13. Combining Landscape Segmentation and Agroecosystem Simulation Model

Xiuying Wang*, Pushpa Tuppad, and Jimmy Williams

Appendix 13

In this exercise, the APEX version APEX0806 and ArcGIS version 10.0 with build 10.0.5.4400 can be used. APEX requires the following input data: site information and watershed characteristics, weather (user input or model generated weather), land use, management such as cropping system, planting and harvesting, tillage, fertilizer and/or pesticide applications, and soil properties by layer. GIS-based user interface can automate the input parameterization. In this exercise, the ArcAPEX interface integrates topographic, land use, and soil spatial datasets and a built-in APEX-parameters database that contains model parameter values required to simulate a wide range of plant growth, tillage, fertilizer, and pesticide applications. In general, within the USA, the topography data can be obtained from the U.S. Geological Survey (USGS) National Elevation Dataset at http://ned.usgs.gov; landuse/land cover can be obtained from the Multi-Resolution Land Characteristics (MRLC) Consortium which developed the National Land Cover Dataset (NLCD) at www.mrlc.gov; soil data can be obtain from the Soil Survey Geographic (SSURGO) or Soil State Soil Geographic (STATSGO) Databases, or USDA-NRCS Soil Data Mart at http://soildatamart.nrcs.usda.gov; and weather data can be obtain from National Climatic Data Center’s National Weather Service (NCDC-NWS) at http://www.ncdc.noaa.gov/oa/climate/stationlocator.html. All the GIS layers should be projected to a common system. In this exercise, all data has been projected to Clarke_1866_Albers (after downloading and installing ArcAPEX, example datasets (e.g., DEM, landuse and soil) can be found in the folder: …ArcAPEX\Databases\ExampleApex).

Download ArcAPEX for the appropriate ArcGIS version you are using from http://apex.tamu.edu/software/. Install as instructed.

A.13.1. Create ArcAPEX Project

This step is designed to create an ArcAPEX project to help in preparing APEX inputs. The project file contains links to your retrieved data and incorporates all customized GIS functions into your project file. The major steps on how to create an APEX project are listed below:
Step 1.1. Start ArcMap. For the first use, the user should turn on the “ArcAPEX” by clicking the Extensions button under the “Tools” menu and go to the “View” menu, click the Toolbars and then click ArcAPEX toolbar to open the main interface of ArcAPEX.

Step 1.2. Create an APEX project and set up the working directory and geodatabases by clicking the New APEX Project button under the APEX Project Setup menu and selecting a Project Directory for the current project. Then the names of APEX Project Geodatabase and Raster Storage Geodatabase are initialized automatically (Figure A-1).

![APEX Project Setup](image)

Figure A-1. Create an APEX project and set up working directory and geodatabases

A.13.2. Sub-area Delineation

In the APEX simulation, a watershed can be divided into one or more sub-areas. A sub-area is conceptually equivalent to a field or landscape unit with homogeneous weather inputs, land cover, vegetation, soils, and agronomic practices. In this respect, an APEX sub-area is functionally equivalent to a SWAT Hydrologic Response Unit (HRU). In addition, each subarea is associated with a channel for routing runoff, sediment, nutrients, and pesticides from one subarea to another. With respect to defining watershed connectivity, APEX sub-areas are functionally equivalent to sub-basins in the SWAT model. Delineation of APEX sub-areas, channels, and sub-area connectivity is the first step in the development of an APEX model project.

Step 2.1. Activate the Automatic Subarea Delineation button under the APEX Subarea Delineation menu and then select APEX Standalone Delineation. Add DEM Grid, click on Load from Disk and select OK. Click yes when asked if the grid or shapefile is projected. Example files were loaded with the installation of APEX. For this example, select the DEM grid located in your disk from “C:\ProgramFiles\ArcAPEX\Databases\ExampleApex”. Check the projection information of the DEM (Figure A-2) by clicking the USA map button. The DEM properties should correctly define the grid size and units. The user can change the Z unit to Meters.
Step 2.2. Mask on Watershed Area to specify the watershed area of interest by loading the mask from disk, in this example from “C:\ProgramFiles\ArcAPEX\Databases\ExampleApex”. The file name of the mask is “amask.”

Step 2.3. Stream Definition. Here choose the DEM-based option button to begin stream delineation; click to calculate Flow Direction and Accumulation. After setting the threshold value of the sub-area, the user can then delineate the stream network and outlets. The threshold area defines the drainage area required to form the beginning of a stream. A small threshold area will produce a very detailed stream network, whereas a larger threshold area will produce fewer streams. Users can start with the default value and rerun the step by adjusting the threshold area accordingly. The drainage network and stream juncture points are displayed on the DEM map grid (Figure A-3).
Step 2.4. Outlet and Inlet Definition, which provides a chance for the user to change the threshold area and rerun the stream and outlet definition routine, add outlet points by importing a table that contains the locations, add outlet points manually, remove outlet points, and/or add point source inputs.

Sub-area outlets represent points where one stream empties into another or is a point which is being monitored and validation data is available. Adding outlets at the location of monitoring stations is useful for comparison of measured and predicted flows and concentrations. Removing outlet points will reduce the number of sub-areas. Try changing the threshold area up or down to see the effect of the sub-area configuration. Higher threshold values will result in a smaller number of sub-areas, while a smaller threshold value will result in a larger number of sub-areas (detailed stream network). If there are point sources input or upstream watershed drained into this study watershed, then inlet/s should be added. Inlets represent any point source loading into the study area or the inlets of drainage into the watershed from an upstream area.

Step 2.5. Main Watershed Outlet(s) Selection and Definition.
In this step the user will select one outlet location to define the boundary of the main watershed. Note that by holding the SHIFT key in your keyboard you can select more than one outlet. If the user selects two watershed outlets, then there will be two watersheds delineated. This feature allows adjacent watersheds to be simulated at the same time using APEX. Do not select an outlet at the upstream of another outlet. At least one outlet must be selected for delineation.
Click on the **SELECT** button to choose the watershed outlet. Draw a box covering the desired outlet location to set the main **Watershed Outlet** (Figure A-3) and then click the **Delineate Watershed** button. The delineated watershed with sub-areas will be added to the map. Click on the **Calculate Subarea Parameters button** to estimate the sub-area parameters. This function calculates basic watershed characteristics from the DEM and sub-area layer. Click **OK** to complete the watershed delineation dialog box. Figure A-4 shows the delineated watershed with sub-areas.

![Figure A-4. Delineated watershed and sub-areas](image)

**Step 2.6.** Exit the Subarea Watershed Delineation Dialog by clicking **EXIT** in the sub-area delineation main dialog. You can save the project at this point by clicking **Save APEX Project** item under the **APEX Project Setup** menu.

Open the **Watershed Reports** by clicking the **Watershed Reports** item under the **APEX Subarea Delineation** menu. The topographic report contains a statistical summary and area and percentage wise distribution of elevation for the entire watershed and each subarea.

A.13.3. Sub-area Analysis

The **Land Use, Soil and Slope Definition** option in the **Subarea Analysis** menu allows the user to specify the land use, soil and slope GIS datasets that will be used for APEX modeling. APEX requires land use data to determine the area of each land category to be simulated within each sub-area. In addition to land use information, APEX relies on soil data to determine the range of hydrologic characteristics found within each sub-area. The **Land Use, Soil and Slope Definition** option guides the user through the process of specifying the data to be used in the simulation and ensuring that those data are in the appropriate format. In particular, the option allows the user to select land use or soil data that are in either shape or grid format. The datasets can be ESRI grid, shapefile, or geodatabase feature class format. Shapefiles are automatically converted to grid, the
format required by ArcGIS to calculate land use and soil distributions within the subareas of interest.

**Step 3.1.** Select the **Land Use/Soil/Slope Definition** option from the **Subarea Analysis** menu. Click on the folder button. Then click on the radial button next to **Load Land Use dataset(s) from disk.** Then click **Open.** Click yes when it asks you if the grid or shapefile is projected. Select the land use grid file from the “C:\Program files\ArcAPEX\Databases\ExampleApex\” folder. After loading the Land use dataset into the map, choose the grid field which will be used as an index to define different land use types. In this example, the “Value” field is selected. Click **OK,** then a table titled “**Land Use Classification Table**” will be created automatically by the interface. The first column contains the unique values in the **Grid Field** chosen above. The second column contains the area of each soil type. The soil map grid must be linked to the U.S. soils database (provided with the interface) or to the User database. Under “Options,” click the button beside the method used to link the soils map grids to the soils data. Choose **Soils (customized)**. Click **OK,** a table titled “**Soil Classification Table**” will be created automatically by the interface. The first column contains the unique values in the **Grid Field** chosen above. The second column contains the area of each soil type. The soil map grid must be linked to the U.S. soils database (provided with the interface) or to the User Soils (customized) database. Under “Options,” click the button beside the method used to link the soils map grids to the soils data. Choose **Stmuid.** The third column contains the soil names in the APEX database corresponding to each index value.

In order to fill correct values in the third column, the land use grid codes must be assigned a land cover/plant description. You may import a look-up table or manually assign a land cover/plant code. The interface includes tables that convert the USGS or NLCD land use/land cover classification codes to APEX land cover/plant codes. If the land use grid being used is classified by an alternate method, you must create a look-up table or enter the information manually.

Select the land use look-up table. A dialog will appear that will ask you which land cover lookup table to use with your land use grid. There are four choices. The LULC USGS Table option will load the USGS LULC classifications. The NLCD 1992 or 2001 table will load the NLCD 1992 or 2001 classifications. The User Table option will open a file browse dialog to select a user-defined lookup table. Users may refer to the USGS table, nlcd_lu, and nlcd2001_lu tables in the APEX.mdb database to review APEX land cover codes for the USGS LULC and NLCD 1992/2001 classifications respectively. Since the dataset being used has a user-defined lookup table, the “**User Table**” option should be selected. The user lookup table is located in: “C:\Program Files\ArcAPEX\Databases\ExampleApex\luc.txt”. After clicking **OK,** the APEX Land use Classification Table will be populated. Click the **Reclassify** button. Then Land use map will be reclassified according the Lookup table.

**Step 3.2.** Click on the “**Soil Data**” tab at the top of the active screen. Click on the folder button, and then click on the radial button next to **Load Soils dataset(s) from disk.** Then click **Open.** Click yes when it asks you if the grid or shapefile is projected. Select the soils data layer from “C:\Program Files\ArcAPEX\Databases\ExampleApex\soil”. Choose the field “**Value**” to define different soil types. Click **OK,** a table titled “**Soil Classification Table**” will be created automatically by the interface. The first column contains the unique values in the **Grid Field** chosen above. The second column contains the area of each soil type. The soil map grid must be linked to the U.S. soils database (provided with the interface) or to the User Soils (customized) database. Under “Options,” click the button beside the method used to link the soils map grids to the soils data. Choose **Stmuid.** The third column contains the soil names in the APEX database corresponding to each index value, which need to be filled through a Lookup table. Click **Lookup Table** and browse to the “C:\Program Files\ArcAPEX\Databases\ExampleApex\soilc.txt” lookup table. To map the soils according to their STMUID, click the **Reclassify** button for soils grid.
Step 3.3. Click the “Slope” tab. There are two options for slope discretization: “Single Slope” and “Multiple Slope”. The “Single Slope” option indicates that sub-areas in the watershed will not be differentiated on the basis of slope, while the “Multiple Slope” option will allow sub-areas to be differentiated by two or more user-defined slope classes. Differentiating sub-areas by slope classes will allow users to display/map modeling results by slope classes. Note that no matter how many slope classes are used, in APEX input file, one sub-area has only one slope value which is either the dominant slope or the slope associated with the Dominant Land Use/Soil/Slope Combination. The “Multiple Slope” option is only for differentiating sub-areas within the watershed. You can get a better idea of how this works by looking at a Reclassified slope grid map. In this example, we will use 2 slope classes with a break in classes of 1%. So, select multiple slopes and “2” for the Number of Slope Classes. To define the first slope class, select the Current Slope Class as “1”. Set the Class Upper Limit as “1%”, then click Add. The Slope Classification Table will be populated with Lower Limit and Upper Limit of each slope class. Click Reclassify to generate the slope grid and add it to the map.

Step 3.4. When the land use, soil and slope layers have all been classified, the OVERLAY button will be activated. Click this button; all three grid layers will be overlaid.

Before executing APEX, the distribution of subareas within the watershed must be determined based on the land use, soil and slope layers specified in the previous step. Subdividing the watershed into areas having unique land use, soil and slope combinations enables the model to reflect differences in evapotranspiration for various crops and soils. Runoff, sediment, nutrient, and pesticide loss is predicted separately for each subarea and routed to obtain the total runoff for the watershed. This increases accuracy and gives a much better physical description of the water balance.

Step 3.5. Click the Subarea Definition button under the Subarea Analysis menu. For this example, select “Dominant land use, Dominant soil, Dominant slope” and click OK. By using this option, ArcAPEX will analyze the land use, soil and slope independently of each other and assign the dominant land use, dominant soil and dominant slope for each sub-area. The “Dominant Land Use/Soil/Slope Combination” option will analyze each land use-soil-slope combination and select the dominant combination and assign it to the entire subarea. The user is also allowed to select the land use, soil and slope for each of the sub-areas. When edits are made, the user must click Update Subarea after each subarea is modified. Once all subareas are edited and updated, click the OK button to exit.

A.13.4. Write Input Files for APEX

This menu contains functions to build database files that include information needed to generate default input for the APEX model. The commands on the menu need to be implemented only once for a project. However, if the user modifies the subarea distribution after building the input database files, these commands must be reprocessed again.

Users can also modify APEX input files manually. The variables and their definitions in each input file are provided in the APEX User’s Manual which can be downloaded from http://epicapex.tamu.edu/downloads/user-manuals/.
Edit Apex database.
The Edit APEX Databases selection under the APEX Input Files menu can be used to make input modifications during the model calibration process to the databases containing specific data used for running the model. You may contact Evelyn Steglich at steglich@brc.tamus.edu to request an ArcAPEX training manual where a general procedure is provided to familiarize you with the APEX input files and editing capabilities in ArcAPEX.

Step 4.1. Define Weather data by selecting the Define Weather Stations button under the APEX Input Files menu. Select the “US Database” option for Weather Generator Data for both the Monthly Weather Statistics and the Monthly Wind Statistics. Under “Observed Weather Data” select “Observed Precipitation” and “Observed Temperature”. Then select the Weather station list text file from “C:\ProgramFiles\ArcAPEX\Databases\ExampleApex\WDLSTCOM.TXT.” Click OK.

APEX’s database contains monthly weather statistics for about 1050 weather stations in the United States. If a long-term daily weather series is not available to generate the monthly weather statistics, the APEX program can reference the latitude and longitude of the simulation site to find the closest weather station in the database and use its monthly weather statistics. Variables in the monthly weather statistics are used to estimate missing daily data or generate daily weather variables (precipitation, maximum and minimum air temperatures, solar radiation, and relative humidity). Other uses include: the monthly temperatures for calculating potential heat units (PHU, thermal time) for tree crops; the maximum 0.5-h rainfall for estimating peak runoff rate and rainfall energy. The monthly wind weather statistics include average monthly wind speed and monthly wind percentage from 16 wind directions, which are critical if simulating wind erosion, as well as dust distribution and air quality from feedlots. These variables are used to generate daily wind speed and direction.

Step 4.2. Select Point Source Discharges under APEX Input Files if there is/are point sources input or upstream watershed drained into this study watershed. Any point source loading into the study area or the inlets of drainage into the watershed from an upstream area can be added here. In this exercise, there are no point sources in this watershed.

Step 4.3. Select Write Default Inputs under APEX Input Files. The user can choose to only write certain input files or write all the input files by clicking the SELECT ALL button. Once the desired input files have been selected click OK. For this example, select all files to be written. Then click EXIT.

APEX contains multiple options for simulating processes, such as hydrology and soil erosion–sedimentation. Therefore, users may prefer one over the other depending on data availability and geographic or agricultural settings. The default inputs include default options in the APEXCONT file and default output options in the APEXPRNT file. Users can edit these defaults by selecting Edit APEX Inputs under APEX Input Files menu.

Step 4.4. Write APEX Input Files.
After changes have been made to any of the APEX database files, the files must be rewritten for the changes to take effect. Select **Write APEX Input Files** under **APEX Input Files**. A “Write APEX Input Files” dialog will open. To rewrite an input table select the input table name and click **OK**. To select several tables, hold down the Control key and select the desired tables. To rewrite all of the input tables, click **SELECT ALL** and then **OK**.

A.13.5. Run APEX Model

**Step 5.1.** Before making any further refinements to our model parameterization, let’s take some time to make our first APEX run using the default parameterization. Select the **Make APEX Model Run** command under the **Run APEX Model** menu. A default APEX run is listed under **APEX Runs**. In this exercise, the watershed was delineated into 14 sub-areas. The watershed’s main outlet is located in the end of the reach in sub-area 14 (Figure A-5), therefore the default APEX run was name as SITE14. You may want to enter a meaningful run name in the ASTN field. This provides a unique ID for each run so that the output files are not overwritten. Select a site from the ISIT field. Select a monthly weather station number. This data will be used to generate weather parameters if daily weather is not specified or if specific daily weather parameters are not input. Select a wind station number from the list. Wind Station number must be one of the stations in the WINDUS.dat file; if left blank, APEX will use the lat and long from ISIT to choose a station. Select the subarea file number. This file contains the subarea data for each of the subareas in the watershed. Click **ADD** to add the new APEX run and it will appear to the **APEX Runs** list.

**Step 5.2.** Select the run in the “APEX Runs” box and click **RUN APEX**. And then the user can choose **Save APEX Model Output**. This dialog allows you to import APEX output files into an Access database and view APEX output in the database.
**Step 5.3. View APEX Output.** We will now look at the contents of the APEX output database. Open up the Access database called **APEXOutput.mdb**. If you have saved the simulation under Test1 as showed in Figure A-6, the mdb file is in the \Test1\TablesOut folder. You will find the output tables from the model run (rch and sub). The variables in .rch file are formatted the same as in the SWAT .rch file. There is an APEX checker program similar to the SWAT checker program. Currently the model website is under construction. You can check the website at
http://epicapex.tamu.edu/ to see if the APEX check program is downloadable from there. If not, then you can contact Jaehak Jeong at jieong@brc.tamus.edu to request it. More resources for the APEX model, such as upcoming ArcAPEX Training Course Events by Texas A&M University, APEX User’s Manual, Theoretical Documentation, and so on, can be found from http://apex.tamu.edu/

Users can refer to the following publications about the general rules for APEX calibration, APEX and SWAT integration, APEX sensitivity analysis, and the ArcAPEX interface.


