SWMM EXERCISE

CEEGR 4800/6965 - Special Topics
Geographic Information Systems and Hydrologic & Hydraulic Modeling

1.1 SWMM Exercise

This exercise is the initial part of the tutorial example included with SWMM 5 up to "Viewing a Profile Plot" step of the tutorial. Thus, you can also do this exercise by following the steps in the tutorial which can be accessed using one of the following methods:

- Open Tutorial.chm file from C:\Program Files\EPA SWMM 5.0, or
- Open Chapter 2 (Quick Start Tutorial) of the Users Manual
- Select Tutorial from SWMM Help menu

Caution: To avoid losing your input data entered during the exercise due to computer problems, repeatedly save the exercise file (tutorial.inp). Save the file on your student (N) drive or your portable drive (e.g., memory stick). The first save should be done using File → Save As menu option. Subsequent saves should be done using File → Save menu option. This approach will allow you to complete the exercise in several sessions (e.g., some steps during the class and other steps later).

2.1 Example Study Area

In this tutorial we will model the drainage system serving a 12-acre residential area. The system layout is shown in Figure 2-1 and consists of subcatchment areas[1] S1 through S3, storm sewer conduits C1 through C4, and conduit junctions J1 through J4. The system discharges to a creek at the point labeled Out1. We will first go through the steps of creating the objects shown in this diagram on SWMM's study area map and setting the various properties of these objects. Then we will simulate the water quantity and quality response to a 3-inch, 6-hour rainfall event, as well as a continuous, multi-year rainfall record.

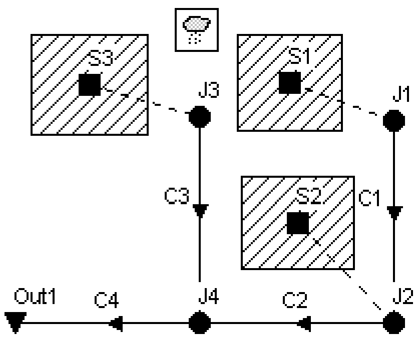


Figure 2-1. Example study area.

2.2 Project Setup

Our first task is to create a new SWMM project and make sure that certain default options are selected. Using these defaults will simplify the data entry tasks later on.

1. Launch EPA SWMM software from Start → All Programs → EPASWMM 5.0
2. Select **File** >> **New** from the Main Menu bar to create a new project.
3. Select **Project** >> **Defaults** to open the Project Defaults dialog.
4. On the ID Labels page of the dialog, set the ID Prefixes as shown in Figure 2-2. This will make SWMM automatically label new objects with consecutive numbers following the designated prefix.

[1] A subcatchment is an area of land containing a mix of pervious and impervious surfaces whose runoff drains to a common outlet point, which could be either a node of the drainage network or another subcatchment.

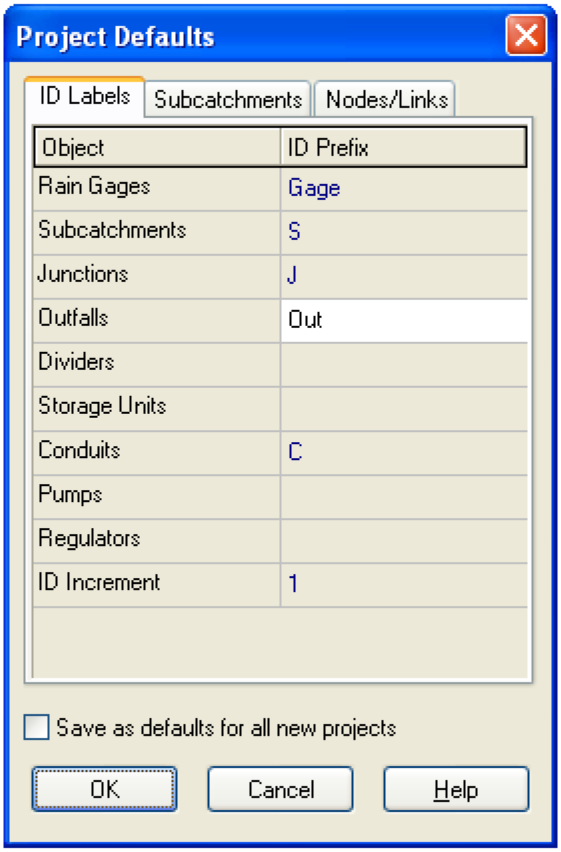


Figure 2-2. Default ID labeling for tutorial example.

5. On the Subcatchments page of the dialog set the following default values:

Area	4
Width	400
% Slope	0.5
% Imperv.	50
N-Imperv.	0.01
N-Perv.	0.10
Dstore-Imperv.	0.05
Dstore-Perv	0.05
%Zero-Imperv.	25
Infil. Model	<click to edit>
- *Method*	Green-Ampt
- *Suction Head*	3.5
- *Conductivity*	0.5
- *Initial Deficit*	0.26

6. On the Nodes/Links page set the following default values:

Node Invert	0
Node Max. Depth	4
Flow Units	CFS

Conduit Length	400
Conduit Geometry	<click to edit>
- *Barrels*	1
- *Shape*	Circular
- *Max. Depth*	1.0
Conduit Roughness	0.01
Routing Model	Kinematic Wave

7. Click **OK** to accept these choices and close the dialog. If you wanted to save these choices for all future new projects you could check the **Save** box at the bottom of the form before accepting it.

Next we will set some map display options so that ID labels and symbols will be displayed as we add objects to the study area map, and links will have direction arrows.

1. Select **View** >> **Map Options** to bring up the Map Options dialog (see Figure 2-3).
2. Select the Subcatchments page, set the Fill Style to Diagonal and the Symbol Size to **5**.
3. Then select the Nodes page and set the Node Size to **5**.
4. Select the Annotation page and check off the boxes that will display ID labels for Areas, Nodes, and Links. Leave the others un-checked.
5. Finally, select the Flow Arrows page, select the Filled arrow style, and set the arrow size to **7**.
6. Click the **OK** button to accept these choices and close the dialog.

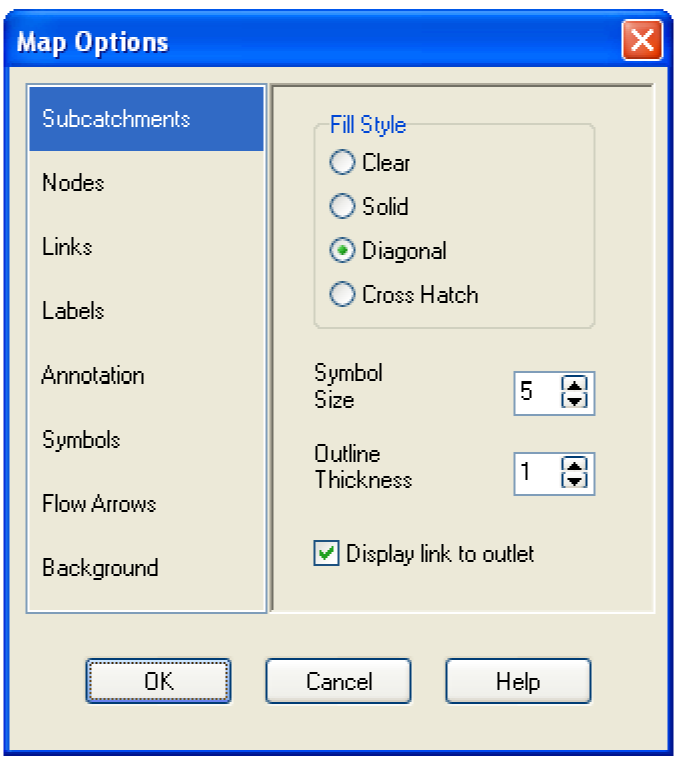


Figure 2-3. Map Options dialog.

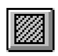
Before placing objects on the map we should set its dimensions.

1. Select **View >> Dimensions** to bring up the Map Dimensions dialog.
2. You can leave the dimensions at their default values for this example.

Finally, look in the status bar at the bottom of the main window and check that the Auto-Length feature is off. If it is on, then right-click over the status bar and select “Auto-Length Off” from the popup menu that appears.

2.3 Drawing Objects

We are now ready to begin adding components to the Study Area Map[2]. We will start with the subcatchments.

1. Begin by clicking the  button on the Object Toolbar. (If the toolbar is not visible then select **View >> Toolbars >> Object**). Notice how the mouse cursor changes shape to a pencil.
2. Move the mouse to the map location where one of the corners of subcatchment **S1** lies and left-click the mouse.
3. Do the same for the next tthree corners and then right-click the mouse (or hit the Enter key) to close up the rectangle that represents subcatchment **S1**. You can press the **Esc** key if instead you wanted to cancel your partially drawn subcatchment and start over again. Don't worry if the shape or position of the object isn't quite right. We will go back later and show how to fix this.
4. Repeat this process for subcatchments **S2** and **S3**[3].

Observe how sequential ID labels are generated automatically as we add objects to the map.

Next we will add in the junction nodes and the outfall node that comprise part of the drainage network.

1. To begin adding junctions, click the  button on the Object Toolbar.
2. Move the mouse to the position of junction **J1** and left-click it. Do the same for junctions **J2** through **J4**.

[2] Drawing objects on the map is just one way of creating a project. For large projects it might be more convenient to first construct a SWMM project file external to the program. The project file is a text file that describes each object in a specified format as described in Appendix D of this manual. Data extracted from various sources, such as CAD drawings or GIS files, can be used to create the project file.

[3] If you right-click (or press **Enter**) after adding the first point of a subcatchment's outline, the subcatchment will be shown as just a single point.

3. To add the outfall node, click the button on the Object Toolbar, move the mouse to the outfall's location on the map, and left-click. Note how the outfall was automatically given the name **Out1**.

At this point your map should look something like that shown in Figure 2.4.

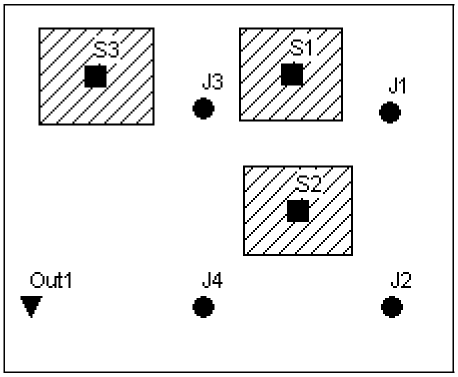


Figure 2-4. Subcatchments and nodes for example study area.

Now we will add the storm sewer conduits that connect our drainage system nodes to one another. (You must have created a link's end nodes as described previously before you can create the link.) We will begin with conduit **C1,** which connects junction **J1** to **J2**.

1. Click the button on the Object Toolbar. The mouse cursor changes shape to a crosshair.
2. Click the mouse on junction **J1**. Note how the mouse cursor changes shape to a pencil.
3. Move the mouse over to junction **J2** (note how an outline of the conduit is drawn as you move the mouse) and left-click to create the conduit. You could have cancelled the operation by either right clicking or by hitting the <**Esc**> key.
4. Repeat this procedure for conduits **C2** through **C4**.

Although all of our conduits were drawn as straight lines, it is possible to draw a curved link by left-clicking at intermediate points where the direction of the link changes before clicking on the end node.

To complete the construction of our study area schematic we need to add a rain gage.

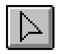
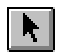
1. Click the Rain Gage button on the Object Toolbar.
2. Move the mouse over the Study Area Map to where the gage should be located and left-click the mouse.

At this point we have completed drawing the example study area. Your system should look like the one in Figure 2.1. If a rain gage, subcatchment or node is out of position you can move it by doing the following:

1. If the button is not already depressed, click it to place the map in Object Selection mode.

2. Click on the object to be moved.
3. Drag the object with the left mouse button held down to its new position.

To re-shape a subcatchment's outline:

1. With the map in Object Selection mode, click on the subcatchment's centroid (indicated by a solid square within the subcatchment) to select it.
2. Then click the  button on the Map Toolbar to put the map into Vertex Selection mode.
3. Select a vertex point on the subcatchment outline by clicking on it (note how the selected vertex is indicated by a filled solid square).
4. Drag the vertex to its new position with the left mouse button held down.
5. If need be, vertices can be added or deleted from the outline by right-clicking the mouse and selecting the appropriate option from the popup menu that appears.
6. When finished, click the  button to return to Object Selection mode.

This same procedure can also be used to re-shape a link.

2.4 Setting Object Properties

As visual objects are added to our project, SWMM assigns them a default set of properties. To change the value of a specific property for an object we must select the object into the Property Editor (see Figure 2-5). There are several different ways to do this. If the Editor is already visible, then you can simply click on the object or select it from the Data page of the Browser Panel of the main window. If the Editor is not visible then you can make it appear by one of the following actions:

- double-click the object on the map,
- or right-click on the object and select Properties from the pop-up menu that appears,
- or select the object from the Data page of the Browser panel and then click the Browser's  button.

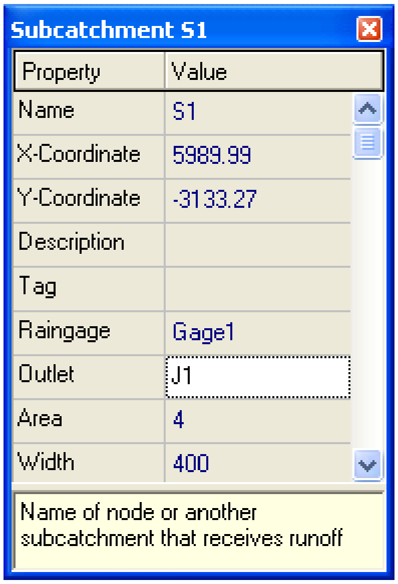


Figure 2-5. Property Editor window.

Whenever the Property Editor has the focus you can press the F1 key to obtain a more detailed description of the properties listed.

Two key properties of our subcatchments that need to be set are the rain gage that supplies rainfall data to the subcatchment and the node of the drainage system that receives runoff from the subcatchment. Since all of our subcatchments utilize the same rain gage, **Gage1**, we can use a shortcut method to set this property for all subcatchments at once:

1. From the main menu select **Edit >>Select All**.
2. Then select **Edit** >> **Group Edit** to make a Group Editor dialog appear (see Figure 2-6).
3. Select **Subcatchment** as the type of object to edit, **Rain Gage** as the property to edit, and type in **Gage1** as the new value.
4. Click **OK** to change the rain gage of all subcatchments to **Gage1**. A confirmation dialog will appear noting that 3 subcatchments have changed. Select "**No**" when asked to continue editing.

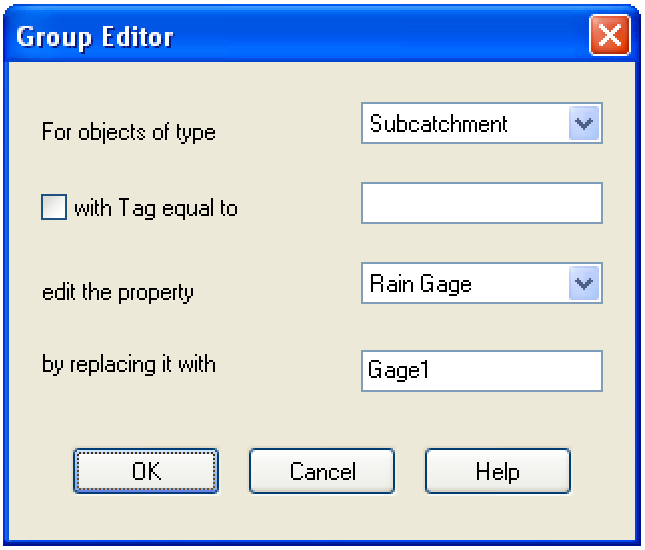


Figure 2-6. Group Editor dialog.

Because the outlet nodes vary by subcatchment, we must set them individually as follows:

1. Double click on subcatchment **S1** or select it from the Data Browser and click the Browser's button to bring up the Property Editor.
2. Type **J1** in the Outlet field and press **Enter**. Note how a dotted line is drawn between the subcatchment and the node.
3. Click on subcatchment **S2** and enter **J2** as its Outlet.
4. Click on subcatchment **S3** and enter **J3** as its Outlet.

We also wish to represent area **S3** as being less developed than the others. Select **S3** into the Property Editor and set its % Imperviousness to **25**.

The junctions and outfall of our drainage system need to have invert elevations assigned to them. As we did with the subcatchments, select each junction individually into the Property Editor and set its Invert Elevation to the value shown below[4].

Node	*Invert*
J1	96
J2	90
J3	93
J4	88
Out1	85

Only one of the conduits in our example system has a non-default property value. This is conduit **C4**, the outlet pipe, whose diameter should be 1.5 instead of 1 ft. To change its diameter:

[4] An alternative way to move from one object of a given type to the next in order (or to the previous one) in the Property Editor is to hit the Page Down (or Page Up) key.

1. Select conduit **C4** into the Property Editor (by either double-clicking anywhere on the conduit itself or by selecting it in the Data Browser and clicking the button).
2. Select the Shape field and click the ellipsis button (or press <**Enter**>).
3. In the Cross-Section Editor dialog that appears (see Figure 2-7), set the Max. Depth value to **1.5** and then click the **OK** button.

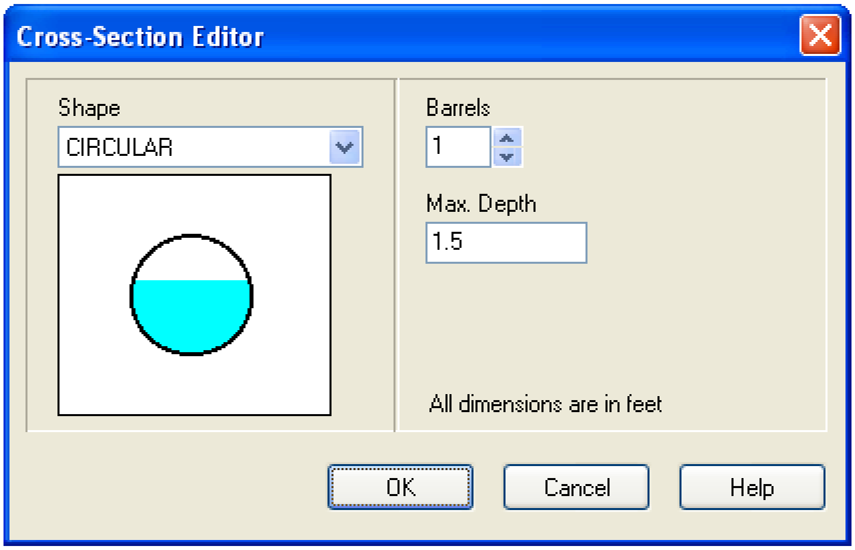


Figure 2-7. Cross-Section Editor dialog.

In order to provide a source of rainfall input to our project we need to set the rain gage's properties. Select **Gage1** into the Property Editor and set the following properties:

Rain Format	INTENSITY
Rain Interval	1:00
Data Source	TIMESERIES
Series Name	TS1

As mentioned earlier, we want to simulate the response of our study area to a 3-inch, 6-hour design storm. A time series named **TS1** will contain the hourly rainfall intensities that make up this storm. Thus we need to create a time series object and populate it with data. To do this:

1. From the Data Browser select the **Time Series** category of objects.
2. Click the button on the Browser to bring up the Time Series Editor dialog (see Figure 2-8)[5].
3. Enter **TS1** in the Time Series Name field.
4. Enter the values shown in Figure 2.8 into the Time and Value columns of the data entry grid (leave the Date column blank[6]).

---

[5] The Time Series Editor can also be launched directly from the Rain Gage Property Editor by selecting the editor's Series Name field and double clicking on it.

5. You can click the **View** button on the dialog to see a graph of the time series values. Click the **OK** button to accept the new time series.

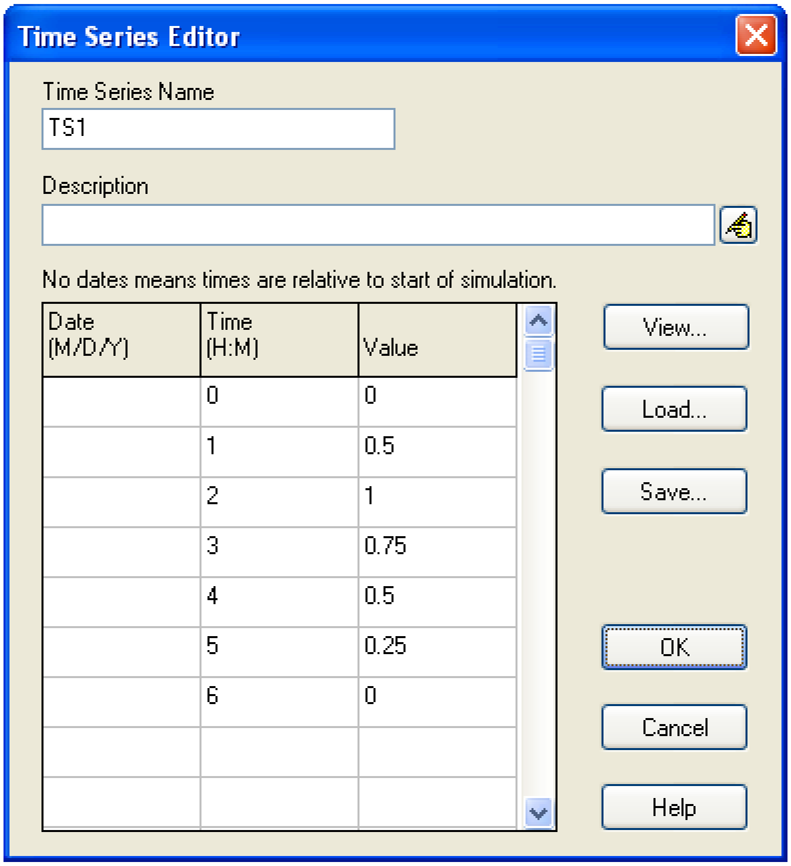


Figure 2-8. Time Series Editor dialog.

Having completed the initial design of our example project it is a good idea to give it a title and save our work to a file at this point. To do this:

1. Select the **Title/Notes** category from the Data Browser and click the button.
2. In the Project Title/Notes dialog that appears (see Figure 2-9), enter "Tutorial Example" as the title of our project and click the **OK** button to close the dialog.
3. From the **File** menu select the **Save As** option.
4. In the Save As dialog that appears, select a folder and file name under which to save this project. We suggest naming the file tutorial.inp. (An extension of .inp will be added to the file name if one is not supplied.)
5. Click **Save** to save the project to file.

---

[6] Leaving off the dates for a time series means that SWMM will interpret the time values as hours from the start of the simulation. Otherwise, the time series follows the date/time values specified by the user.

The project data are saved to the file in a readable text format. You can view what the file looks like by selecting **Project** >> **Details** from the main menu. To open our project at some later time, you would select the **Open** command from the **File** menu.

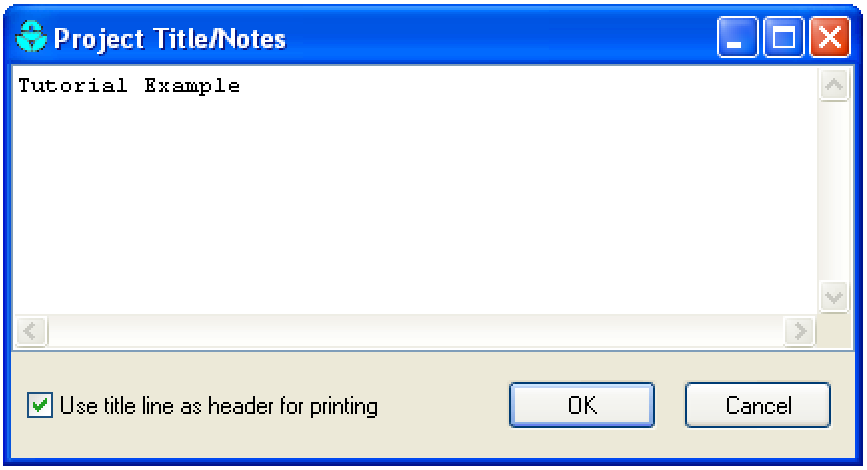


Figure 2-9. Title/Notes Editor.

2.5 Running a Simulation

Setting Simulation Options

Before analyzing the performance of our example drainage system we need to set some options that determine how the analysis will be carried out. To do this:

1. From the Data Browser, select the **Options** category and click the button.
2. On the General page of the Simulation Options dialog that appears (see Figure 2-10), select **Kinematic Wave** as the flow routing method. The flow units should already be set to **CFS** and the infiltration method to **Green-Ampt**. The Allow Ponding option should be unchecked.
3. On the Dates page of the dialog, set the End Analysis time to **12:00:00**.
4. On the Time Steps page, set the Routing Time Step to **60** seconds.
5. Click **OK** to close the Simulation Options dialog.

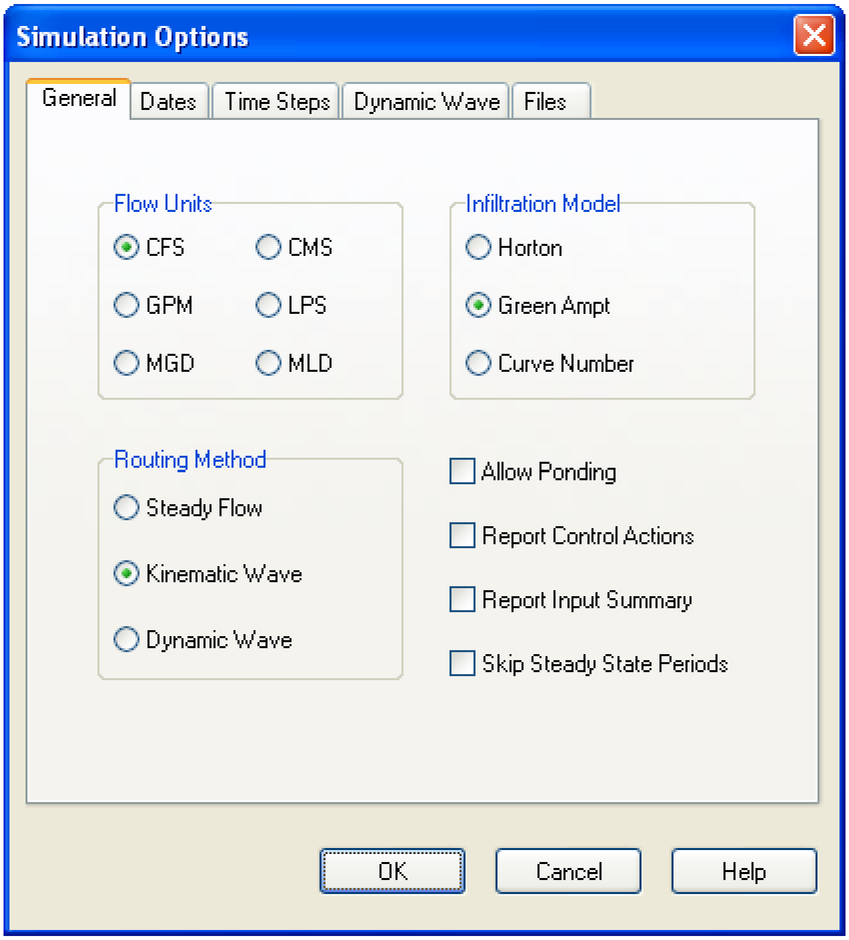
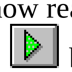


Figure 2-10. Simulation Options dialog.

Running a Simulation

We are now ready to run the simulation. To do so, select **Project** >> **Run Simulation** (or click the  button). If there was a problem with the simulation, a Status Report will appear describing what errors occurred. Upon successfully completing a run, there are numerous ways in which to view the results of the simulation. We will illustrate just a few here.

Viewing the Status Report

The Status Report contains useful summary information about the results of a simulation run. To view the report, select **Report** >> **Status**. A portion of the report for the system just analyzed is shown in Figure 2-11. The full report indicates the following:

- The quality of the simulation is quite good, with negligible mass balance continuity errors for both runoff and routing (-0.23% and 0.03%, respectively, if all data were entered correctly).
- Of the 3 inches of rain that fell on the study area, 1.75 infiltrated into the ground and essentially the remainder became runoff.

- The Node Depth Summary table (not shown in Figure 2-11) indicates there was internal flooding in the system at node **J2**[7].
- The Conduit Flow Summary table (also not shown in Figure 2-11) shows that Conduit **C2**, just downstream of node **J2**, was surcharged and therefore appears to be slightly undersized.

```
EPA STORM WATER MANAGEMENT MODEL - VERSION 5.0
-----------------------------------------------

Tutorial Example

****************
Analysis Options
****************
Flow Units ............... CFS
Infiltration Method ...... GREEN_AMPT
Flow Routing Method ...... KW
Starting Date ............ JUN-27-2002 00:00:00
Ending Date .............. JUN-27-2002 12:00:00
Wet Time Step ............ 00:15:00
Dry Time Step ............ 01:00:00
Routing Time Step ........ 00:01:00
Report Time Step ......... 00:15:00


**************************        Volume         Depth
Runoff Quantity Continuity     acre-feet        inches
**************************     ---------       -------
Total Precipitation ......         3.000         3.000
Evaporation Loss .........         0.000         0.000
Infiltration Loss ........         1.750         1.750
Surface Runoff ...........         1.241         1.241
Final Surface Storage ....         0.016         0.016
Continuity Error (%) .....        -0.228

**************************        Volume        Volume
Flow Routing Continuity        acre-feet      Mgallons
**************************     ---------     ---------
Dry Weather Inflow .......         0.000         0.000
Wet Weather Inflow .......         1.246         0.406
Groundwater Inflow .......         0.000         0.000
RDII Inflow ..............         0.000         0.000
External Inflow ..........         0.000         0.000
Internal Flooding ........         0.054         0.018
External Outflow .........         1.192         0.388
Evaporation Loss .........         0.000         0.000
Initial Stored Volume ....         0.000         0.000
Final Stored Volume ......         0.000         0.000
Continuity Error (%) .....         0.029
```

Figure 2-11. Portion of the Status Report for initial simulation run.

[7] In SWMM, flooding will occur whenever the water surface at a node exceeds the maximum defined depth. Normally such water will be lost from the system. The option also exists to have this water pond atop the node and be re-introduced into the drainage system when capacity exists to do so.

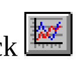
Viewing Results on the Map

Simulation results (as well as some design parameters, such as subcatchment area, node invert elevation, link maximum depth) can be viewed in color-coded fashion on the study area map. To view a particular variable in this fashion:

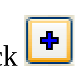
1. Select the Map page of the Browser panel.
2. Select the variables to view for Subcatchments, Nodes, and Links from the dropdown combo boxes labeled Subcatch View, Node View, and Link View, respectively. In Figure 2-12, subcatchment runoff and link flow have been selected for viewing.
3. The color-coding used for a particular variable is displayed with a legend on the study area map. To toggle the display of a legend, select **View** >> **Legends**.
4. To move a legend to another location, drag it with the left mouse button held down.
5. To change the color-coding and the breakpoint values for different colors, select **View** >> **Legends** >> **Modify** and then the pertinent class of object (or if the legend is already visible, simply right-click on it). To view numerical values for the variables being displayed on the map, select **View** >> **Map Options** and then select the Annotation page of the Map Options dialog. Use the check boxes for Areas (i.e., sub-catchments), Nodes, and Links in the Values panel to specify what kind of annotation to add.
6. The Date / Time / Elapsed Time controls on the Map Browser can be used to move through the simulation results in time. Figure 2-12 depicts results at 5 hours and 45 minutes into the simulation.
7. To animate the map display through time, select **View** >> **Toolbars** >> **Animator** and use the controls on the Animator Toolbar to control the animation. For example, pressing the  button will run the animation forward in time.

Viewing a Time Series Plot

To generate a time series plot of a simulation result:

1. Select **Report** >> **Graph** >> **Time Series** or simply click  on the Standard Toolbar and select **Time Series** from the pull-down menu that appears.
2. A Time Series Plot dialog will appear. It is used to select the objects and variables to be plotted.

For our example, the Time Series Plot dialog can be used to graph the flow in conduits **C1** and **C2** as follows (refer to Figure 2-14):

1. Select **Links** as the Object Category.
2. Select **Flow** as the Variable to plot.
3. Click on conduit **C1** (either on the map or in the Data Browser) and then click  in the dialog to add it to the list of links plotted. Do the same for conduit **C2**.
4. Press **OK** to create the plot, which should look like the graph in Figure 2-15.

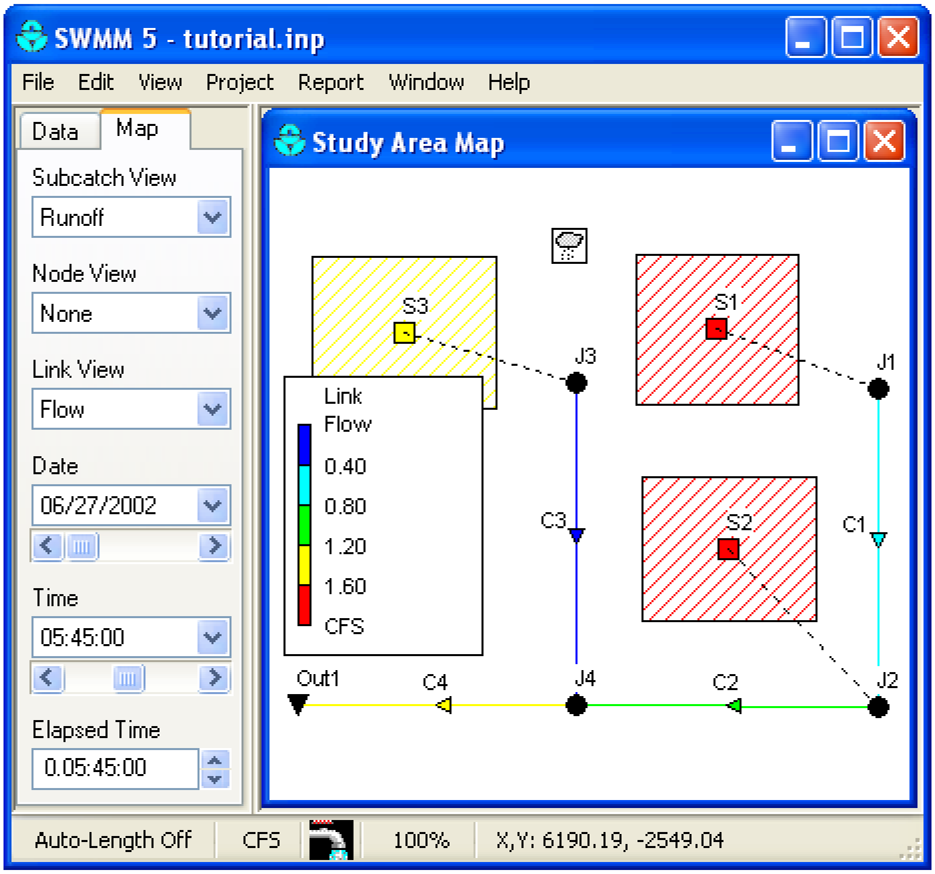


Figure 2-12. Example of viewing color-coded results on the Study Area Map.

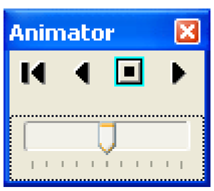


Figure 2-13. The Animator Toolbar.

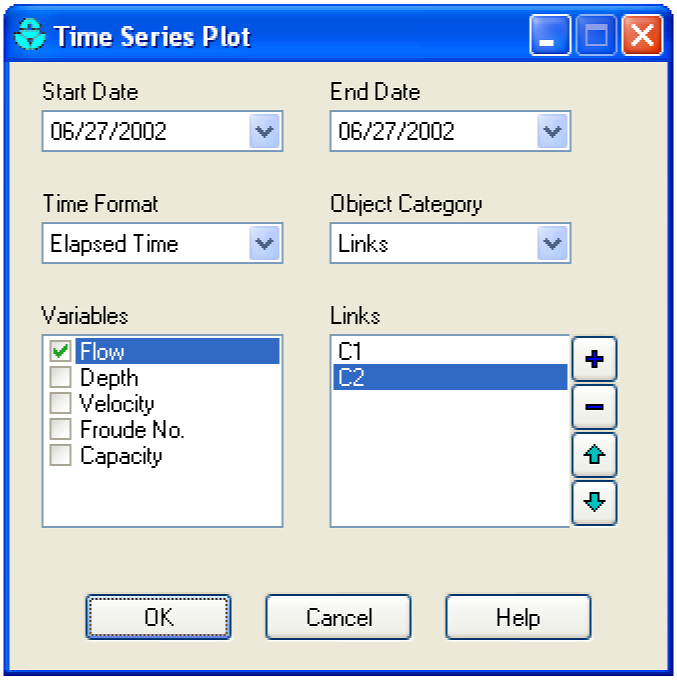


Figure 2-14. Time Series Plot dialog.

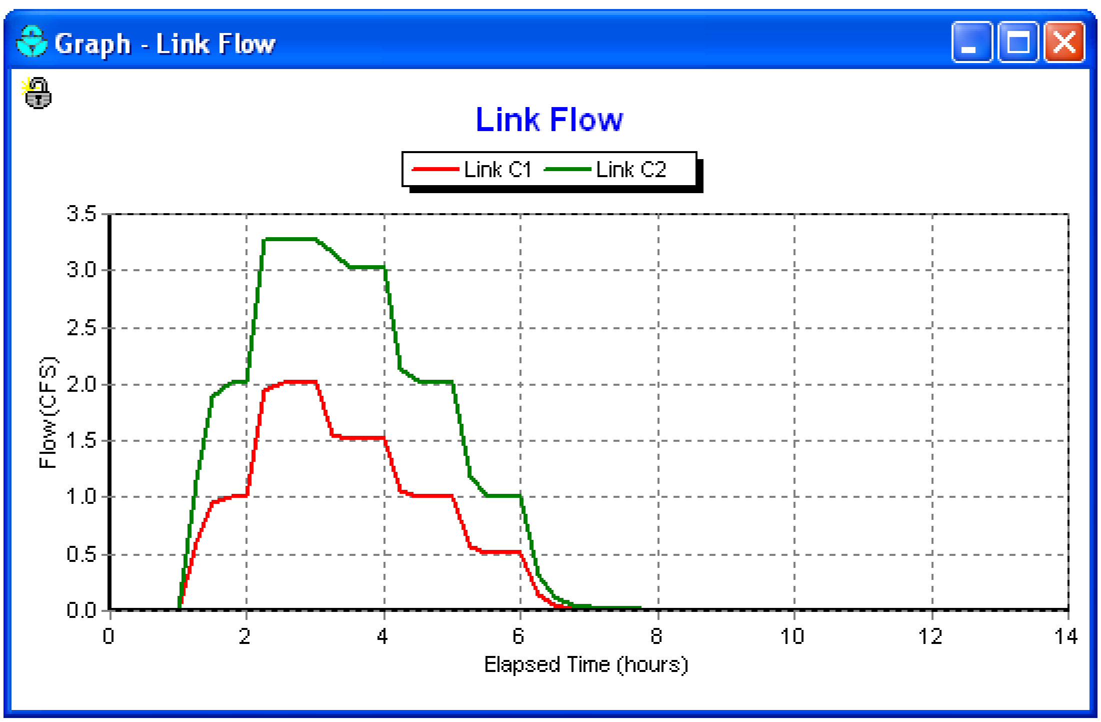


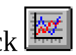
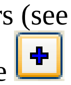
Figure 2-15. Time Series plot of results from initial simulation run.

After a plot is created you can:

- customize its appearance by selecting **Report** >> **Options** or right clicking on the plot,
- copy it to the clipboard and paste it into another application by selecting **Edit** >> **Copy To** or clicking  on the Standard Toolbar
- print it by selecting **File** >> **Print** or **File** >> **Print Preview** (use **File** >> **Page Setup** first to set margins, orientation, etc.).

Viewing a Profile Plot

SWMM can generate profile plots showing how water surface depth varies across a path of connected nodes and links. Let's create such a plot for the conduits connecting junction **J1** to the outfall **Out1** of our example drainage system. To do this:

1. Select **Report** >> **Graph** >> **Profile** or simply click  on the Standard Toolbar and select **Profile** from the pull-down menu that appears.
2. Either enter **J1** in the Start Node field of the Profile Plot dialog that appears (see Figure 2-16) or select it on the map or from the Data Browser and click the  button next to the field.

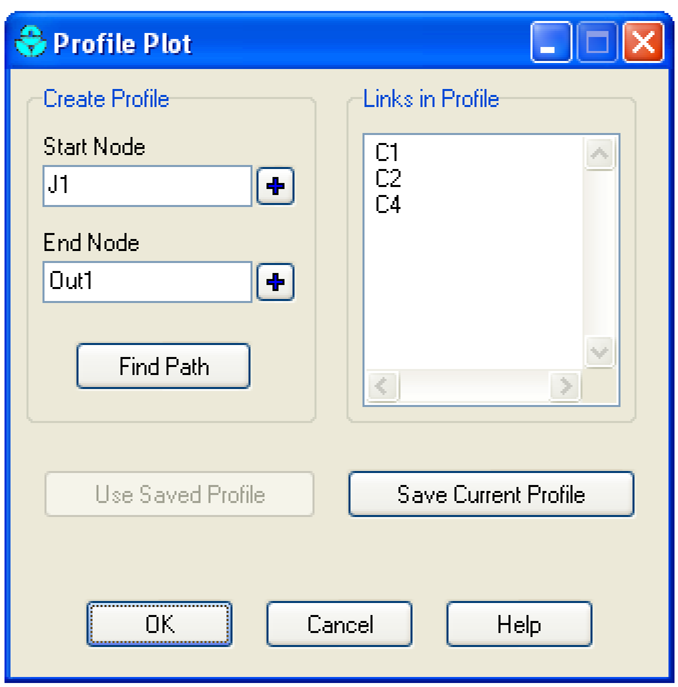


Figure 2-16. Profile Plot dialog.

3. Do the same for node **Out1** in the End Node field of the dialog.
4. Click the **Find Path** button. An ordered list of the links forming a connected path between the specified Start and End nodes will be displayed in the Links in Profile box. You can edit the entries in this box if need be.
5. Click the **OK** button to create the plot, showing the water surface profile as it exists at the simulation time currently selected in the Map Browser (see Figure 2-17).

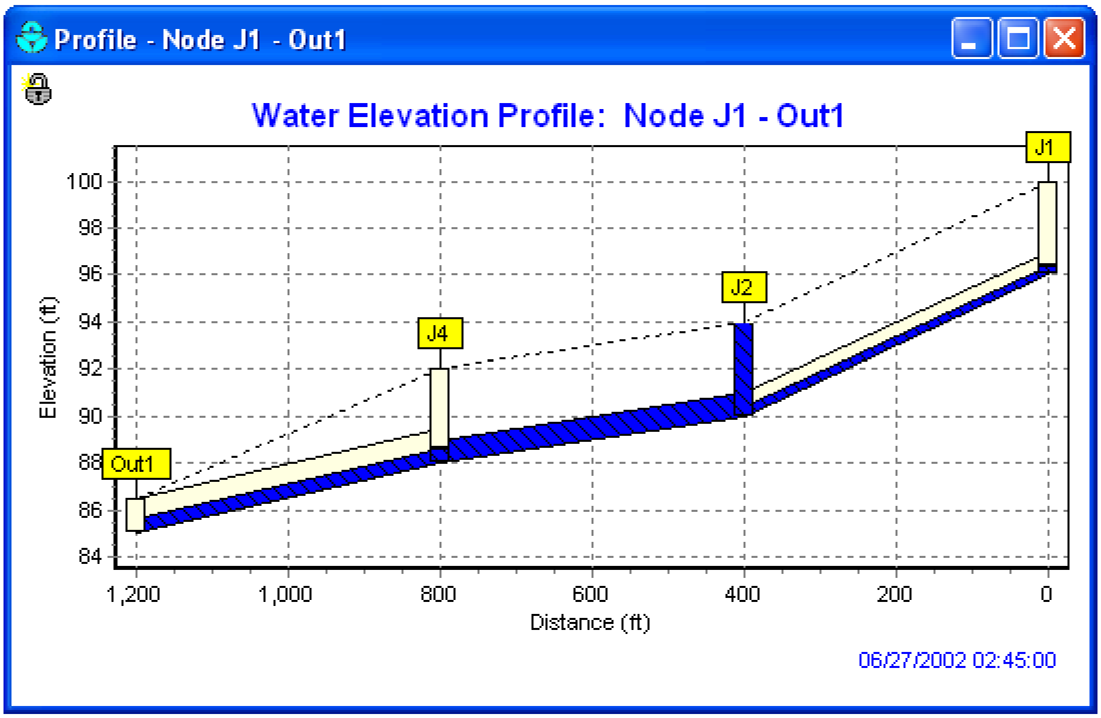


Figure 2-17. Example of a Profile plot.

As you move through time using the Map Browser or with the Animator control, the water depth profile on the plot will be updated. Observe how node **J2** becomes flooded between hours 2 and 3 of the storm event. A Profile Plot's appearance can be customized and it can be copied or printed using the same procedures as for a Time Series Plot.