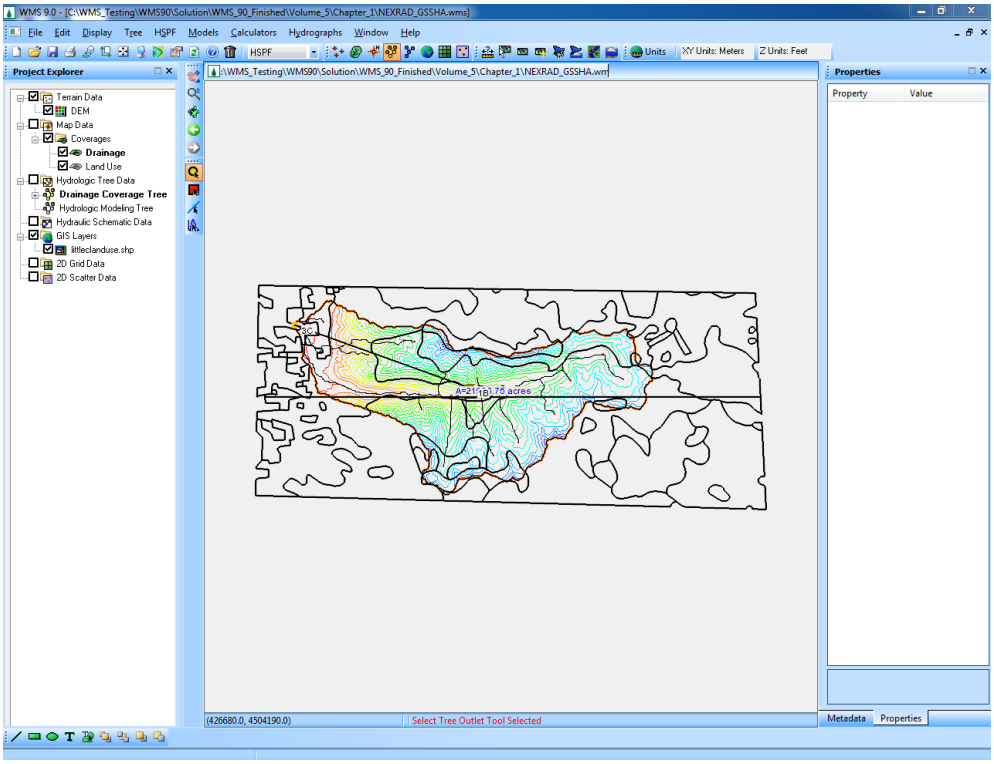
WMS

v. 9.1

WMS 9.1 Tutorial

Water Quality Modeling – HSPF Interface

Setup a basic HSPF model



Objectives

Read a delineated watershed and setup and run an HSPF model for the watershed.

Prerequisite Tutorials

- Watershed Modeling – DEM Delineation
- Watershed Modeling – Time of Concentration Calculations and Computing a Composite CN

Required Components

- Data
- Drainage
- Map
- Hydrology
- Hydrologic Models

Time

- 30-60 minutes

AQUAVEO™

1 Contents

2 Introduction

This chapter demonstrates how WMS can be used to process digital elevation and land use data to develop an HSPF input (.uci) file. The graphical user interface used to define input parameters is demonstrated for a basic hydrology simulation that includes doing the following:

- Delineating watershed segment boundaries from a digital terrain model and USGS land use file
- Defining segment parameters for a hydrologic analysis
- Developing reach segment parameters
- Defining precipitation time series data from standard WDM database files
- Entering mass links to define transformations from basin to reach

3 Opening the Watershed and Initializing the HSPF Model

You will open a watershed that has been delineated from a USGS DEM in GridFloat format downloaded from USGS' seamless DEM web site at http://seamless.usgs.gov.

1. Close all instances of WMS
2. Open WMS
3. Select ***File | Open***
4. Locate the ***hspf*** folder in your tutorial files. If you have used default installation settings in WMS, the tutorial files will be located in ***\My documents\WMS 9.1\Tutorials\.***
5. Open the file named *"lc.wms"*

The watershed will appear on your screen with the area displayed in the center of the basin. This indicates that the Basin Data has been computed and you are ready to set up the HSPF model. To initialize the model:

6. Make sure that the projections are correct by selecting ***Edit | Current Projection...*** and ensuring that the horizontal and vertical units are both set to *Meters*.
7. Switch to the *Hydrologic Modeling* module 
8. Choose HSPF from the Model drop-down list at the top of the screen
9. Select ***HSPF | New Simulation***
10. Select ***HSPF | Global Options***

You will now enter the parameters to indicate that the model will simulate about 3 years of time with a time step of 1 day. You will also name the model, choose the units to use, specify output levels, and name the input/output files:

11. Enter "Little Cottonwood Canyon Model" in the Title field
12. Set the Start Time to January 1, 1996 (1/1/1996) with Hour: 0, Minute: 0, Second: 0.0 (12:00:00 AM)
13. Set the End Time to July 31, 1999 (7/31/1999) with Hour: 0, Minute: 0, Second: 0.0 (12:00:00 AM)
14. Enter 24 in the hours field under the Time step heading (leave minutes at 0)
15. Ensure that the Units flag is set to English
16. Change the Run Flag to 1; this indicates that HSPF will interpret the data and run the simulation (0 indicates that HSPF will only interpret – error check – the data)
17. Set the OUTLEV to 10 (under Output Levels); this indicates maximum output to the Error and Warning files. Leave the SPOUT flag at 0 (this is only for Special Actions)
18. In the Files section, enter "littleco" in the Prefix field, then click the Update Filenames button. This names all the HSPF input/output files associated with this model to the same name.

NOTE: You have just indicated that you will be using a file named *littleco.wdm* for time series input and output – this file must be created and named appropriately outside of WMS (this has been done for you in this case). You can create a ".wdm" (watershed data management) file using the WDMUtil program installed when you install HSPF from the WMS installation.

19. Select OK

The Global Options are now set for your model. You are now ready to proceed with segmenting the watershed and entering parameters for the segments.

4 Importing Land Use and Segmenting the Watershed

To divide this watershed into hydrologically similar segments, you will overlay a land-use data layer. You will read this data from an ArcView Shapefile:

1. Select ***File | Open***
2. Open the file named *"littleclanduse.shp"*

3. Right-click on the Coverages folder in the Project Explorer and select New Coverage
4. Change the coverage type to Land Use and click OK
5. Switch to the *GIS* module
6. Select ***Mapping | Shapes->Feature Objects***
7. Select Yes to use all available shapes
8. Select Next
9. Ensure that LUCODE is mapped to Land Use
10. Select Next and then Finish
11. Turn off the display of *littleclanduse.shp* by deselecting it in the Project Explorer

You will see the land use data polygons overlaid on the watershed. You are now ready to compute HSPF segments based on the land use polygons that fall inside the watershed.

12. Switch to the *Hydrologic Modeling* module
13. Select ***Calculators | Compute GIS Attributes***
14. Choose HSPF Segments from the Computation drop-down list
15. In the central section of the dialog, ensure that coverage appears next to "Use a" text and that Land Use is the selected Land use coverage name
16. A text file that correlates the land use ID to land use attributes (name, perviousness, etc.) must be read in. Click the Import button to browse for this file.
17. Select OK to overwrite existing land use table
18. Open the file named *"littleclanduse.tbl"*
19. Choose OK in the Compute GIS Attributes dialog to compute the HSPF segments
20. Click OK if warned that Continuing will delete all HSPF segment data.
21. Deselect the toggle box next to *Land Use* in the Project Explorer
22. Select the *Drainage* coverage in the Project Explorer to make it the active coverage

5 Aggregating Segments

You will note from the display that there are several land segments in the watershed. Some are quite large, such as the Evergreen Forest Land segment, while others are rather insignificant, like Bare Exposed Rock. To make this simulation simpler, you will aggregate some of these segments.

1. Switch to the *Hydrologic Modeling* module
2. Double-click on the basin icon (or select it and then select ***HSPF | Edit Parameters***)

3. Note in the Edit Parameters dialog the list of segments in the Basin Data window. There are 8 separate segments (or land use classifications) in this watershed.

To reduce the number of segments computed by WMS, first delete one segment, and then add the area of the deleted segment, manually, to another similar segment.

4. Choose the segment entitled Other Urban or Built Up from the Basin Data window. Note that the area is 57.89 acres – insignificant in this watershed. You will delete this area and add it to the Shrub & Brush Rangeland segment to aggregate these segments.
5. Select the Delete button
6. Click Yes for the message that asks if the segment is to be removed from further use
7. Repeat these steps for the segment entitled Bare Ground. Note the area of 378.73 for the Bare Ground segment.
8. Enter a new area of 7999.06 to the Shrub & Brush Rangeland segment (7562.44 (Shrub & Brush Rangeland area) + 378.73 (Other Urban Or Built Up area) + 57.89 (Bare Ground area))
9. Delete the Mixed Forest segment and add its area to the Evergreen Forest segment. The new total area for the Evergreen Forest segment will be 12162.63 acres.
10. Delete the Bare Exposed Rock segment and add its area to the Mixed Tundra segment. The new total area for the Mixed Tundra segment will be 1189.25 acres.

You are now left with 4 land segments. These are the segments for which you will now input parameters and simulate with HSPF.

6 Defining Land Segment Parameters

You are ready to begin defining parameters for the land segments of the HSPF model. This model will be set up for purely hydrologic analysis; thus, you will be activating and inputting parameters only to the SNOW and WATER modules of HSPF. Further, to decrease the time needed to complete the exercise, you will fully set up one segment then copy those parameters to the other 3 in the model. Do the following to complete these tasks:

1. Choose the Evergreen Forest segment from the Basin Data window
2. Click the Define… button for this segment
3. Click the boxes next to Snow (SNOW) and Water (PWAT) to checkmark them.

You have just activated the Snow and Water modules for this land segment (Evergreen Forest). You must now enter the parameters for each active module to allow HSPF to simulate the segment correctly.

4. Click on the Snow (SNOW) text to highlight it

5. Toggle between the tabs to enter the following values in the appropriate fields (leave other fields as the default values; these values are either generally recommended values or HSPF defaults):

SNOW-INIT1	SNOW-INIT2	SNOW-PARM1	SNOW-PARM2
Pack-snow = 4.0 in Pack-ice = 2.0 in Pack-watr = 2.0 in	SKYCLR = 1.0	LAT = 40.5° MELEV = 8410.8 ft SHADE = 0.40	TSNOW = 35.0° F MWATER = 0.2

NOTE: These values were extracted from meteorological datasets.

6. Now that all values are entered for this segment, click the Apply Parameters to Segments button. This will allow you to assign these same parameters to other segments in the model.
7. Choose Shrub & Brush Rangeland in the Available Segments window; move it to the Selected Segments window by clicking the --> button.
8. Repeat for Residential and Mixed Tundra
9. Click OK. The selected segments will be assigned the same parameters input here.

The final step in entering parameters for a module in HSPF is to define time series input (External Sources) and time series output (External Targets). For the land segments in this model, you will define External Sources for input data, but no External Targets will be specified – you will choose output to be given at the outlet of the watershed, not at each land segment.

10. Click on the External Sources button with the Snow (SNOW) module highlighted. The Assign External Sources dialog will appear with a list of the datasets available in the WDM file specified for this model (*littleco.wdm*).
11. If the dialog is empty (and you might also get a warning message that the file is not there), open the *"littleco.wdm"* file and click OK
12. Assign the following datasets as sources by setting the fields to the appropriate values, then clicking the Assign button for each line. If you make a mistake, choose the incorrect line in the lower window and click Delete.

Member Name	Units	Missing Data	Transfor-mation	Quality Flag	Multiplication Factor	Dataset
PREC	ENGL	UNDF	SAME	0	0.25	3
PREC	ENGL	UNDF	SAME	0	0.75	7
DTMPG	ENGL	UNDF	AVER	0	1.0	14
WINMOV	ENGL	UNDF	AVER	0	1.0	15
SOLRAD	ENGL	UNDF	SAME	0	1.0	16
AIRTMP	ENGL	UNDF	AVER	0	0.4	17
AIRTMP	ENGL	UNDF	AVER	0	0.6	18

13. Since you will add other External Sources to this segment with the PWAT module input, you will not assign the External Sources to other segments yet. Click Done to return to the SNOW dialog.

You have completed the set up of the *SNOW* module. You will now repeat the same basic steps to set up the PWAT module for the land segments of the model.

14. Click on Water (PWAT) to highlight it
15. Toggle between the tabs to enter the following values in the appropriate fields (leave other fields as the default values):

PWAT-PARM1	PWAT-PARM2	PWAT-PARM3	PWAT-PARM4	PWAT-STATE1
CSNOFG = On RTOPFG = On UZFG = On	LZSN = 5.0 INFILT = 0.48 LSUR = 2258.0 SLSUR = 0.25 KVARY = 0.7 AGWRC = 0.997	DEEPFR = 0.1 PETMAX = 35.0 PETMIN = 30.0	CEPSC = 0.1 UZSN = 1.0 NSUR = 0.25 INTFW = 3.0 IRC = 0.7 LZETP = 0.8	CEPS = 0.2 UZS = 1.0 LZS = 6.0 AGWS = 1.6 GWVS = 1.9

16. Now that all values are entered for this segment, click the Apply Parameters to Segments button. This will allow you to assign these same parameters to other segments in the model.
17. Choose Shrub & Brush Rangeland in the Available Segments window; move it to the Selected Segments window by clicking the --> button.
18. Repeat for Residential and Mixed Tundra
19. Click OK. The selected segments will be assigned the same parameters input here.

Now you must define the additional External Sources needed for the PWATER simulation. Most of the External Sources entered for the SNOW module are also used for the PWATER module, but there is one input series that is not yet entered.

20. Click on the External Sources button with the Water (PWAT) module highlighted. The Assign External Sources dialog will appear with the sources that you created for this segment in the SNOW module listed in the lower window.
21. Assign the following dataset (evapotranspiration) as a source by setting the fields to the appropriate values, then clicking the Assign button. If you make a mistake, choose the incorrect line in the lower window and click Delete.

Member Name	Units	Missing Data	Transfor-mation	Quality Flag	Multiplication Factor	Dataset
PETINP	ENGL	UNDF	SAME	0	1.0	23

All the External Sources needed for the simulation are now assigned to this segment. You will now copy these same External Sources to all other land segments in the model.

22. Click the Apply Sources to Segments button. This will allow you to assign these same External Sources to other segments in the model.
23. Choose Shrub & Brush Rangeland in the Available Segments window; move it to the Selected Segments window by clicking the --> button
24. Repeat for Residential and Mixed Tundra
25. Select OK

26. Select Done in the Assign External Sources dialog
27. Select OK in the HSPF Pervious Data dialog

The Edit Parameters dialog should now be active. You have successfully set up the SNOW and PWATER modules for all land segments in the model (remember that you set up the modules explicitly for the Evergreen Forest segment and then copied the parameters to all other segments). You may want to ensure/review the set up for the other segments by choosing the segment in the Basin Data window, then clicking the Define Activities button and reviewing the SNOW and PWATER input.

Now that the land segment data is complete, you will enter data for the reach (stream) segment of the model.

28. Click Done to close the Edit Parameters dialog

7 Defining Reach Segment Parameters

To simulate runoff in-stream, you will need to activate the HYDR module for the reach segment of the model. You will also need to specify the output dataset needed (a hydrograph in this case) to view the results of the simulation.

1. Double-click on the watershed outlet in the Graphics Window (this can be done while the Edit Parameters dialog is still present by only single-clicking the outlet). The Reach/Reservoir Data section of the Edit Parameters dialog will become active and display the name of the outlet chosen (3C).
2. Click on the Define… button for this segment
3. Check the box to activate the Hydraulics (HYDR) module of HSPF for this reach
4. Toggle between the tabs to enter the following values in the appropriate fields (leave other fields as the default values):

HYDR-PARM1	HYDR-PARM2
ODFVFG = 4	Length = 13.82 DeltaH = 2200.0

The next input you must enter is the FTABLE for the reach. The FTABLE is a spreadsheet-like table that contains the conveyance parameters of the reach (depth, area, volume, and outflow). This table may be calculated manually or with the help of the Channel Calculator in WMS.

5. Click on the Define button under the FTABLE row in the HYDR-PARM1 tab
6. In the FTABLE dialog that appears, enter the following values (it is possible to change the number of rows and columns in the FTABLE, but the default values will be sufficient in this case. These values are saved in a text file called *ftable.txt* in the hspf directory of the tutorials. You can open this file, copy the data (use CTRL-C to copy), and paste the data to the FTABLE dialog (use CTRL-V to paste) if you wish):

Depth (ft)	Area (ac)	Volume (ac-ft)	Outflow (cfs)
0.0	0.0	0.0	0.0

0.25	25.99	6.393	13.876
0.5	26.828	12.995	43.954
0.75	27.666	19.807	86.269
1.0	25.50	26.828	139.241
1.25	29.340	34.059	201.950
1.5	30.181	41.499	273.794
1.75	31.019	49.150	354.355
2.0	31.858	57.009	443.334
2.25	35.211	90.540	879.902

7. Click OK to save the FTABLE

The final task in defining the reach parameters is to specify the output you want to see from the reach and where HSPF will write the output. This is done by defining an External Target. You will specify that you want a hydrograph to be output to a dataset in the WDM (*littleco.wdm*) file for this reach.

8. With the HYDR option selected, click on the External Targets button.
9. Set the following fields to the specified values:

Member Name	Units	Access	Aggreg-ation	Multiplication Factor	Quality Flag	Transformation Function
RO	ENGL	REPL	Aggregate	1.0	0	AVER

10. Select the Use existing dataset option
11. Click on the Select Dataset button
12. In the Select Dataset dialog, scroll down and select dataset 20 (Flow), then click the OK button
13. The Dataset name, type, and number should appear in the Assign External Targets dialog
14. Click Assign to add the External Target line to the lower window
15. Click Done

You have now completed the input for the HYDR module in this reach. This will be the only module active for this simulation; thus, you can exit the parameter editing dialogs.

16. Click OK in the HSPF Reach/Reservoir segment attributes dialog
17. Click Done in the Edit Parameters dialog

The input for all land and reach segments is now complete in the model. The last task before saving and running the HSPF model is to assign Mass Links.

8 Creating Mass Links

Mass Links control how materials (water, sediment, constituents) are transferred from land segments to reaches, and from one reach to the next. Conversions in units, such as inches/acre per day of runoff to cubic feet per second, are defined in the Mass Links also. You will enter Mass Links to transfer water from land to reach and from reach to reach for this model.

1. Select ***HSPF | Mass Link Editor***

2. When you enter the Mass Link Editor, it is set to begin adding/deleting mass links for PERLND segments. Set the fields to the values below:

Volume Name	Volume Group	Volume Member Name	Multiplication Factor	Target Name	Target Group	Target Member Name
PERLND	PWATER	PERO	0.08333	RCHRES	INFLOW	IVOL

NOTE: 0.08333 is the conversion for inches/acre per day to cfs.

3. Click Add Link (if you make a mistake, select the link in the window and click the Delete Link button)
4. Change the Segment Type (upper left) to IMPLND
5. Set the fields to the values below:

Volume Name	Volume Group	Volume Member Name	Multiplication Factor	Target Name	Target Group	Target Member Name
IMPLND	IWATER	SURO	0.08333	RCHRES	INFLOW	IVOL

6. Click Add Link
7. Change the Segment Type (upper left) to RCHRES
8. Set the fields to the values below:

Volume Name	Volume Group	Volume Member Name	Multiplication Factor	Target Name	Target Group	Target Member Name
RCHRES	HYDR	ROVOL	1.00	RCHRES	INFLOW	IVOL

9. Click Add Link

The Mass Links needed for PERLND, IMPLND, and RCHRES segments are now set up. You are ready to save and run the model

10. Click OK to exit the Mass Link Editor

9 Saving and Running an HSPF Simulation

The last step is to save and run the HSPF UCI (User control input) file.

1. Select ***HSPF | Save HSPF UCI File…***
2. Select Save

WMS will save the HSPF UCI file. You should save your project from WMS, close WMS, and try running the project in either WinHSPF or WinHSPFLt. WinHSPF and WinHSPFLt are installed when you install HSPF from the WMS installation. If you try to run the UCI file from WinHSPF while you have WMS open, you will receive an error since both WMS and WinHSPF are trying to access the WDM file associated with the UCI file, and only one program can access the WDM file at a time.

TIP: HSPF cannot read WDM files in folders with long path/file names (combined path/file names longer than 64 characters). If your WDM file is located in a folder with a long path name, move your .wdm file to another folder with a short path name (such as "c:\models\hspf") and save your .uci file from WMS here. Then save your WMS project file, close WMS, read your .uci file into WinHSPF or WinHSPFLT, and run your model.

