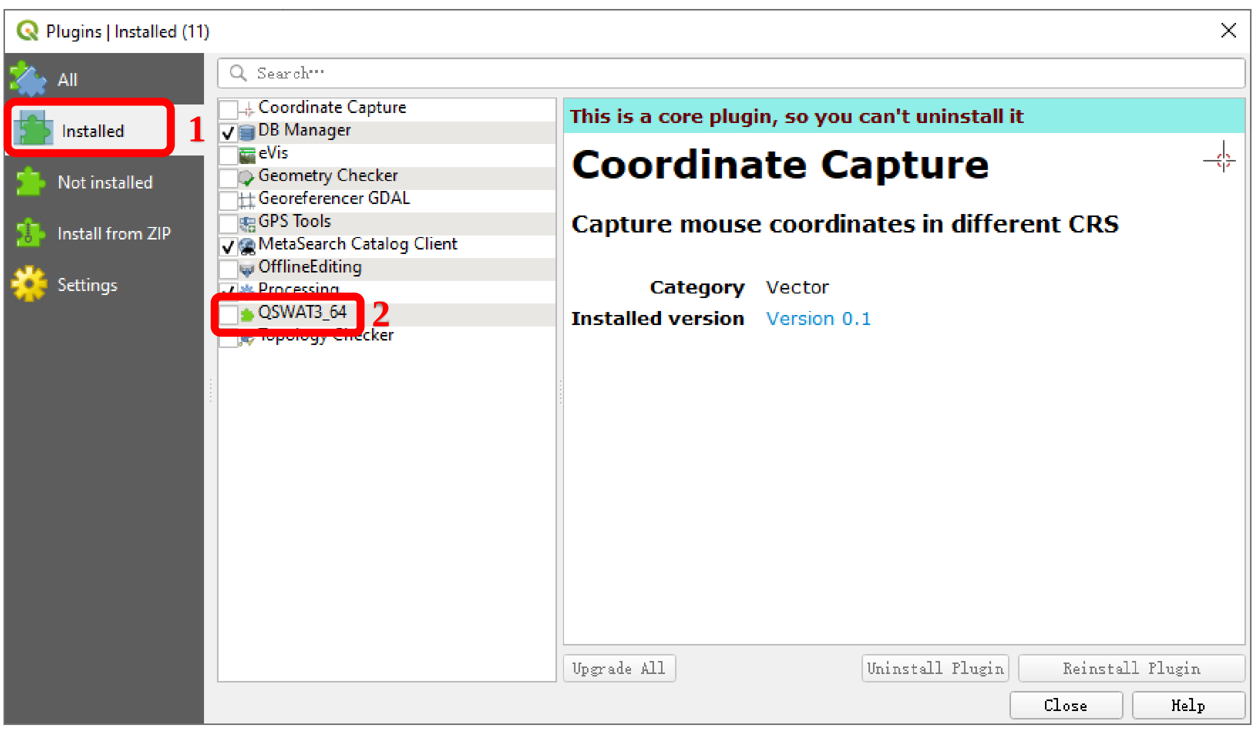
[If you are using this tutorial as a guide for another watershed, make sure that your raster data have some buffer around the study area, and all raster data are in TIFF format]

Adding Soil Data

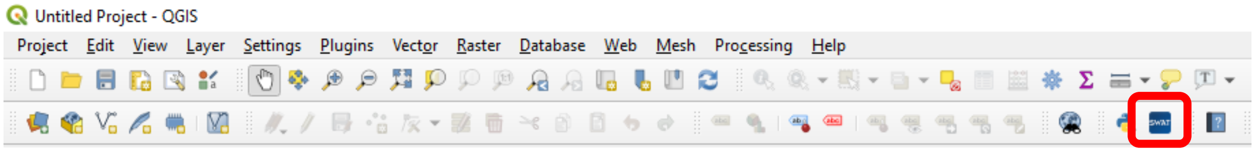
SWAT requires the data for the soils in the watershed. This is stored in an .mdb file SWAT_US_SSURGO_Soils.mdb. In this exercise, we are using STATSGO2 soil data. Both, SSURGO and STATSGO2 use the same database and it can be obtained from the SWAT website. It is stored on the ArcSWAT page at: http://swat.tamu.edu/software/arcswat/ (*you can also find a copy in the zipped data folder*). Place the .mdb file (SWAT_US_SSURGO_Soils.mdb) in the Databases subfolder of the SWAT Editor folder (C:\SWAT\SWATEditor\Databases). QSWAT can find it and pass it to the SWAT Editor for generating the soil input files. If you are unable to access and store the .mdb file on C:\ drive, save it in your folder and make sure you specify the correct path in the SWAT project (described later).

Instructions

Open QGIS 3.x Desktop to create an empty document. After installation, QSWAT toolbar may not appear in the toolbar. To enable QSWAT, navigate to *Plugins→Manage and Install Plugins*. Select *Installed* tab on the left and check the QSWAT plugin as shown below.



Close the window and you will notice that SWAT button appears on the toolbar.

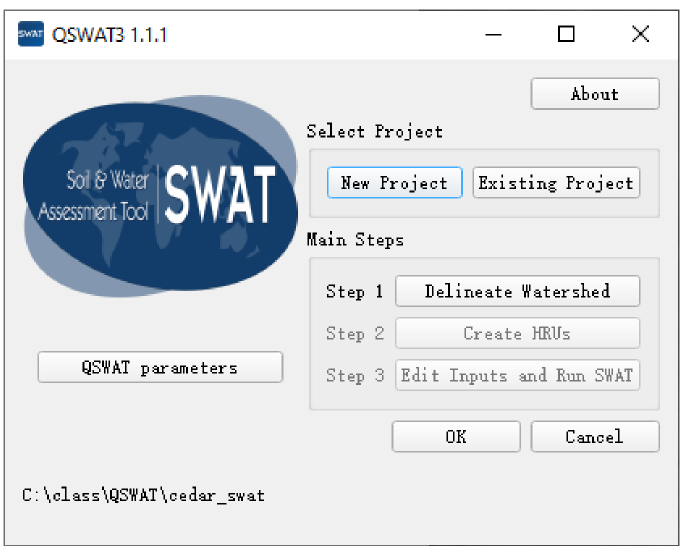


Checking the Projection

Add digital elevation model (DEM), landuse and soil data to the map document. Check that all the datasets are in same projected coordinate system (*NAD1983 UTM 16 N for this tutorial*). To add raster data, go to *Layer→ Add Layer→Add Raster Layer*. The STATSGO soil data may appear with only one color (white), but can uncheck this layer or move other layers (using right click) to see other datasets. Next, add the outlet shapefile to the map document. The outlet point shapefile is needed only to delineate the outlet point in the watershed delineation step. If you know the location of the outlet on the map, you do not need the point shapefile. To add vector data, go to *Layer→Add Layer→ Add Vector Layer*.

Project Setup

Click the SWAT icon ( ) in the toolbar. This will open a new window where you can either create a new project or select an existing project (this window is called QSWAT main form). Go ahead and **click** on *New Project* (name it as cedar_swat) and save it in your working directory. After you save the project in the working directory, the *Main Steps* frame below the Select Project frame becomes activated. You will notice that Step 1 is now active and other two are still inactive. Also, you will see location of your working directory in the bottom left corner.

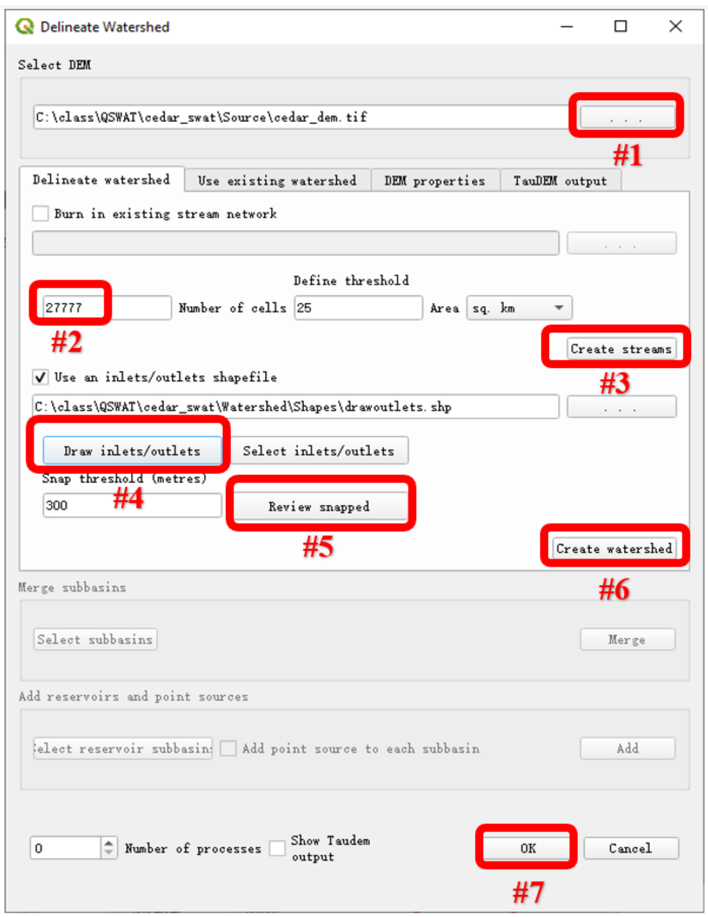


Now, you will notice that a QGIS project and a folder is created in your working directory with the same name. In windows explorer, if you open the folder, you will see few folders and databases are created in this folder as shown below.

Q1. Go to your project folder and note what new folders or databases are created. Write down the name of these databases and folders. Some folders will also have sub-folders. Write down their names and the name of the folder under which they are created.

At a later stage, all these folders will hold the data you will create in this project. Without creating any data, it is interesting to observe that the size of **QSWATRef2012.mdb** file has a considerable size.

Click on *Delineate Watershed (Step 1)* and a new window opens as below.

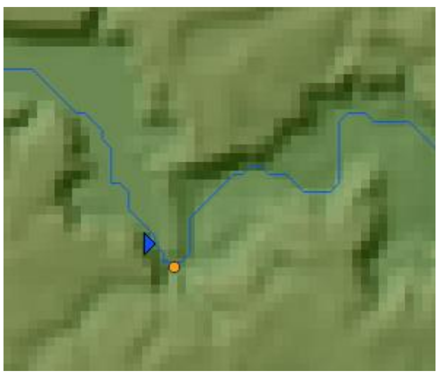


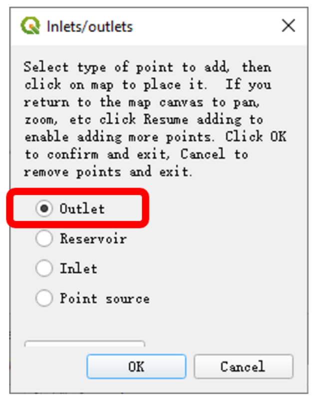
Firstly, **select** (#1) the projected digital elevation model (DEM) for Cedar Creek, which is available in the data folder (cedar_dem.tif). A copy of the DEM will be added to the *Source* folder. If you look in the DEM properties tab, you will notice properties of the selected DEM. Now, go back to Delineate Watershed tab. In the *Define threshold* text box (#2), number of cells will be populated automatically based on the DEM that was provided in the earlier step. This critical stream area threshold will be used to define the stream network. We will use 27777 cells (25 km^2) as the area, which is equivalent to 2500 Ha.

<<u>Note: The threshold area of 25 km^2 (2500 Ha) is approximately 3.5% of the Cedar Creek watershed area. Make sure you use appropriate percentage area threshold for your own watershed. Blindly using 25 km^2 may give undesirable stream network delineation for other watersheds.</u>>

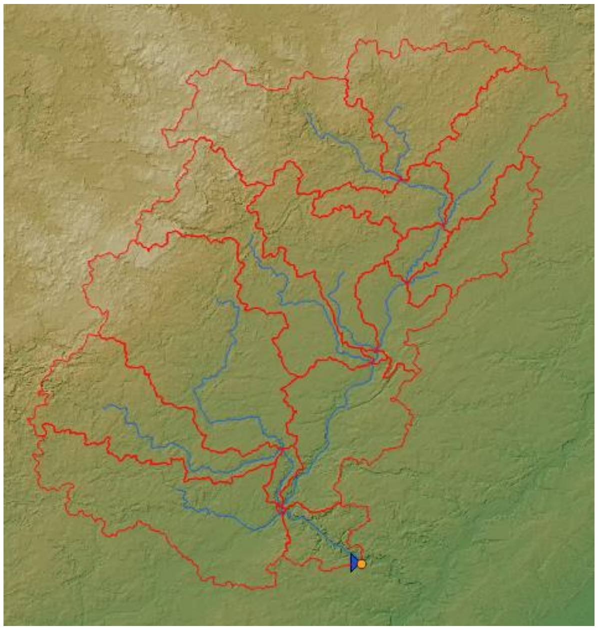
Now, **click** on *Create streams* option (#3) to start the process of generating the stream network [if you get a message regarding MPI program is not found, **press** *OK*. *Also if you get other warnings or messages related to firewall access, simply cancel those messages for the program to proceed*]. You will see that the tool is creating flow direction grid, flow accumulation and the stream network based on the information provided. Open the attribute table of the stream features and check the fields that are added to the attribute table. How many streams features are populated for this DEM?

We will use a point feature (add if it is not present in the map : cedar_outlet.shp) to create an outlet point on the stream network. Zoom in to the outlet as much as possible. Now, **click** on Draw inlets/outlets (#4). **Select** a point (upstream of outlet, close to the streamline) on the map along the stream, and **press** *OK* in the intermediate window.



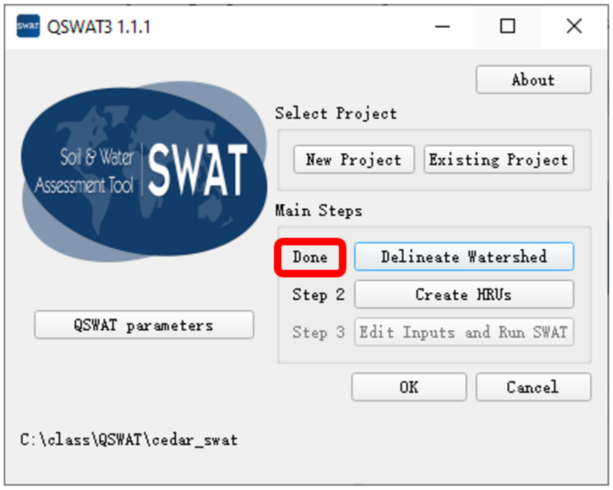


If the delineated point does not fall on the stream for some reason, the threshold distance specified in the *Snap threshold* option should pull the point towards the stream. Use default 300 m and click on *Review snapped* (#5). You will get a message in this case as “1 snapped”. Now, click on *Create watershed* (#6) and the watershed polygon feature class with sub-basins will be created and added to the map display as shown below. What happened to streams outside the watershed?



Open the attribute table of the watershed and you will see different fields that are created and populated in the table. Also, it does not have any hydrology or watershed specific attributes (e.g., slope, elevation). The Sub-basin field also has *Null* values. Let us leave the other two options default (*Merge subbasins & Add reservoirs and point sources*). Finally, **press** *OK* (#7) at the bottom of the window. If you again open the attribute table, you will see that Subbasin field is populated. How many sub-basins are created with the specified threshold? Close the attribute table.

You are done with delineating the watershed and creating sub-basins for a specific threshold area. Save your project. In the QSWAT main form, you will notice that the status of *Step 1* is changed to *Done* and *Step 2 Create HRUs* is now active as shown below (but Step 3 is still inactive).

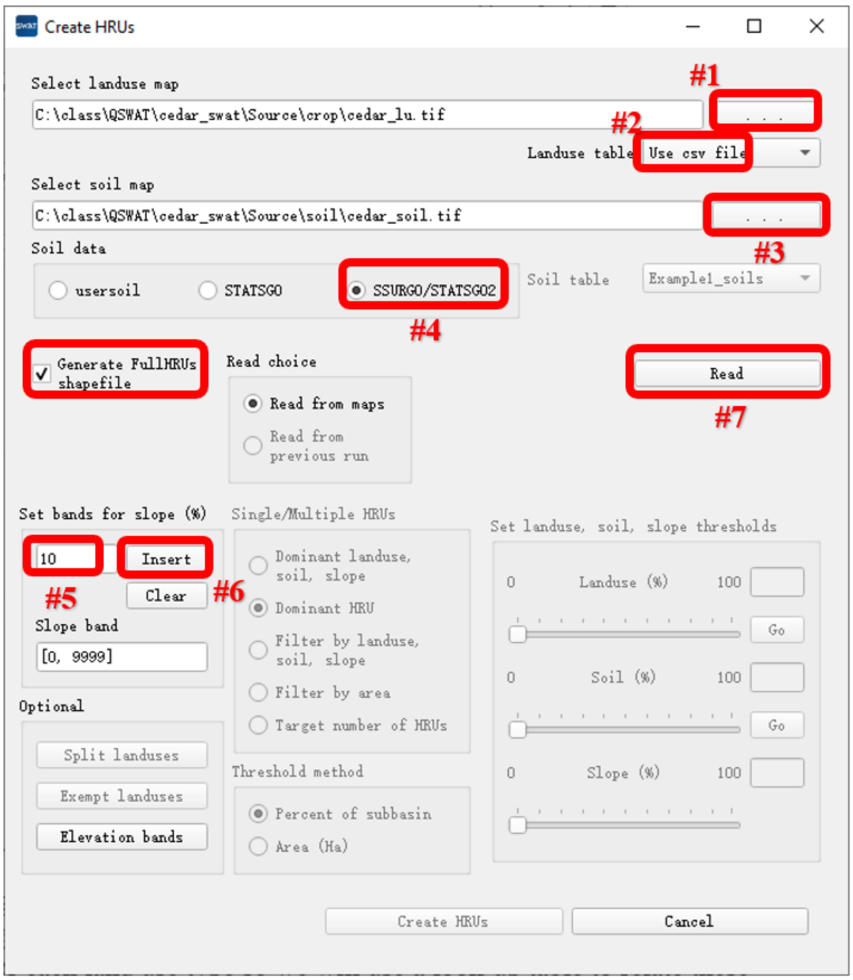


After you are done with watershed delineation, the next step is to create HRUs by using land use, soil and elevation (slope) information.

HRU Analysis

Once, you have completed the *Step 1 – Delineate Watershed*, the *Step 2 – Create HRUs* will be activated. Now, click on Create HRUs and new window opens. We have to provide input related to three different data:

a) Landuse Raster
b) Soil Raster
c) Slope Definition



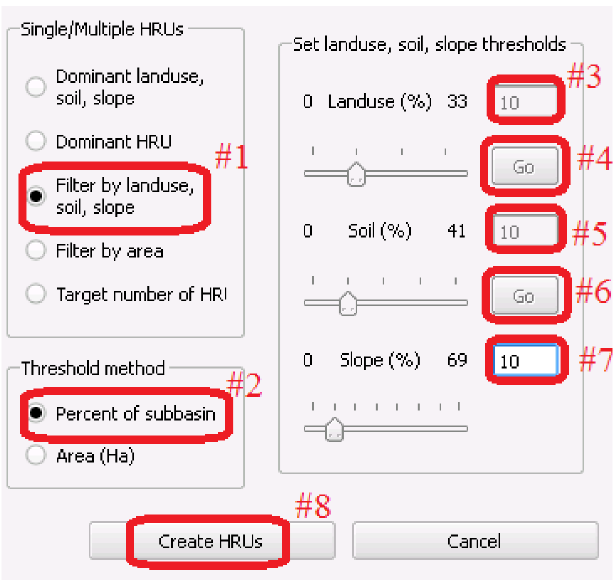
Click on the *Select landuse grid* button (#1 in the figure above) to load the landuse data (*cedar_lu.tif*) and change the landuse table to *Use csv file* option (#2). Land use data typically comes with numbers (values) for each land use type so we will use a look-up table to relate these numbers to specific land use types. Now, load the soil map (#3) and check the SSURGO/STATGO2 option (#4). In this exercise, we are dealing with soil map from STATSGO2. Unlike, usersoil and STATSGO options, SSURGO and STATSGO2 do not require a lookup table.

The next step is to assign slope attributes to each HRU. Now, in the text box for slope, enter value of 10 (#5). It would create two bands for slope, namely 0-10% and 10% & more. Click on *Insert* (#6) to introduce the value between 0 and 9999. After checking the box for *Generate FullHRUs* *shapefile*, click on *Read* (#7). In the browse window, load the landuse lookup table (available in the data folder - *lu_nlcd2011.csv*).

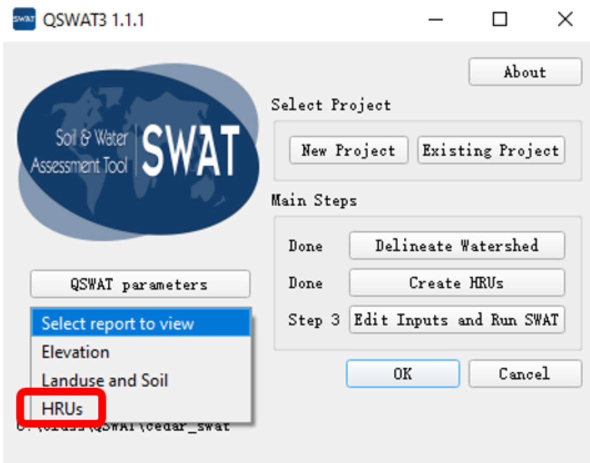
[There is also an option Read from previous run. This can be used when re-running the project, provided you have not re-run the delineation step, and not changed the soil or landuse inputs, to recover information from the project database instead of rereading the grids or maps. Reading from the project database (database created in previous run) is substantially faster.]

HRU Threshold Definition

To create HRU by using the combined land use, soil and slope data, we have to input the threshold in either percentage or area. Let us now define the threshold for landuse, soil and slope as shown below. Depending on the version of QSWAT, you may have to push the Go button after specifying one threshold and then move on to another data. After the thresholds are specified, push the Create HRUs button.



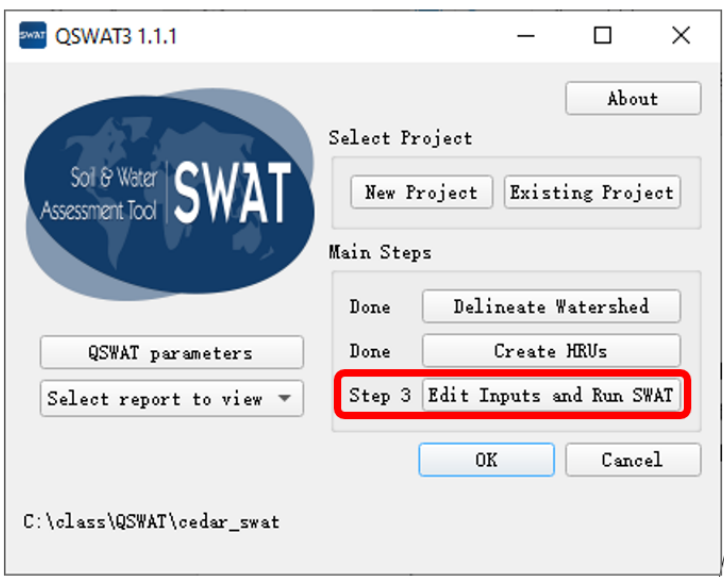
After the process is completed, you can view the report on HRU creation.



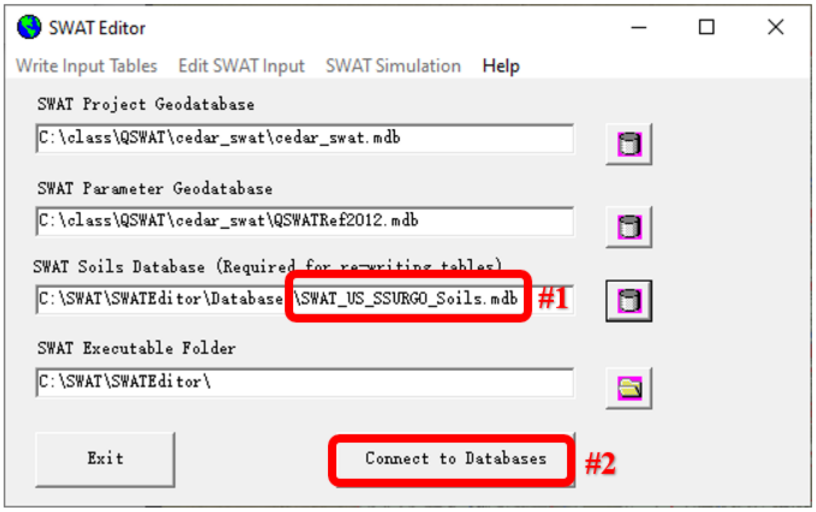
In the report, you will see that several (around 100 +/- 10) HRUs with unique combination of landuse, soil and slope are created for the study area. These combinations are listed under each subbasin in the report.

Creating Input Files/Tables

After all geo-processing is done on DEM, land use, and slope data to create sub-basins and HRUs, the next step is to create input files for SWAT including weather data. Now click on *Step 3 – Edit input* in the QSWAT main form.



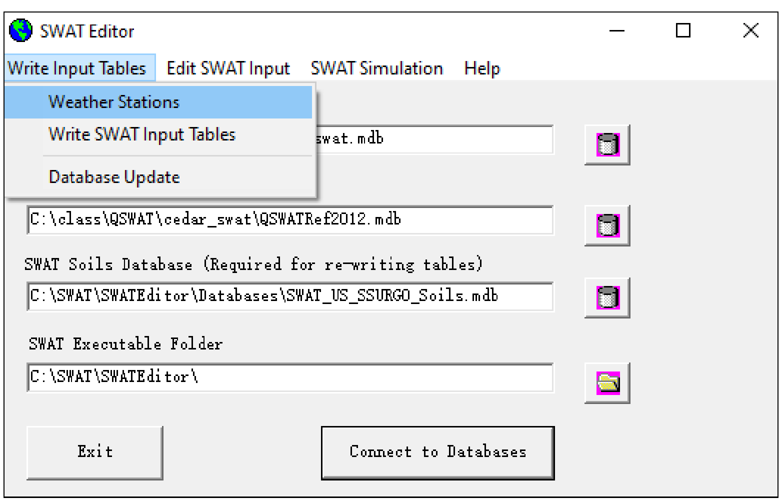
Once the *SWAT Editor* window is open, check whether Soil Database (#1 below) is selected as *SWAT_US_SSURGO_Soils.mdb* (this was manually added to C:\SWAT…..as we are using STATSGO2 soil data). Make sure the locations of these databases are correct. Press *Connect to Databases* (#2 below). This process will connect the data in the map to the SWAT database. Now, you will get a message indicating that the connection was successful and the drop-down menus are activated (Write Input Table, Edit Swat Input, and SWATSimulation menu).



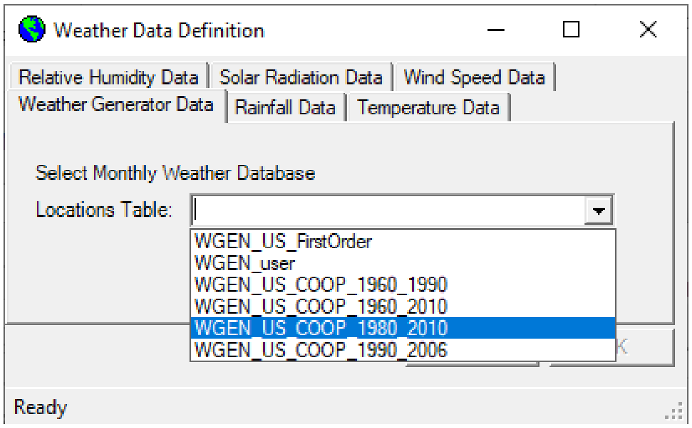
There are four connections addresses in this window. The first one indicates the address to project geodatabase which is located in the project folder. The second one is associated with SWAT Parameter geodatabase. It cloned from the QSWAT database located along with the installation folder. The third one refers to soil database. In this exercise, we manually added this file for using STATSGO2 (or SSURGO) soil data. The last one refers the location of SWAT Executable folder.

Create Weather Input (using weather generator for this class)

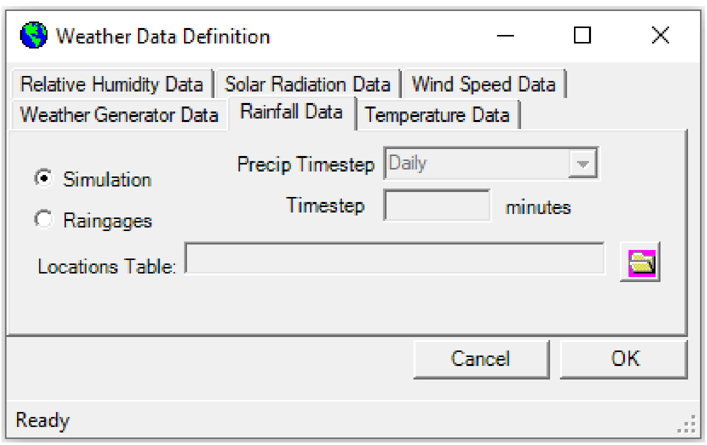
To create weather input for SWAT, click on *Write Input Tables→Weather Stations*



There are two options to input weather data. The first option is to use the weather geodatabase from SWAT, and the second option is to use your custom database. For this exercise, we will just use the internal weather geodatabase within SWAT. In the Weather Generator Data tab, select *WGEN_US_COOP_1980_2010* from the locations table drop down menu to load the data.



Next, in the *Rainfall Data* tab, leave the *Simulation* option checked with daily precipitation time step. The selection of "Simulation" option will tell the program to generate weather data from the internal weather generator within SWAT. For other weather variables, including temperature, relative humidity, solar radiation and wind speed, leave the *Simulation* option checked.



Finally, **click** *OK*. This will create a weather database for the study area. [If you get a warning message stating GetOleDbSchemaTable, ignore the message by clicking OK]. Then, you will get a message that processing is complete. Now, **click** *OK* on the dialogue box.

Weather Data (using observed precipitation - for your class project)

First, create a precipitation (pcp file) and temperature (tmp file) data for your watershed. A typical examples for creating weather input files can be obtained at:
C:\SWAT\SWATEditor\Databases\ExInputs.

To use observed rainfall data, click on Rainfall Data, check the Raingages option, and browse to the pcp file. Click OK.

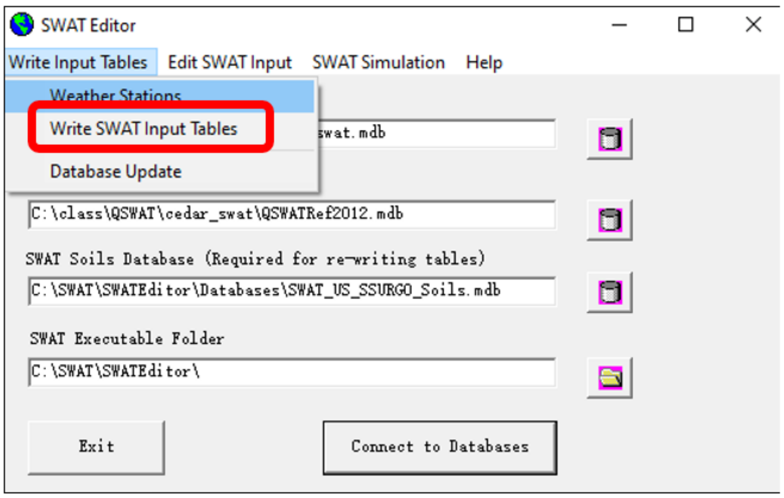
To use observe temperature data, Click on Climate Station, and browse to tmp file. Click OK.

All other weather files (solar radiation, relative humidity, wind speed) can be used from SWAT weather generator.

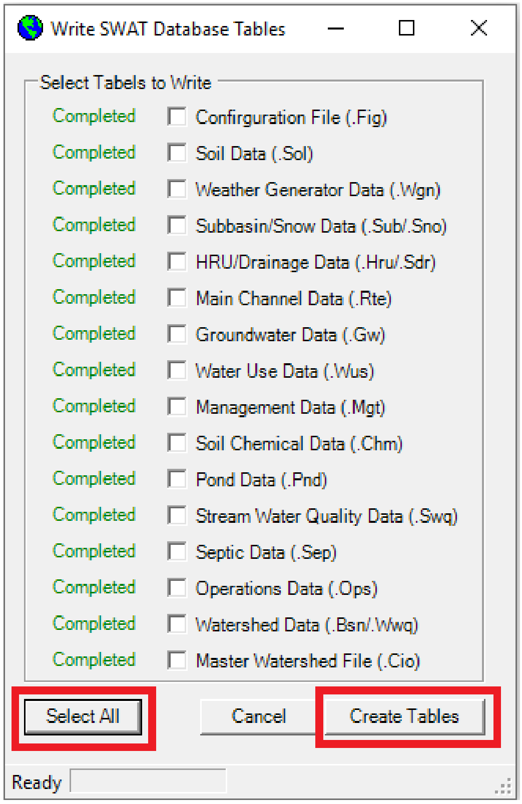
Now, click on Cancel in the Weather Data Definition window.

SWAT Input Files

After generating weather database, you can create all other input files by selecting *Write Input Tables*→ *Write SWAT Input Tables*.



A new window named as *Write SWAT Database Tables* will be displayed. Click on *Select All* and then click on *Create Tables*.

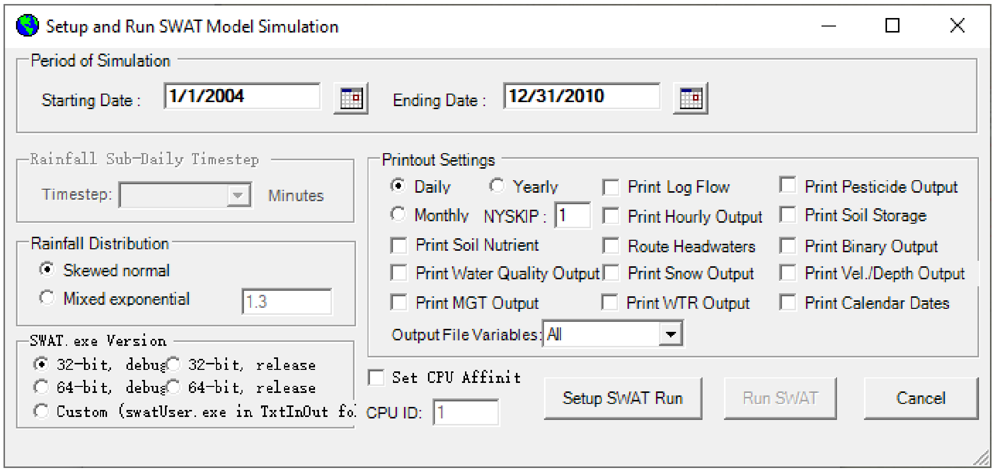


Choose Yes in all the subsequent messages which may show up. You should get a final message saying all input tables were created. Now you are ready to run the simulation! Click on Cancel to exit the window

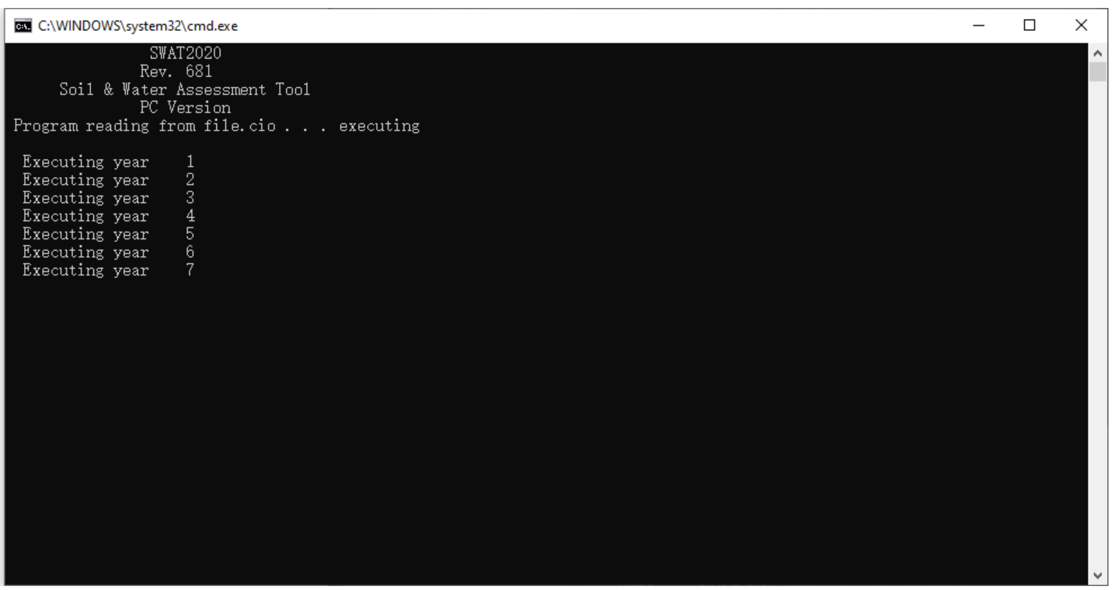
Running SWAT Simulation

Click on *SWAT Simulation→Run SWAT*

Set the period of simulation from 01/01/2004 to 12/31/2010 and change the printout settings to daily, and leave other default options unchanged. NYSKIP means model simulation warm-up period and for this tutorial, we will use NYSKIP as 1 year. Click on *Setup SWAT Run*. You will get a message indicating SWAT Setup is finshed. Now, **click** on *Run SWAT*.



While SWAT is running, you will see a DOS window showing you the progress, and after the simulation is done, you will get a final simulation completion message.

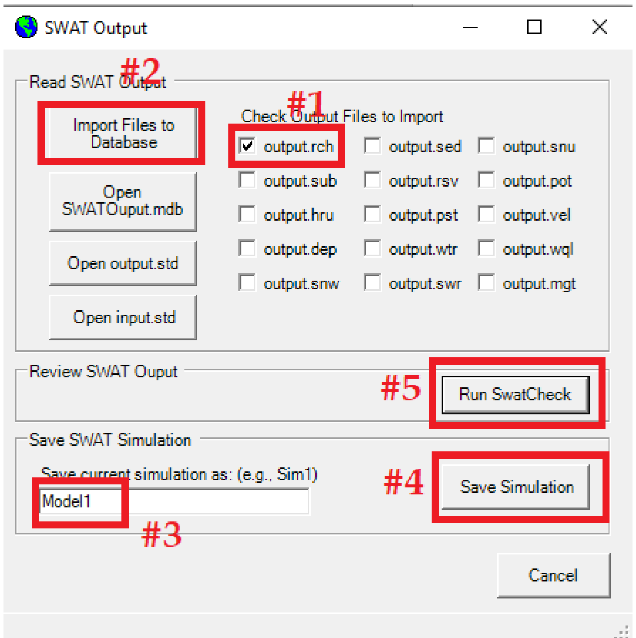


After getting a successful simulation message, close the SWAT simulation setup window. You are now ready to read and visualize the model output.

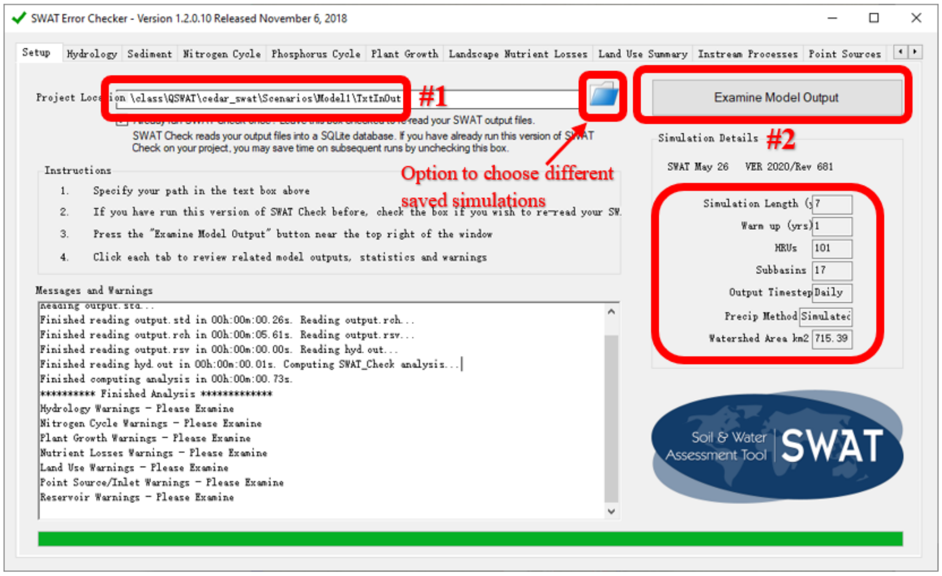
Error Checking

The SWAT Error Check tool can be used to find out issues with the modeling in the early stage. It enables us to find out any hidden problems and reduce the time in regenerating or recalibrating the model at a later stage. SWAT Check performs mainly three operations: (i) it reads the output and notify the users if the values lie outside the possible limits, (ii) it produces process-based figures for quick visualization, and (iii) also find and notify the common model errors.

Click on *SWAT Simulation*→*Read SWAT Output*

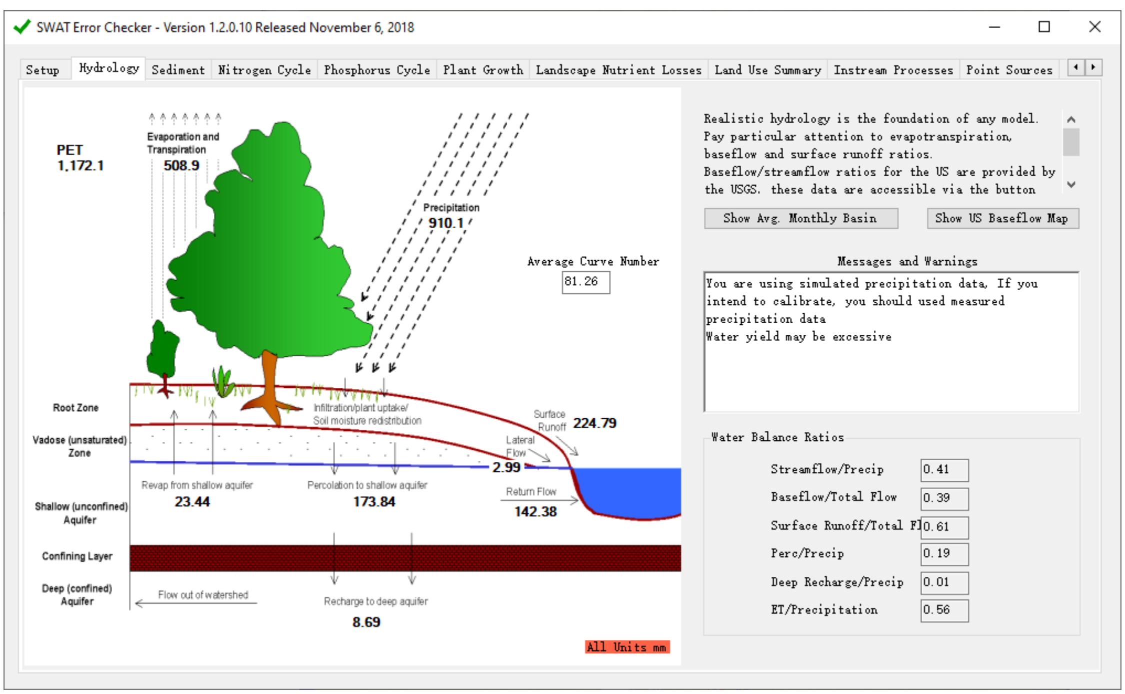


Select *output.rch* as shown above (#1), and click on *Import Files to Databases (#2)*. Also, SWAT simulations can be saved using the *Save SWAT Simulation* with a user input name. Type in simulation name as *Model1* (#3) and press on *Save Simulation*. This may be useful when you change the parameters during the calibration and validation of the model. Finally, click on *Run SwatCheck* option.



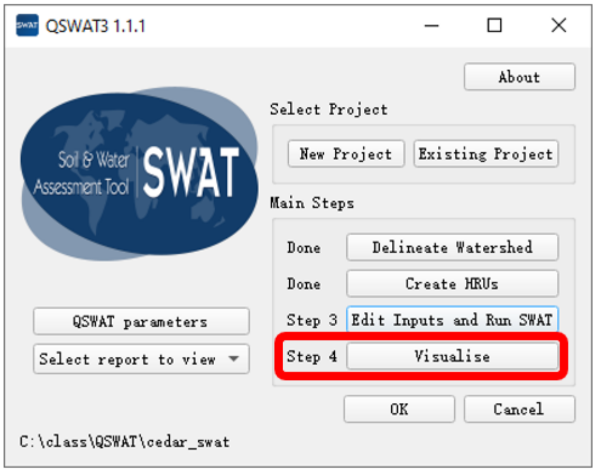
In the SWAT Error Checker option, change the Project Location to *Model1 (*saved in the earlier step*)*. **Click** on *Examine Model Output* option to read the output from SWAT Project. This will examine the simulation and create different statistics related to the project. The watershed area might be different based on outlet selection in Delineation step.

Once it is finished, click on Hydrology to get a watershed view for the selected simulation as shown below.

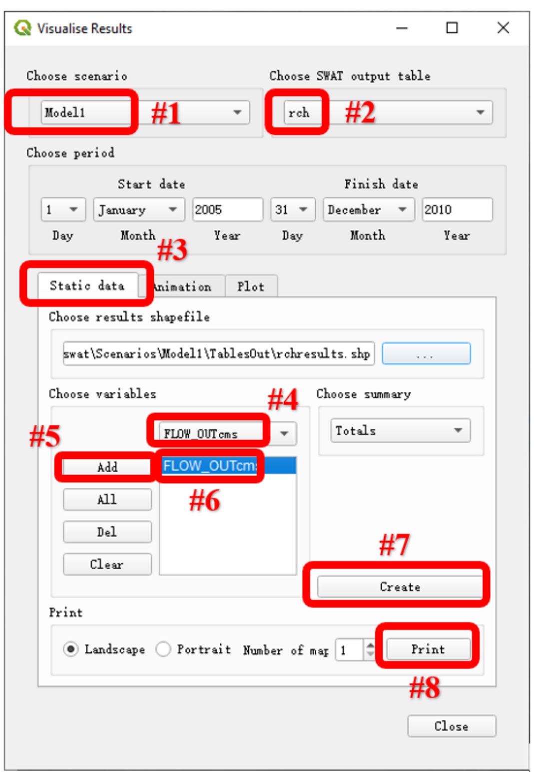


Visualization with QSWAT

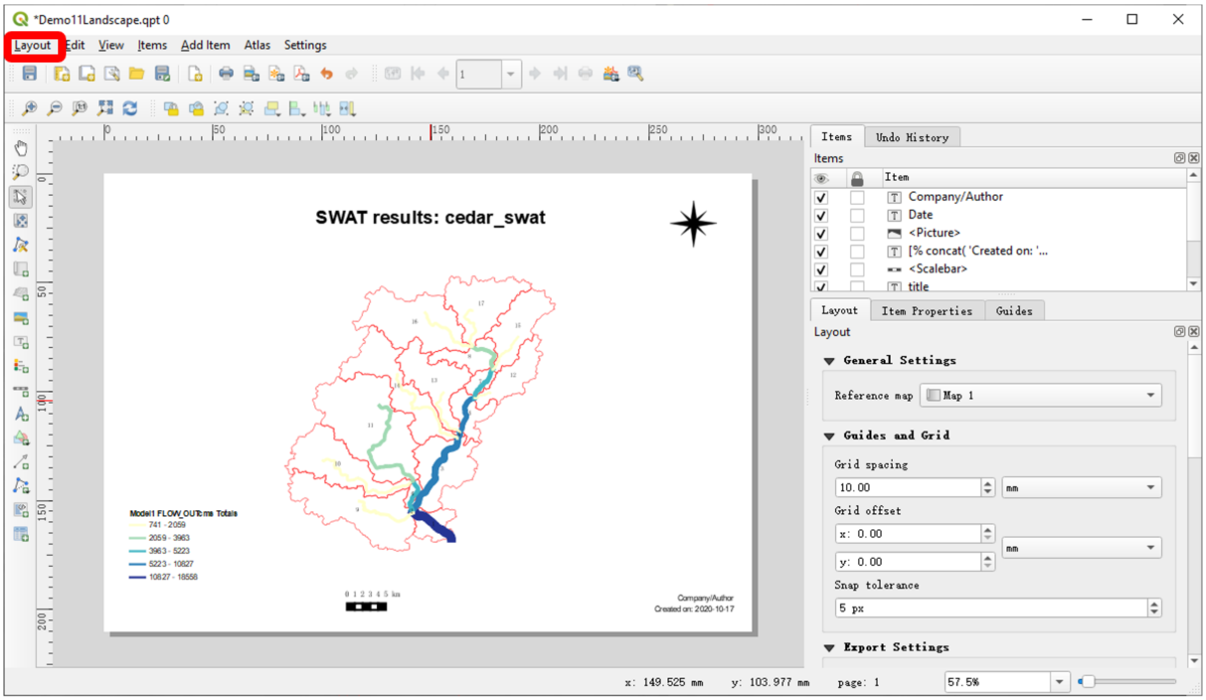
After running the model and error check, Step 4 Visualize will be added to the QSWAT main form. **Click** on *Step 4 – Visualise* in the Main QSWAT form as shown below.



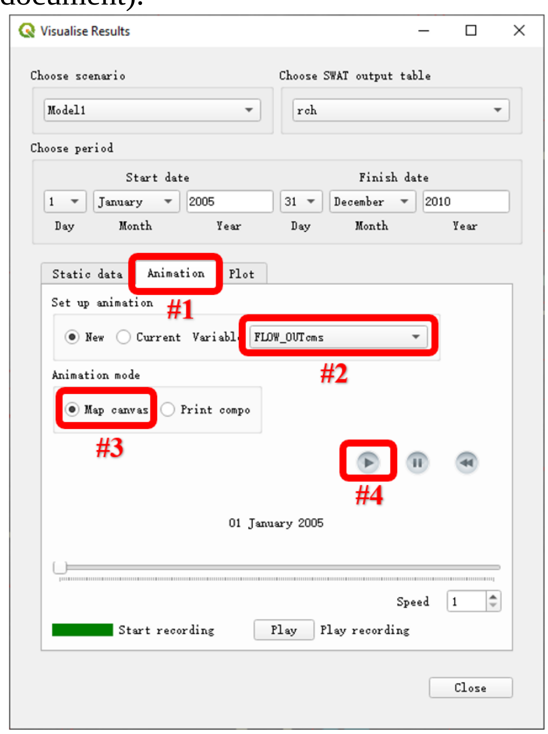
There are three possibilities for visualization: Static data, Animation and Plot. For *Static data* option, a single summary value is calculated for each sub-basin, reach, or HRU and displayed as a map. For *Animation* option, we can view the animated display of the value in each sub-basin, reach, or HRU at each time step in the map display. *Plot* option supports the creation and display of plots of selected variables. First, let us look at *Static data* option.



Follow the steps shown in figure above and finally press on Print option. This will open QGIS Print Composer. This allows us to make a map of the SWAT project. The output for this option is shown below. You can add different features to the map by selecting feature in *Layout* option.

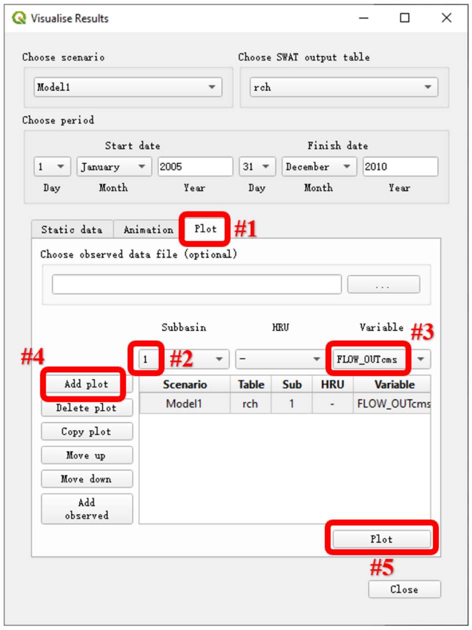


Next, let us use the Animation option to see the animation in the QGIS Desktop (Map canvas/ document).

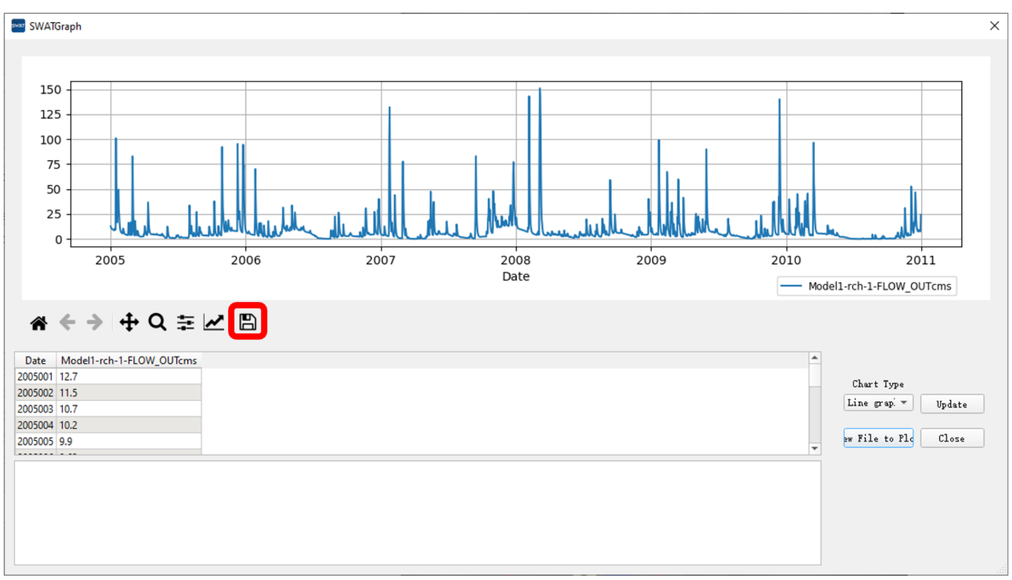


There are controls to run, pause and rewind the simulation: it steps through daily, monthly or annual time steps according to the reporting period chosen for the simulation. The speed of the animation can be adjusted using the Speed option. You can also record the animation. You would see the animation in the map window. You can increase the speed to 10 [if you do not see the changes happening in the map document].

Now, we will use Plot option as shown below.

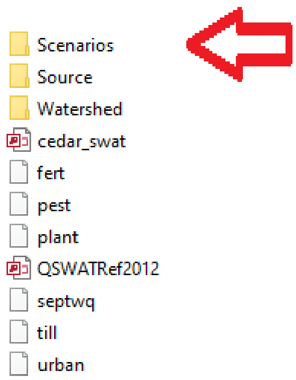


After you press the Plot option, save it as csv file with a suitable name. The graph for the exercise would look similar to below. If you have observed data for your watershed, you can include it in *Choose observed data file* option. Also, you can use the *Save* button to save the plot in different formats.



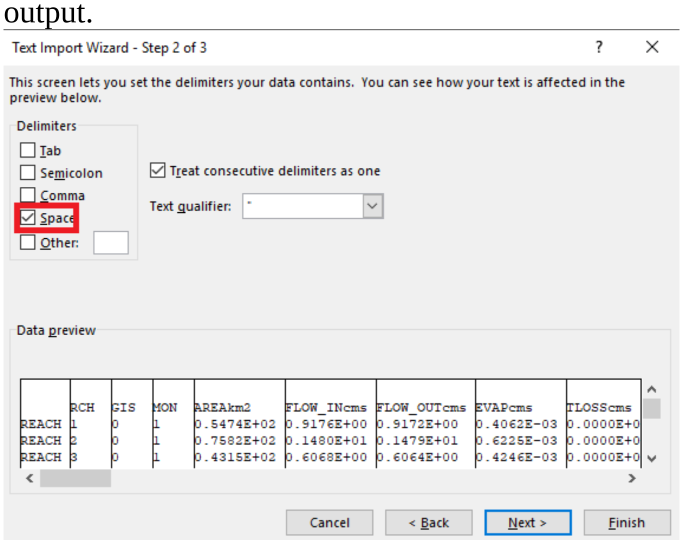
Manual Plotting of SWAT outflow in MS Excel

You can also create a plot by manual method. Find a folder named as ***TxtInOut*** inside the ***Scenarios*** folder, which holds most of the input/output information for the model that you just created. Choose the Model 1 folder (*remember you have saved your first simulation as Model 1 in an earlier step*) in the *Scenarios* folder.



Inside the *TxtInOut* folder, find a file that is called output.rch. Once you know where this file is located, start any spreadsheet software (here we will use MS Excel) and browse to open *output.rch* file. When you try to open the file using MS Excel, make sure you choose "All Files" option and then select *output.rch*.

Open output.rch using "delimited" and "space" as options as shown below. You will then see the output.



Depending on how many reaches (or streams) you have in your model, each reach will have a time series of streamflow. This streamflow is stored in the "FLOW_OUTcms" (flow output in cubic meter per second) column. You will also see a column called "Area_KM2". This column shows the area draining to each reach. The reach that has the maximum area is your most downstream stream. Now, filter your data to only include the rows that has the maximum area. Simply copy the “FLOW_OUTcms”, paste it in another sheet. If you want you can copy the date next to these values as well. Once you have this, you can create a chart of day (x axis) and flow output (y-axis). This will be your hydrograph for the duration of your simulation.

Your plot should have a good title and appropriate labels for x and y axes.

OK, you are done with the QSWAT tutorial! Congratulations!!