# Setting up a SWAT Model with ArcSWAT

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

February 2018

---

## Introduction

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. The objective of this tutorial is to expose users to some of the basic functionalities of ArcSWAT, and how to use it to create a SWAT model for a watershed. This exercise is developed by using data for Cedar Creek in northeast Indiana, but the process can be duplicated for any other watershed in the United States by using a digital elevation model (DEM) and landuse data. For areas, outside the United States soil data will also be needed in addition to DEM and landuse.

## Computer Requirements

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

1. ArcGIS 10.x (with ArcInfo)
2. ArcSWAT 2012

Make sure ArcSWAT 2012 is installed on the system. It can be downloaded from the following link: http://swatmodel.tamu.edu/software/arcswat. You need to have administrative access to install ArcSWAT on your computer.

## Data Requirements and Description

This tutorial requires the following datasets:
(1) DEM for the study area (30 m projected NED DEM)
(2) 2011 NLCD land cover grid from USGS
(3) Soil Data
(4) Outlet point

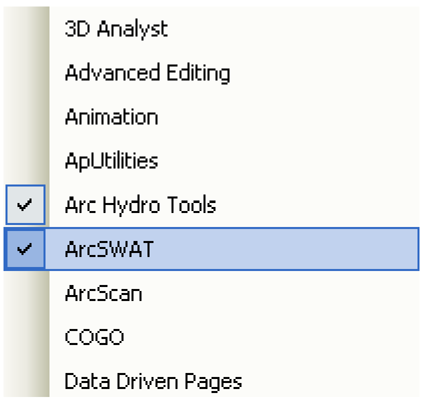
The data are available to Purdue students on blackboard under SWAT folder. Others can get the data from the following link: ftp://ftp.ecn.purdue.edu/vmerwade/download/data/swat.zip. Copy and unzip the data in your project folder.

## Project Folder

(**Note:** This instruction on project folder is only for Purdue students who are doing class project). You will create a folder called Cedar on drive directly, and work from this folder for cedar creek SWAT project. For your project, just create folder called Project when working with your watershed. \\project When opening your project at a later time, open ArcMap and use *Open SWAT Map Document* from the ArcSWAT Toolbar.

## Getting Started

**Open** ArcMap to create an empty document. If ArcSWAT toolbar is not loaded in the map document. **Right click** on the menu bar to see a menu of all the available tools and **select** *ArcSWAT* as shown below.

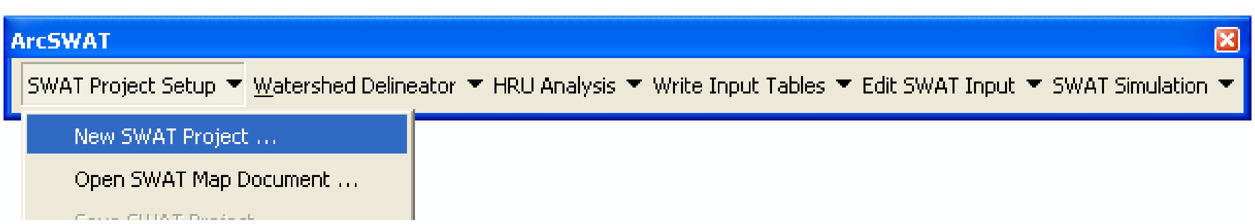



You should now see that the ArcSWAT toolbar added to the map document as shown below. You can leave it floating or you may dock it in one of ArcMap menubars.


Similarly add Spatial Analyst extension and **activate** it by ***clicking*** on *Customize→Extensions…*, and **checking** the box next to *Spatial Analyst*.

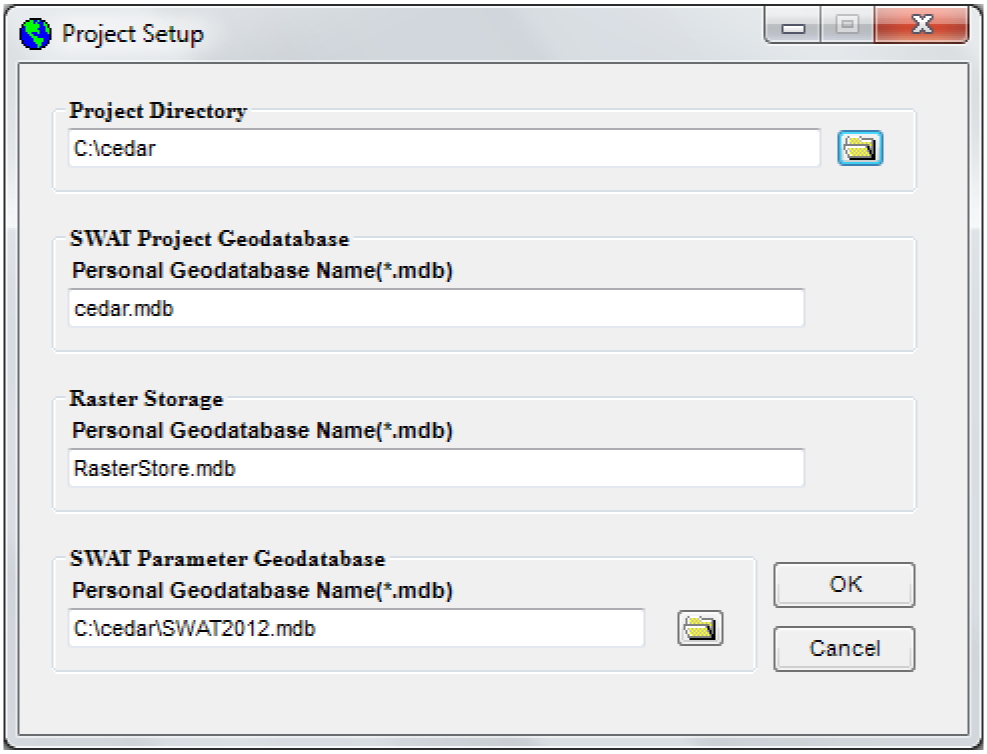
## Project Setup

The first step in using ArcSWAT is to set up a project so that necessary folders and databases are created to store all the data. **Click** on *SWAT Project Setup →New SWAT Project.*




If you get a message asking to save the current map document, **click** No, and proceed.

After few seconds, you should get the *Project Setup* window as shown below. Locate the project directory (Example: E:\cedar or E:\project). The swat project geodatabase, raster storage geodatabase and the SWAT parameter geodatabase automatically get a name. Click *OK*. After few seconds, you should get a project set-up done message. Click *OK* to proceed. If you want to make a SWAT model in your personal computer, it is preferable that you create your project folder in C: drive inside the SWAT folder (where the ArcSWAT has been installed).




The ArcMap document will now be saved with the name of the directory you specified, and *project.mdb* and *RasterStore.mdb* will also be created in your working directory. After the project setup is done, go ahead and start the watershed delineation process.

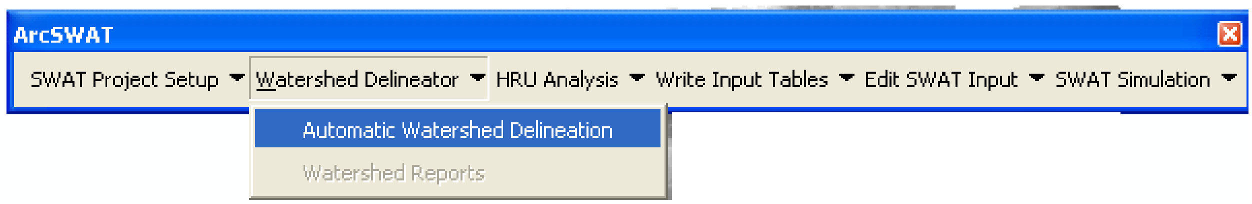
Q1. Go to your project folder and note what new folders or databases are created. Write down the name of these databases and folders. Some folders will also have sub-folders write down their name and the name of the folder under which they are created. In ArcCatalog window, click on the databases and write down what you see inside them.

## Adding Data

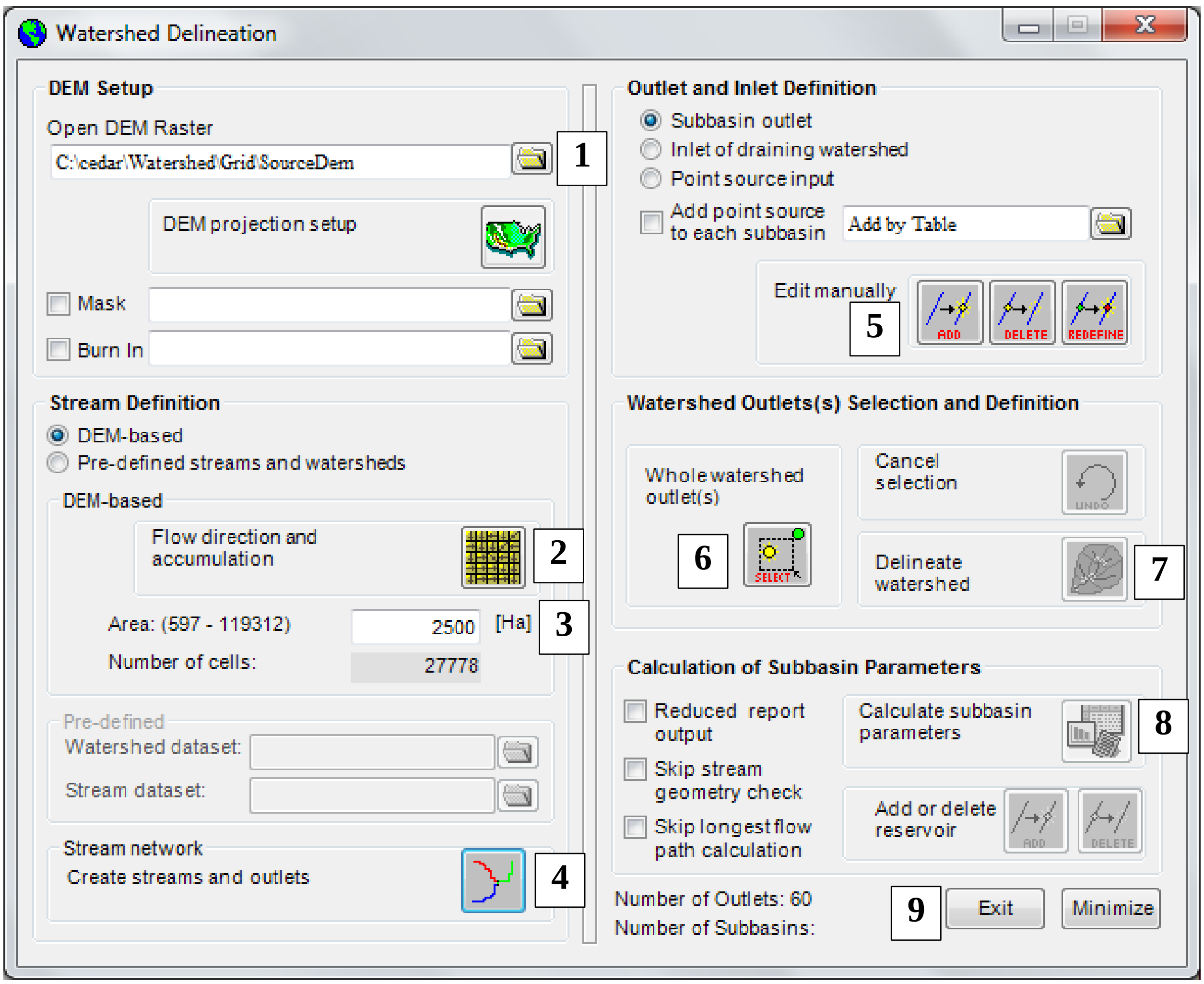
Add the DEM, landuse and the outlet point from your project directory to the map document. All datasets should be in the same projected coordinate system.

## Watershed Delineation

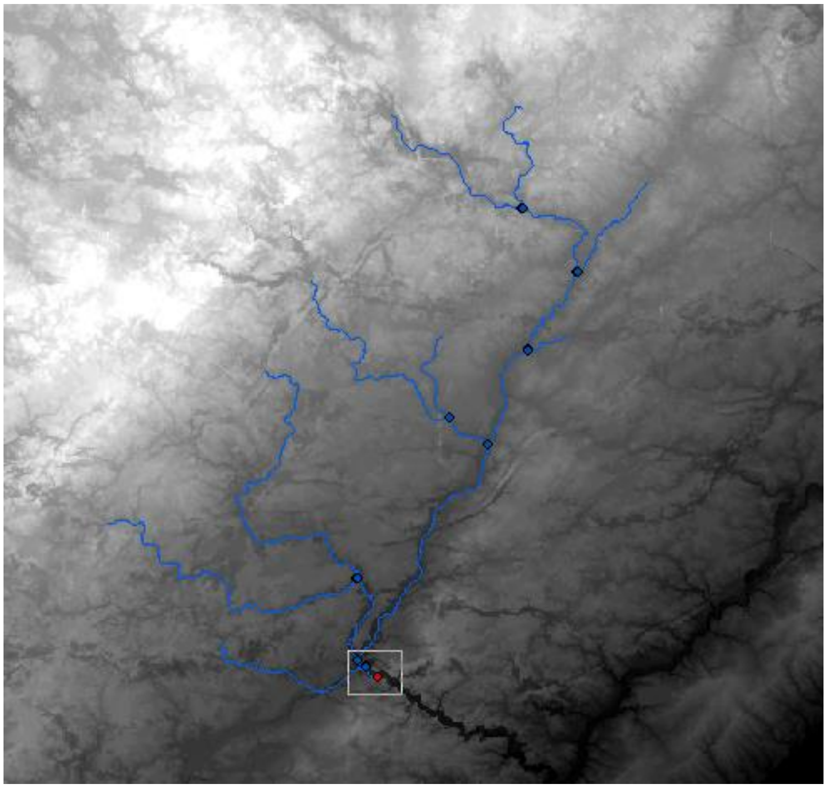
**Click** on *Watershed Delineator→Automatic Watershed Delineation* as shown below.




Once the watershed delineation window is activated, **click** on the open file button (#1 in the figure below) in DEM setup portion, and then **click** on select from map, and then **select** *your DEM*. **Click** *OK*. You will see that the name of the DEM will change to SourceDEM. Lets **skip** the Mask and Burn in option, and proceed to creating flow direction and flow accumulation grids. By leaving the default DEM based option unchanged in the stream definition portion, **click** on *Flow direction and accumulation* button (#2 in figure below).




After flow direction and flow accumulation is complete, the area box will be populated with some number. This is the critical stream area threshold that you will use to define the stream network. We will use 25km$^2$ as the area, which is equivalent to 2500 Ha (#3). Click the *Create stream and outlets* button (#4) to create stream network, subbasins and outlet for each of the subbasin. Once this step is over, just minimize the watershed delineation window and look at the new features (streams and junctions/outlet for each stream segment) added to the map document. Zoom to the outlet point that you added to the map document. If a new point is created very near to the outlet then select *Whole watershed outlet* button (#6) in the watershed outlets selection and definition, and select the point closest to your outlet point using the cursor as shown below. You can use the *undo* button if you have mistakenly selected a wrong outlet. If outlet does not exist at the point you are interested in, use the Add button in *Edit Manually* frame (#5 in the figure above) to add a point, and then do the selection using the outlet (#6) button.



After the outlet is selected, the *Delineate watershed* button (#7) will be activated in green. Click on the delineate watershed button to delineate the watershed. Click OK on the delineation done message. After the watershed is delineated, you will see that a polygon feature class with sub-basins is added to the map document. If you open its attribute table (shown below), you will see that it does not have any hydrology or watershed specific attributes (e.g., slope, elevation).

Attributes of Watershed

| OBJECTID * | Shape * | GRIDCODE | Shape_Length | Shape_Area |
|---|---|---|---|---|
| 1 | Polygon | 1 | 74663.0496 | 73457115.374517 |
| 2 | Polygon | 2 | 49198.679 | 39515607.996882 |
| 3 | Polygon | 3 | 15029.9262 | 6126541.490254 |
| 4 | Polygon | 4 | 51307.194 | 38986550.706045 |
| 5 | Polygon | 5 | 55848.61 | 41792600.237912 |
| 6 | Polygon | 6 | 77366.2746 | 50881569.971676 |

Record: 1 Show: All Selected Records (0

The final step in watershed delineation is to generate parameters/attributes for all the sub-basins. Click on the *Calculate subbasin parameters* button (#8) to populate the watershed feature class with necessary attributes for SWAT model. Be patient because this process takes some time. After all attributes are calculated, click OK on the final message box. Now open the attribute table (if you closed it), and you will see that it has more attributes as shown below.

Attributes of Watershed

| Area | Slo1 | Len1 | Sll | Csl | Wid1 | Dep1 | Lat | Long_ | Elev | ElevMin | ElevMax | Bnan |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 7345.711537 | 3.342777 | 21895.898887 | 91.435538 | 0.222579 | 16.990652 | 0.725032 | 41.466005 | -85.131138 | 293.664687 | 266 | 323 | <Null> |
| 3951.5608 | 2.37553 | 16038.985457 | 91.435538 | 0.255894 | 11.712535 | 0.565784 | 41.478138 | -85.040948 | 289.991919 | 268 | 311 | <Null> |
| 612.654149 | 2.509297 | 4519.681509 | 91.435538 | 0.48587 | 3.827533 | 0.268429 | 41.439979 | -85.023491 | 276.781489 | 266 | 289 | <Null> |
| 3898.655071 | 1.634681 | 15356.949917 | 121.914051 | 0.184492 | 11.618193 | 0.562742 | 41.470505 | -84.965914 | 279.208258 | 266 | 302 | <Null> |
| 4179.260024 | 2.152634 | 16217.974585 | 91.435538 | 0.238122 | 12.112934 | 0.578606 | 41.396055 | -85.09709 | 276.795286 | 260 | 301 | <Null> |

Record: 1 Show: All Selected Records (0 out of 15 Selected) Options

Close the attribute table. You are done with delineating the watershed and creating sub-basins for a specific threshold area. Exit the watershed delineation window by clicking on Exit (#9). After you exit, the program will do some more processing, cleaning-up of data, etc. so wait until all this is done. Save the map document.

The image below shows the product of watershed delineation step using ArcSWAT for Cedar Creek dataset.

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

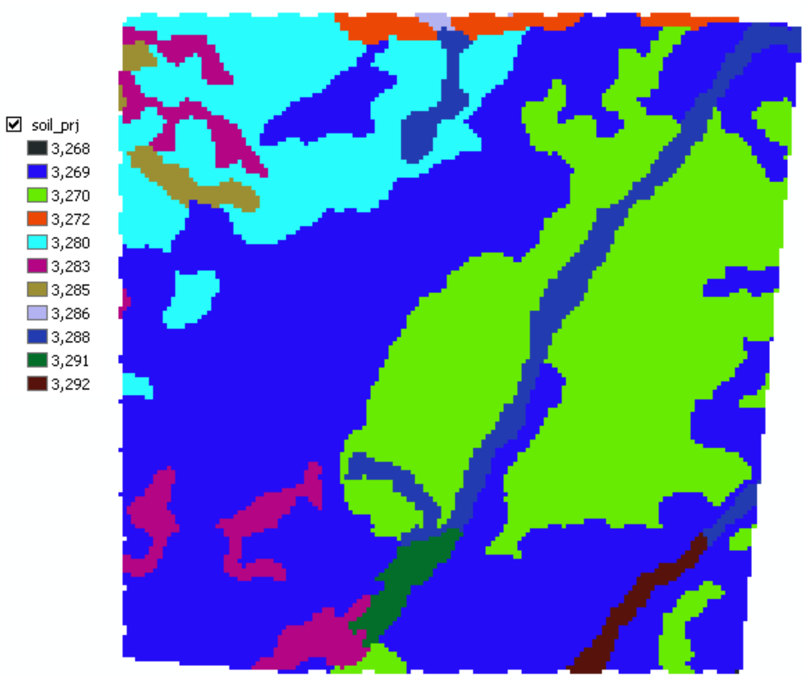
Q2. What changes do you see in all the databases and folders in your project directory. Go back to your answer to Q1 and write down the changes you see.

## HRU Analysis

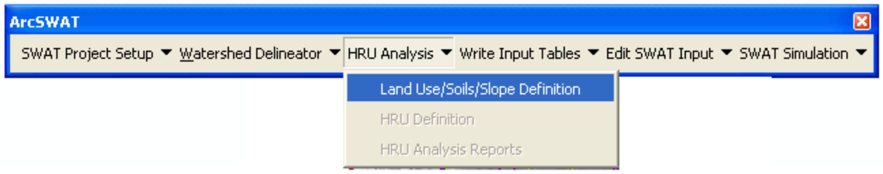
### <u>Preparing STATSGO Soil Data</u>

When you install ArcSWAT, a raster with STATSGO soil data for the entire United States is stored where your ArcSWAT is installed. Typically at \ArcSWAT\Databases\SWAT_US_Soils.mdb. Browse to this mdb file, and add the raster (named as **stastgo_grd**) to the map document. Next, you will clip this raster to the DEM for the study area. Clipping is done by using ArcToolbox in ArcMap. In ArcToolbox, go to Spatial Analyst Tools→Extraction→Extract by Mask. Use your <u>original DEM (SourceDEM)</u> as the Mask during this clipping. Use a logical name to save the output from this step (e.g., soil_clip). In order to use this clipped soil raster, it should have the same coordinate system as the other data in your map document. The soil grid has geographic coordinates so go ahead and project this raster by using ArcToolbox. In ArcToolbox, go to Data Management Tool→Projections and Transformations→Raster→Project Raster. You can import the projection from one of the existing layers in the map document or the original DEM. Name the output as soil_prj.

For Cedar Creek, soil_prj looks like the following:

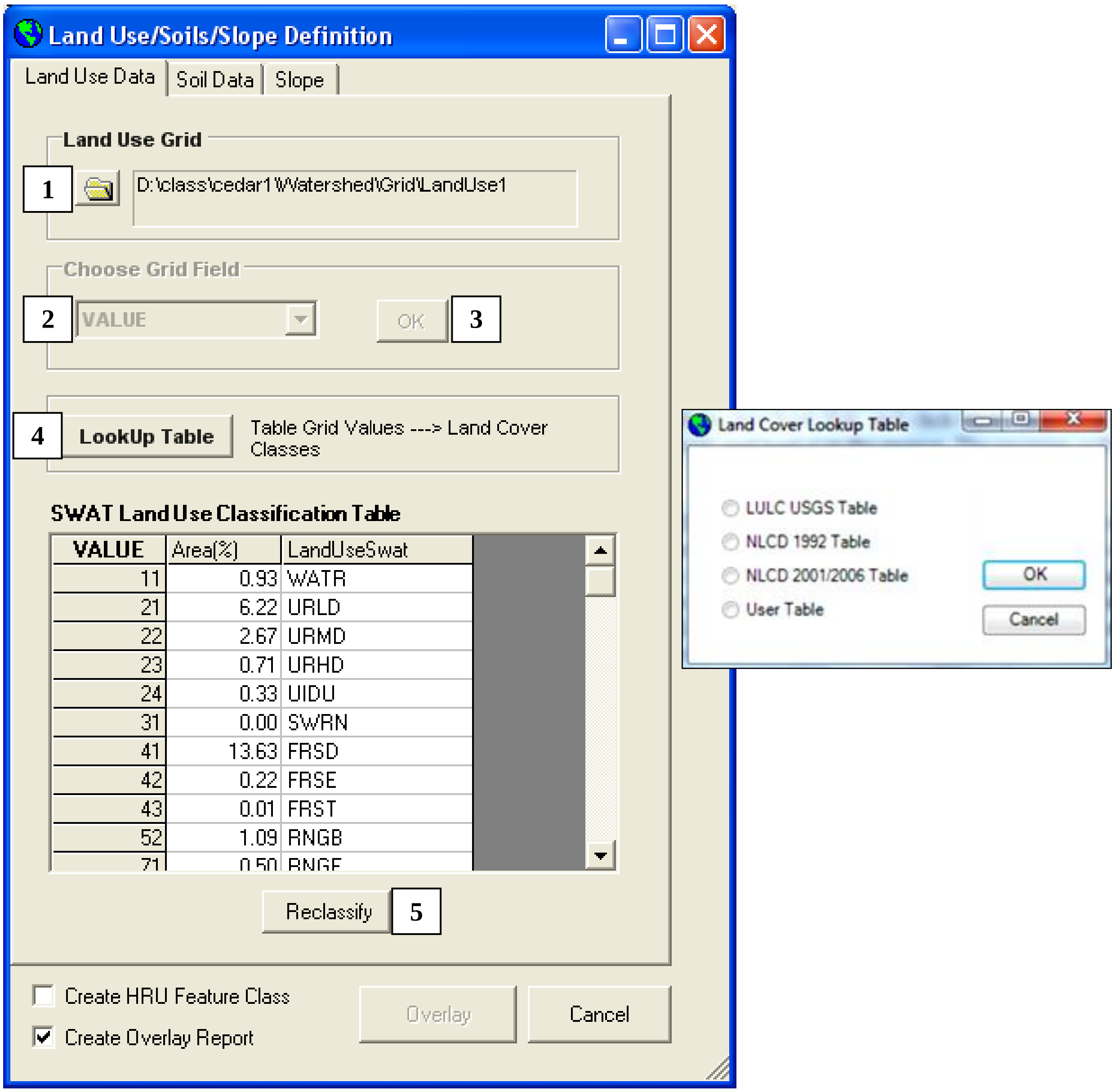



With the soil analysis done, we are now ready for HRU analysis. On the ArcSwat toolbar, click on *HRU Analysis→Land Use/Soils/Slope Definition* as shown below.




The landuse/soils/slope definition window has three tabs for land use, soil and slope each. Lets work with land use first. Click on the open file button (#1 in the figure below) to load the land use data. In the next window, select land use data from map (assuming your projected landuse is added to the map document), and click Open. If your land use is not on the map, browse to the projected landuse raster in your working directory and load it in the project. The program will clip the data before loading it in the project.

Next, select *VALUE* for the choose grid field (#2), and click *OK* (#3). 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. Click on *LookUp Table* (#4), and select *NLCD 2001/2006*. Even though we are using NLCD 2011 data, the landuse description has not changed between the two, so choosing NLCD 2006 for LookUp table is fine. This will create a description for each land use number in the SWAT land use classification table. Finally, click on *Reclassify* (#5) to finish the land use data processing portion of HRU analysis.

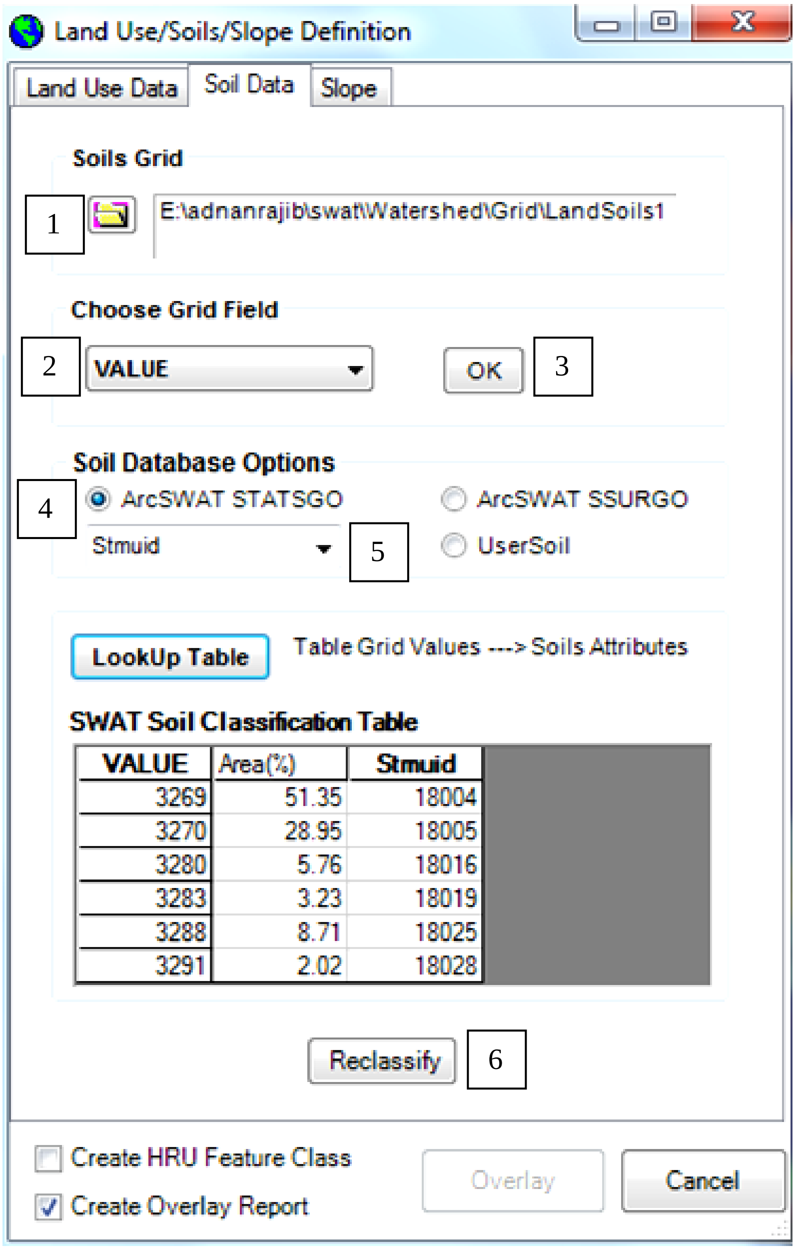



Q3. Have your ever classified a raster before? Explain what reclassification does to a raster.

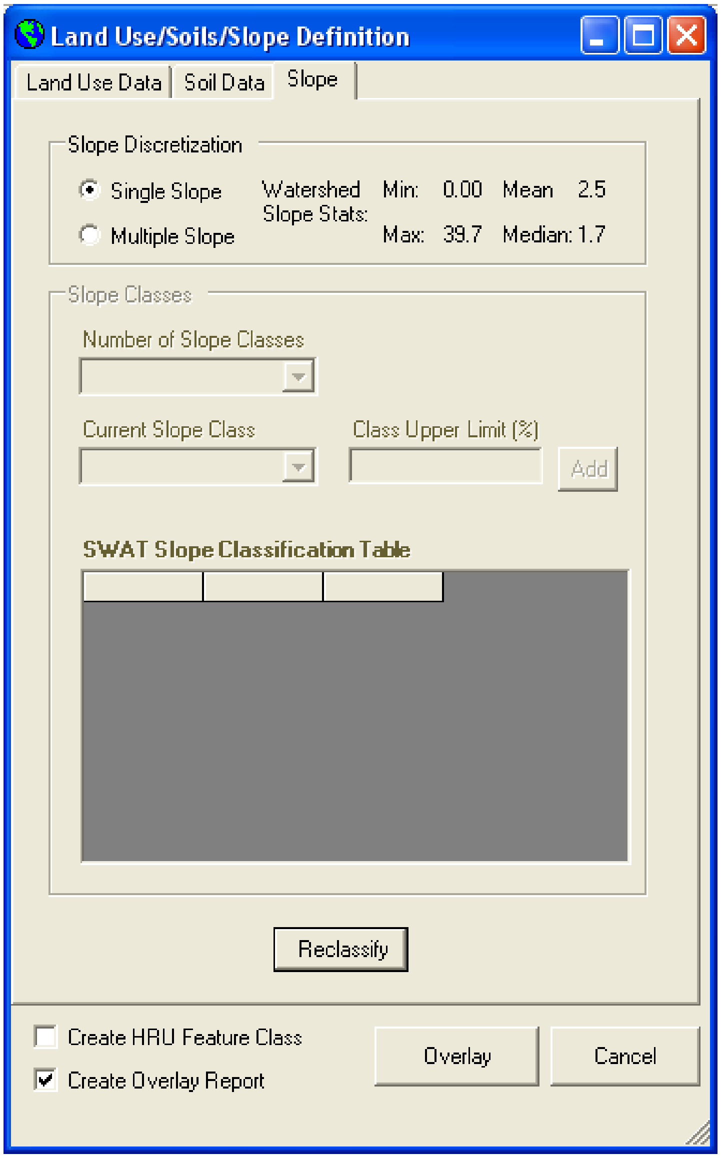
After processing land use data, click on the Soil Data tab.

To process soil data, click on the open Soils Grid button (#1), load soils dataset from disk, and browse to your clipped projected soil raster that you just created, and click Select. This step will take your input raster, and then clip that raster to the watershed area. This process sometimes takes longer depending on the size of your watershed so please be patient.

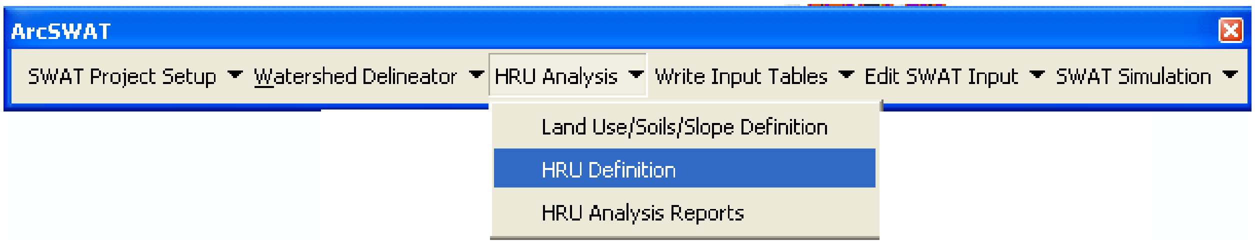
After the soil data are loaded, choose *VALUE* (#2) as the grid field, and click *OK*(#3). This will add all numerical values present in the landsoil dataset to the SWAT soil classification table. You will see that for each value there is an area associated with each soil type. The next step is to link these numerical values to a certain soil type. This is done by done by using either a lookup table or using one of the STATSGO attributes. In order to use the lookup table, it is necessary to define what attribute in the lookup table corresponds to the soil type. In the options frame, click on *ArcSWAT STATSGO* (#4) and then *Stmuid* (#5) from the drop down list. This will add Stmuid field to the SWAT soils classification table and the corresponding Stmuid. Finally, **click** on *Reclassify* (#6). This will finish the soil data processing for HRU analysis. In ArcSWAT2012, you can also work with SSURGO soil database.



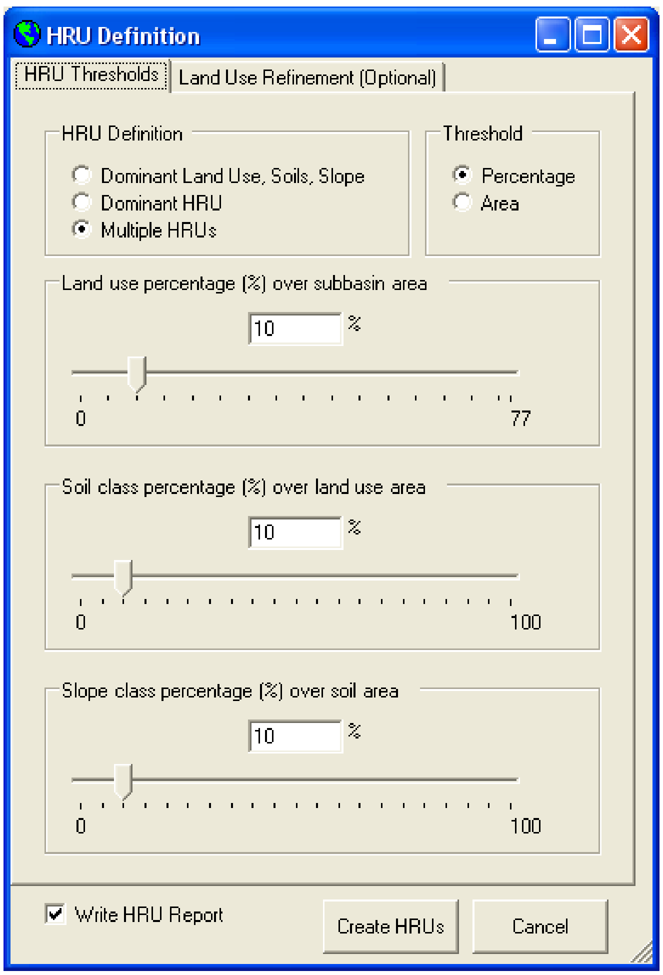

After soil data processing, the next step is to assign slope attributes to each HRU. Click on the *Slope* tab. To keep the process simple (and also considering that Cedar Creek is a relatively flat region), use the single slope option in the soil descritization frame, and click on *Reclassify*. If you are working in a mountainous watershed, use multiple slope option. This will complete the processing of landuse, soil and slope data for HRU analysis. Finally, click on Overlay button to create combined information on land use, soil type and slope, which will then be used to create HRU based on the threshold specified by the user. Close the Landuse/soil/slope definition window.



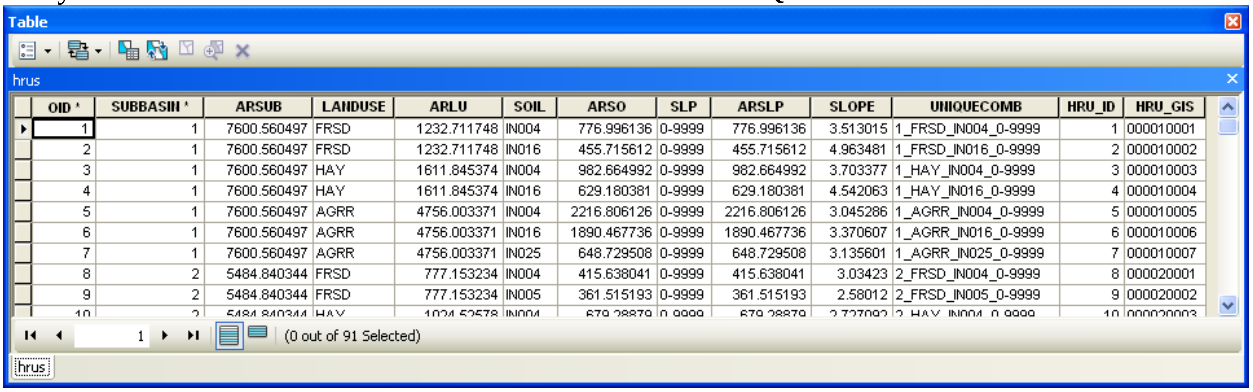

To create HRU by using the combined land use, soil and slope data, click on *HRU Analysis* →*HRU Definition*




Leave the default options of *Multiple HRUs* in HRU Definition frame, and Percentage in the Threshold frame. For this exercise, use a threshold value of 10% for land use, soil and slope, and click Create HRUs. Note that because we are using a single slope value, the threshold for slope does not really matters in this exercise.




A table called hrus will be added to the map document, and if you open that table, you will see that several (around 100) HRUs with unique combination of landuse, soil and slope are created for the study area. These combinations are listed in the UNIQUECOMB field as shown below.



| OID * | SUBBASIN * | ARSUB | LANDUSE | ARLU | SOIL | ARSO | SLP | ARSLP | SLOPE | UNIQUECOMB | HRU_ID | HRU_GIS |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | 7600.560497 | FRSD | 1232.711748 | IN004 | 776.996136 | 0-9999 | 776.996136 | 3.513015 | 1_FRSD_IN004_0-9999 | 1 | 000010001 |
| 2 | 1 | 7600.560497 | FRSD | 1232.711748 | IN016 | 455.715612 | 0-9999 | 455.715612 | 4.963481 | 1_FRSD_IN016_0-9999 | 2 | 000010002 |
| 3 | 1 | 7600.560497 | HAY | 1611.845374 | IN004 | 982.664992 | 0-9999 | 982.664992 | 3.703377 | 1_HAY_IN004_0-9999 | 3 | 000010003 |
| 4 | 1 | 7600.560497 | HAY | 1611.845374 | IN016 | 629.180381 | 0-9999 | 629.180381 | 4.542063 | 1_HAY_IN016_0-9999 | 4 | 000010004 |
| 5 | 1 | 7600.560497 | AGRR | 4756.003371 | IN004 | 2216.806126 | 0-9999 | 2216.806126 | 3.045286 | 1_AGRR_IN004_0-9999 | 5 | 000010005 |
| 6 | 1 | 7600.560497 | AGRR | 4756.003371 | IN016 | 1890.467736 | 0-9999 | 1890.467736 | 3.370607 | 1_AGRR_IN016_0-9999 | 6 | 000010006 |
| 7 | 1 | 7600.560497 | AGRR | 4756.003371 | IN025 | 648.729508 | 0-9999 | 648.729508 | 3.135601 | 1_AGRR_IN025_0-9999 | 7 | 000010007 |
| 8 | 2 | 5484.840344 | FRSD | 777.153234 | IN004 | 415.638041 | 0-9999 | 415.638041 | 3.03423 | 2_FRSD_IN004_0-9999 | 8 | 000020001 |
| 9 | 2 | 5484.840344 | FRSD | 777.153234 | IN005 | 361.515193 | 0-9999 | 361.515193 | 2.58012 | 2_FRSD_IN005_0-9999 | 9 | 000020002 |
| 10 | 2 | 5484.840344 | HAY | 1024.52578 | IN004 | 679.28879 | 0-9999 | 679.28879 | 2.727092 | 2_HAY_IN004_0-9999 | 10 | 000020003 |



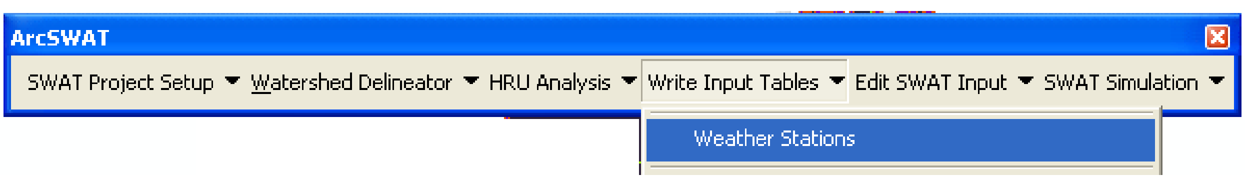
If you want to learn more about the HRUs that are created and their combination, you can select HRU Analysis→HRU Analysis Reports look at land use, soil and slop distribution and the final HRU distribution.

## Creating Input Files/Tables

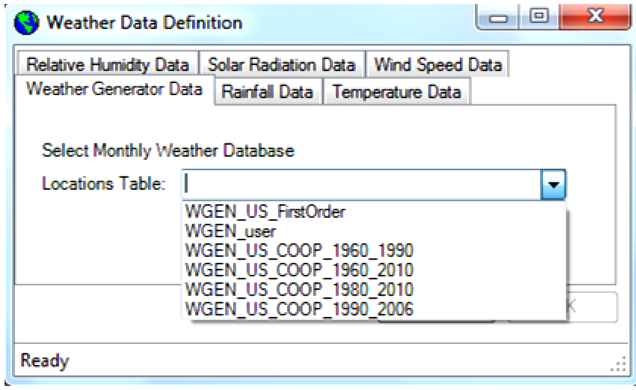
After all geoprocessing is done on DEM, land use, and slope data to create sub-basins and HRUs, the next step is to create input files for SWAT including weather data.

### *Weather Data (using weather generator – not needed for this class)*

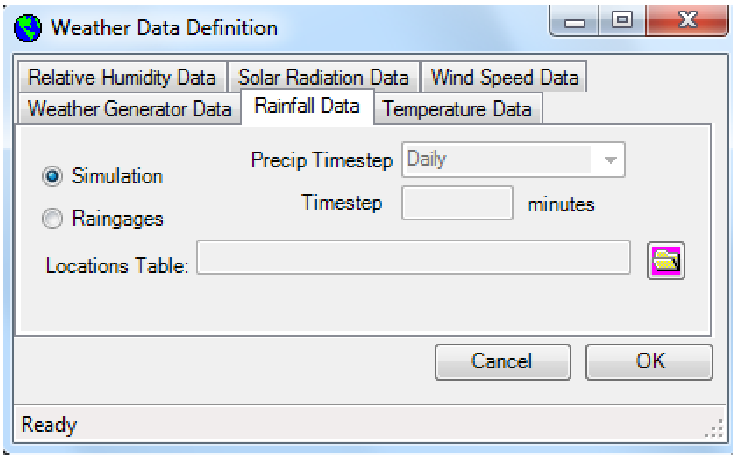
To create weather input for SWAT, click on *Write Input Tables*→*Weather Stations*




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. Selecting simulation option means that the data will be generated through the internal weather generator within SWAT. Similarly for 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.

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

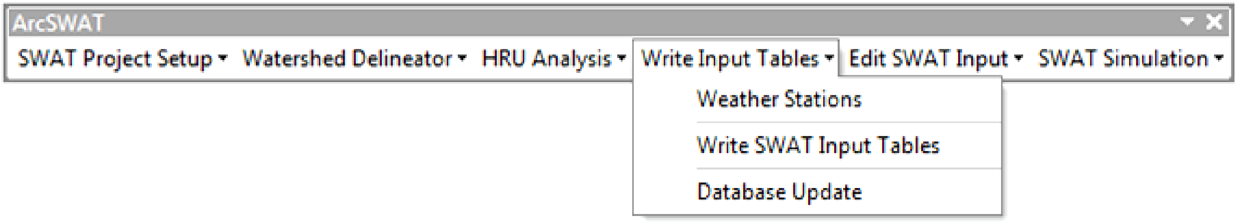
To use observed rainfall data, click on Rainfall Data, check the Raingages option, and browse to the pcp file. Click OK.

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

All other files will be used from SWAT weather generator.

***Other Input Files***

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



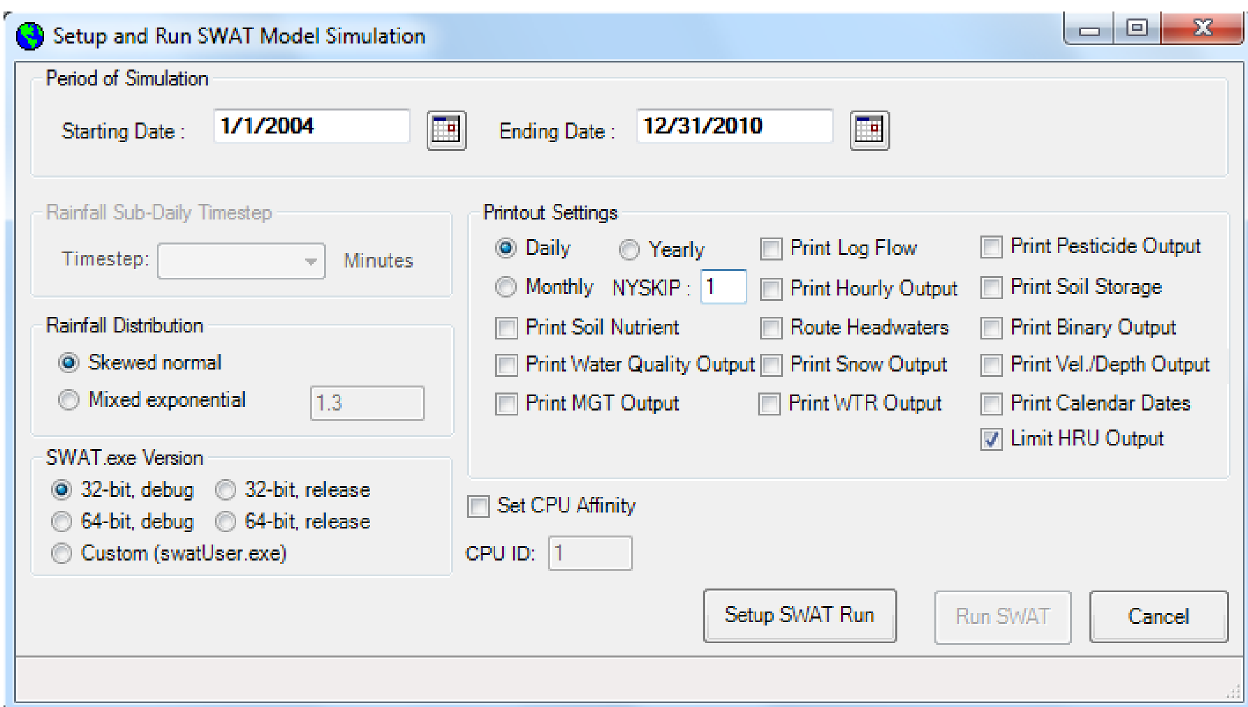

A new window will pop up named as 'Write SWAT Database Tables'. There, click on *Select All* and then click on *Create Tables*.

If you get a message asking about default Manning's n, choose Yes or OK, and proceed. Similarly, 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!

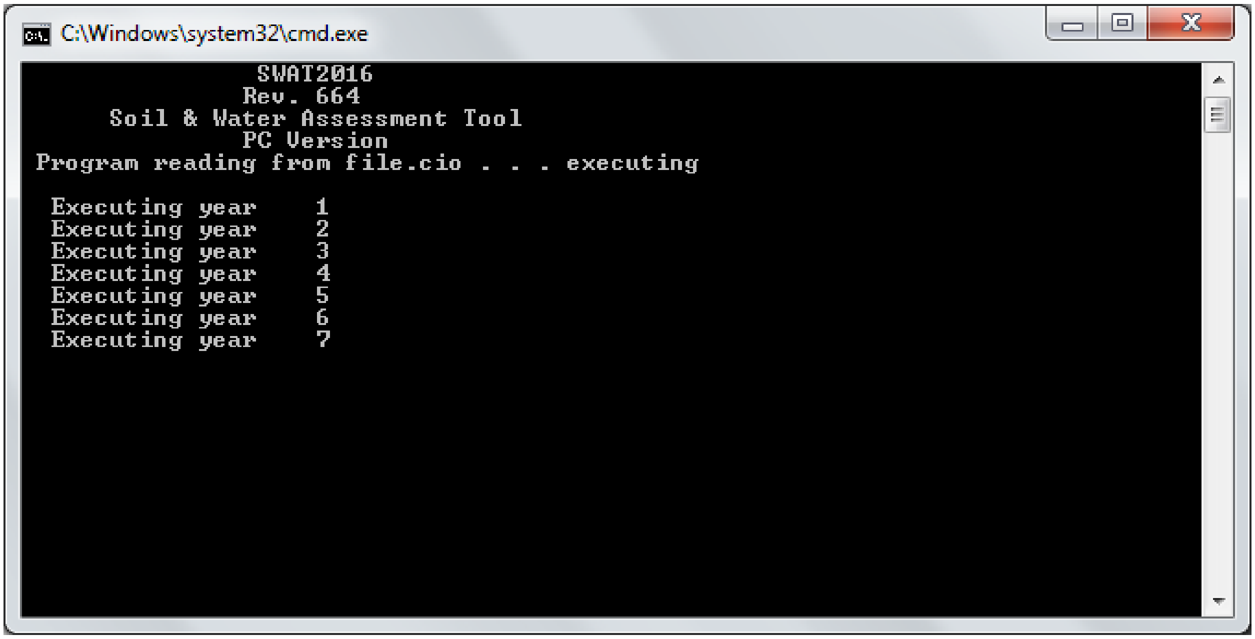
## Running SWAT Simulation

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 in this class we will put NYSKIP=1. Click on *Setup SWAT Run*. After getting the finished swat setup message, click on *Run SWAT*.



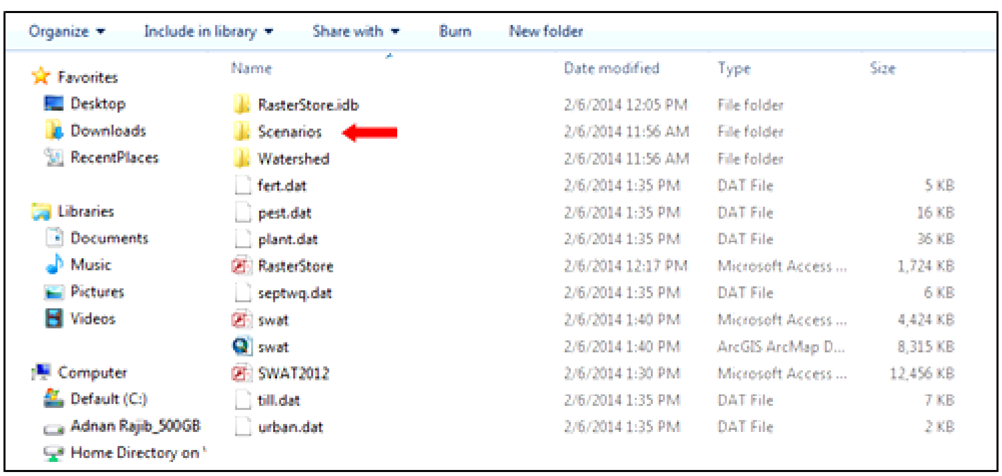
While SWAT is running, you will see a DOS window showing you the progress, and after the simulation is done, you will get a final simulation done message.




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

## Plotting SWAT outflow in Excel

When you are done with running your SWAT model, your project folder will contain the following items as seen in the figure below. Find a folder called ***TxtInOut*** inside the ***Scenarios*** folder, which holds most of the input/output information for the model that you just created. TxtInOut may or may not be immediately inside Scenarios so just look for it.



Inside the TxtInOut folder, find a file that is called output.rch. Once you know where this file is located, start Excel and open outut.rch. When you try to open the file using Excel, make sure you choose "All Files" option and then select output.rch.

Open output.rch using "delimited" and "space" as options. 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!