

Training Course

Hydrologic Modeling with GIS and the Watershed Modeling System (WMS)

June 9, 12-14 2000

Flood Control District of Maricopa County

Training Manual

Property of
Flood Control District of MC Library
Please Return to
2801 W. Durango
Phoenix, AZ 85009

ems-i

Environmental Modeling Systems

Training Course

*Hydrologic Modeling with
GIS and the Watershed
Modeling System (WMS)*

June 9, 12-14 2000

Flood Control District of Maricopa County

Training Manual

ems-i

Environmental Modeling Systems

WMS TRAINING SCHEDULE

DAY 0 (JUNE 9, 2000): BY FCDMC STAFF

Time	Activity	Topic
8:00 - 9:15	Lecture	Introduction - Customization of WMS for FCDMC
9:15 - 9:30	Break	
9:30 - 10:15	Lecture	Demos
10:15 - 10:30	Break	
10:30-12:00	Tutorial Session	Hands-on
12:00-1:00	Lunch on your own	
1:00-1:45	Lecture	Demos
1:45 - 2:00	Break	
2:00-4:00	Tutorial Session	Hands-on
4:00 - 5:00	Discussions	Prepare a list of questions, suggestions, and possible bugs

DAY 1 (JUNE 12, 2000): BY WMS DEVELOPER

Time	Activity	Topic
8:00 - 9:15	Lecture	Introduction - Overview of WMS
9:15 - 9:30	Break	
9:30 - 10:30	Lecture	Feature Objects and the Map Module
10:30-12:00	Tutorial Session	Feature Objects
12:00-1:00	Lunch on your own	
1:00-1:30	Lecture	Basin Delineation with Feature Objects
1:30-2:15	Tutorial Session	Basin Delineation with Feature Objects
2:15 - 2:30	Break	
2:30 - 3:15	Lecture	DEMs and Automated Basin Delineation with DEMs
3:15 - 4:00	Tutorial Session	DEMs
4:00 - 5:00	Lecture	Tins and Automated Basin Delineation with TINs

DAY 2 (JUNE 13, 2000): BY WMS DEVELOPER

Time	Activity	Topic
8:00 - 8:15	Lecture	Review
8:15 - 9:15	Tutorial	TINs
9:15 - 10:15	Lecture	Mapping Curve Numbers and Time of Travel with GIS Objects
10:15 - 10:30	Break	
10:30 - 11:30	Tutorial Session	GIS Mapping
11:30 - 12:00	Lecture	Model Interfaces - National Flood Frequency (NFF) Program
12:00-1:00	Lunch on your own	
1:00 - 1:30	Tutorial Session	NFF Interface
1:30 - 2:30	Lecture	HEC-1 and Special Features for Maricopa County
2:30 - 2:45	Break	
2:45 - 3:45	Tutorial Session	HEC-1
3:45 - 4:15	Lecture	Rational Method
4:15 - 5:00	Tutorial Session	Rational Method

DAY 3 (JUNE 14, 2000): BY WMS DEVELOPER

Time	Activity	Topic
8:00 - 8:15	Lecture	Review
8:15 - 9:00	Lecture	Hydrologic and Hydraulic Calculators
9:00 - 9:45	Tutorial Session	Hydrologic and Hydraulic Calculators
9:45 - 10:00	Break	
10:00 - 10:30	Lecture	Flood Plain Delineation
10:30 - 11:15	Tutorial Session	Flood Plain Delineation
11:15 - 12:00	Lecture	ArcView, WMS-Hydro, Data Collection
12:00-1:00	Lunch on your own	
1:00 - 1:30	Lecture	Questions and Answers
1:30 -2:00	Lecture	Summary problems overview
2:00 - 2:15	Break	
2:15 - 5:00	Tutorial Session	Summary problem



WMS Training

An Introductory Course with Applications

Who Are You?

- Where are you from?
- Do you have any previous experience using WMS?
- What do you expect to learn?
 - Hydrologic models you want to use with WMS?

What Should You Know?

- Basic understanding of hydrology
 - The more modeling you have done the better, but not considered a prerequisite
- Basic computer operations
 - Windows 95, or NT
- Course will focus on WMS as a tool for hydrologic modeling

Summary of Course

- Learn how to process digital elevation data in support of hydrologic modeling
 - Automatic watershed delineation and geometric parameter computations using WMS
 - Use watershed data with NFF, HEC-1, TR-20, and Rational Method
- Learn how to use GIS data layers to set up compute CN's and time of concentration
- Create reports from WMS output

Demonstration

- Lectures will be mixed with live demonstrations of WMS
- Workshops will reinforce lectures
 - Don't be afraid to get off the beaten path
 - Basic tutorial sessions (WMS tutorials)
 - Advanced tutorial sessions
- **ASK QUESTIONS PLEASE!**

History of WMS

- Large library of graphics software developed at ECGL in 70's and 80's.
- Development of application for Watershed Delineation in early 1990's.
- ECGL and COE partner to develop WMS
- FHWA
- Under development for 6.0 years
- First version released in 1993 (GeoShed)

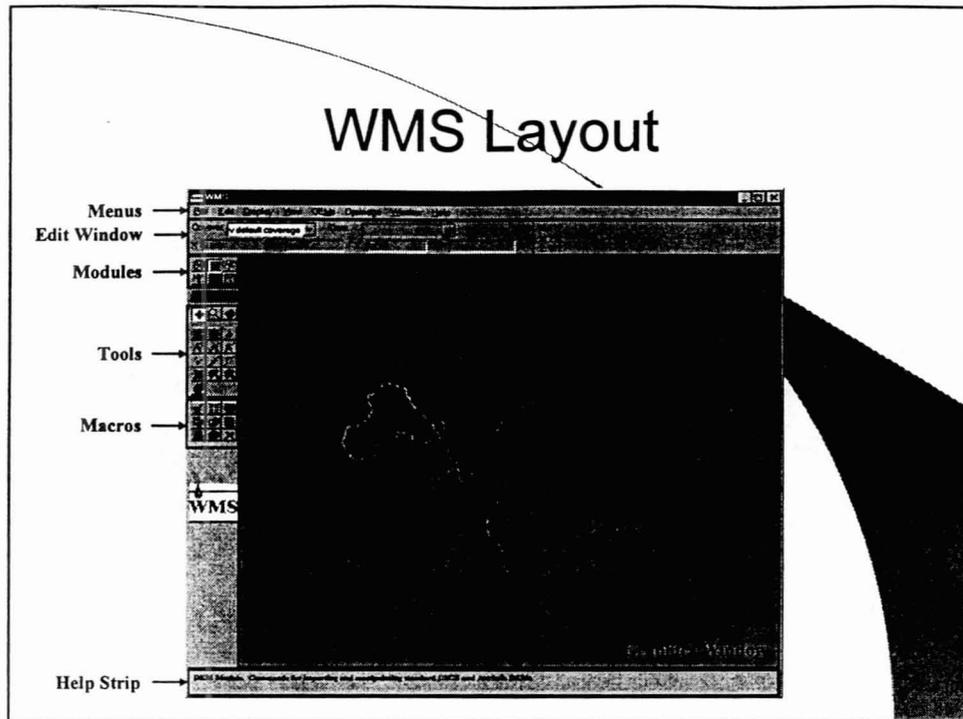
WMS Distribution

- As a result of partnerships, WMS is distributed
 - to Corps of Engineers through U.S. Army Engineer WES
 - to state and federal highway administrators through the Denver regional office.
- EMRL reserves rights to private distribution
 - Approximately 600 private licenses
- EMS-I now handles all distribution

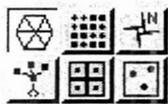
WMS Web Site

- http://www.ems-i.com/software_wms_index.html
- WMS mailing list
- General information on WMS
- News
- Demo versions of WMS
- Bug fixes posted periodically

WMS Layout



Modules



WMS is organized into six modules

-  **TINs**
-  **DEMs**
-  **Map (GIS Layers)**
-  **Hydrologic Modeling**
-  **2D Grids**
-  **Scatter Point**

Tools

The tool set you see depends on the active module



Graphics manipulation tools are in every module

Tools with arrows are for selection

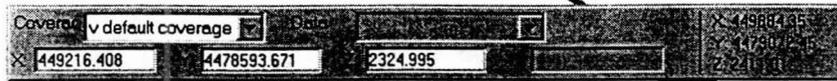
Tools without arrows are for creation

Macros



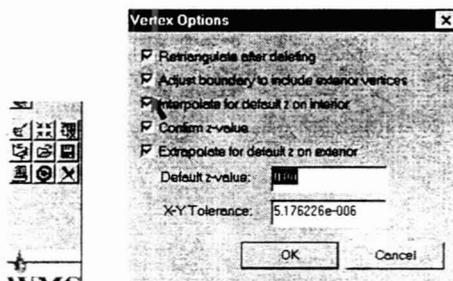
Macros are available for commonly used menu commands

Edit Window



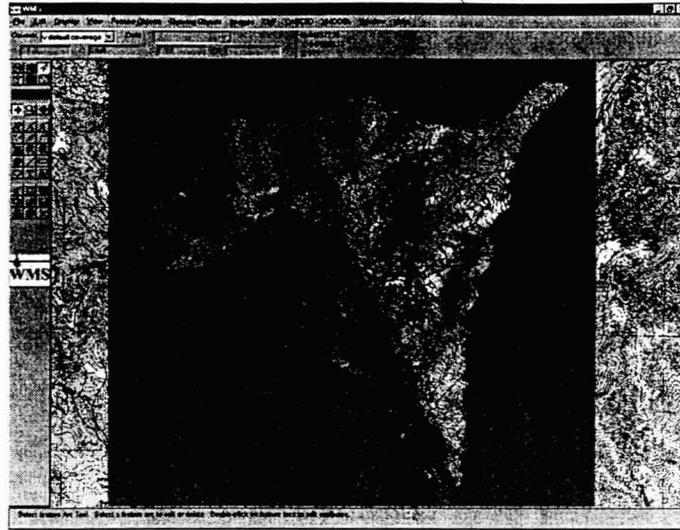
- Edit coordinates of selected objects (points, vertices, etc.)
- Change the active coverage, TIN, data set
- Track the coordinates of the cursor in the graphics window

Help Strip



Toggle on if the newly inserted vertex should be interpolated for the Z value.

Images



Watershed Delineation

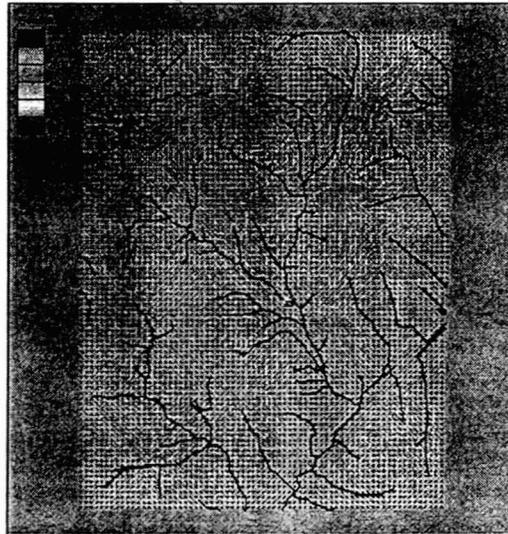
- Traditionally done from paper maps
- WMS provides 3 different ways
 - From Map/GIS-based data (semi-automated)
 - From DEMs (automated)
 - From TINs (automated)

The Basic Goal

- Create a digital representation of a watershed
 - automatically generate a topologic tree
- Compute geometric parameters
- Run Hydrologic model
 - HEC-1, TR20, TR-55, Rational, NFF, others

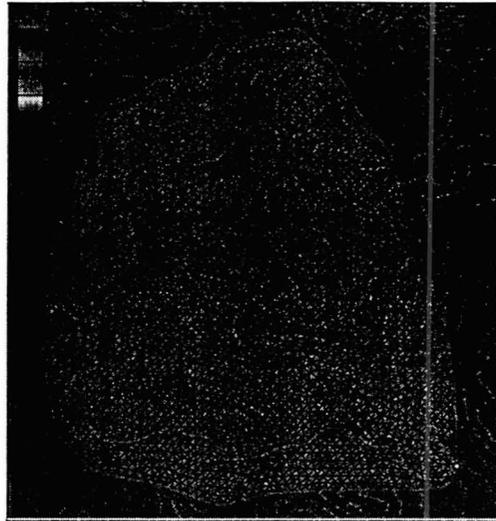
DEMs

Elevation grid
Flow direction
Flow accumulation
Outlet location
Delineate watershed



TINs

Elevation data
Bounding polygon
and streams
Generate TIN
Outlet location
Delineate
watershed



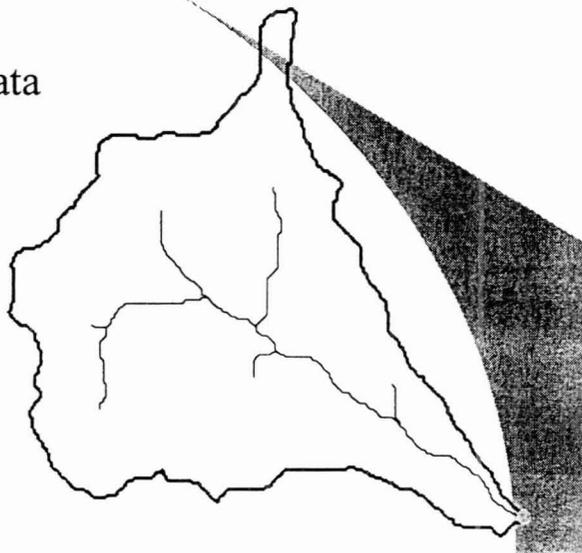
Feature Objects

- GIS Vector Data

Points

Lines

Polygons



Uses

DEM delineation

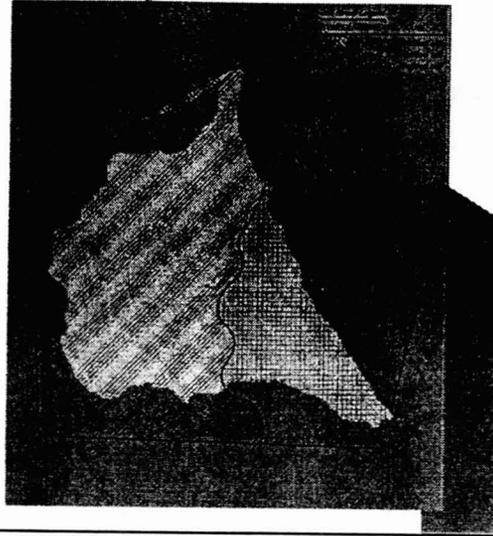
TIN delineation

Alone

Land use

Soil

Others

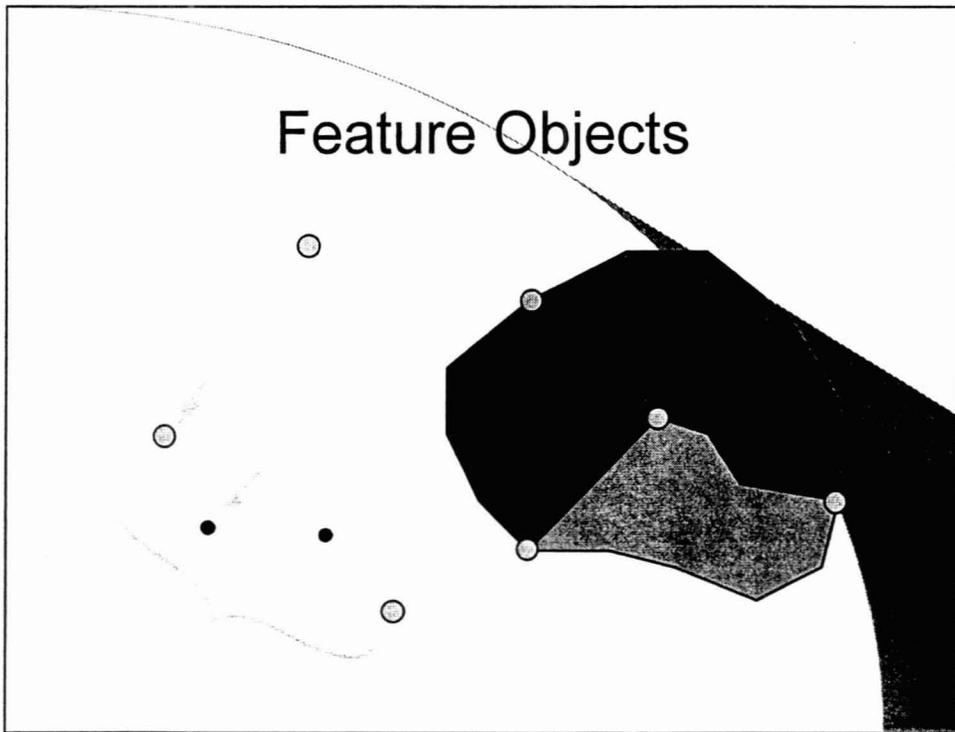




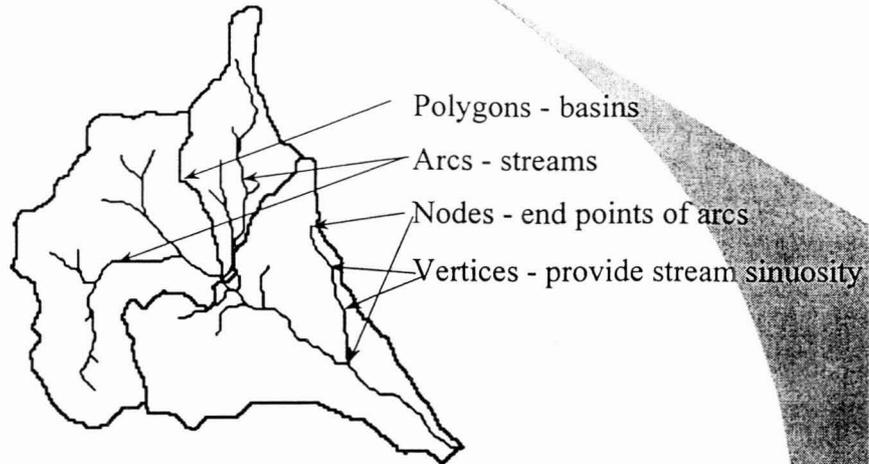
WMS Map Module

- At the heart of most WMS operations
 - GIS data
 - DEMs
 - TINs

Feature Objects

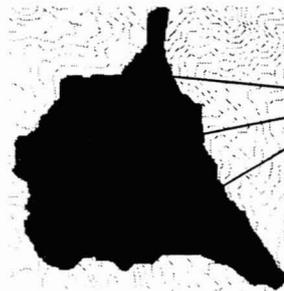


Watershed Feature Objects



Feature Object Attributes

- Attributes are associated with the point/node, arc, polygon



Attributes of Testplys.shp					
	Zone	Area	Perim	Centroid	Area
Polygon	1	14	0.41317	0.2803911000	4735.501
Polygon	1	15	0.71930	0.2997462477	7508.254
Polygon	1	16	0.88506	0.2903568905	7702.242

Purposes of Feature Objects

- To create watershed representations directly
- Store results of basin delineation with DEMs
- As a “conceptual model” for TIN creation
 - In version 5.1 TIN results can also be transferred back to feature objects (primarily for export reasons)
- Provide a common gateway to GIS data

Coverages

- Feature Objects can be grouped into sets or layers called "coverages".
- The attributes assigned to a feature object depend on the coverage type.
 - drainage, land use, soil type, etc.
- One coverage is designated as the "active" coverage. New objects are added to the active coverage. Only objects in the active coverage can be edited.

Coverage Types

- Generic
- Drainage (default)
- Land Use
- Soil Type
- Runoff Coefficient
- Time Computation
- Others...

Drainage Coverage

- Create watersheds and hydrologic models from feature objects and DEMs
- Create TINs of an appropriate resolution from another elevation source
- Primary coverage type for WMS
 - **IMPORTANT:** Only one drainage coverage type per model can exist

Time Computation Coverage

- Use arcs to specify runoff paths
- Define equations for each arc segment
 - TR55, FHWA, Maricopa County, Others
- Compute Time of Concentration or Lag Time for a basin
- Compute travel time between outlet points

Land Use Coverage

- Assign a land use id to each polygon
 - polygons can be imported from DXF or shape files

- Define a Land use table

ID, "Description of land use type", SoilACN, SoilBCN, SoilCCN, SoilDCN

1, "Fully developed urban area - open space - Poor", 68, 79, 86, 89

2, "Fully developed urban area - open space - Fair", 49, 69, 79, 84

3, "Fully developed urban area - open space - Good", 39, 61, 74, 80

4, "Paved parking lots, roofs, driveways, etc. (excl. ROW)", 98, 98, 98, 98

5, "Paved streets with curbs and storm sewers (excl. ROW)", 98, 98, 98, 98

6, "Paved streets with open ditches (incl. ROW)", 83, 89, 92, 93

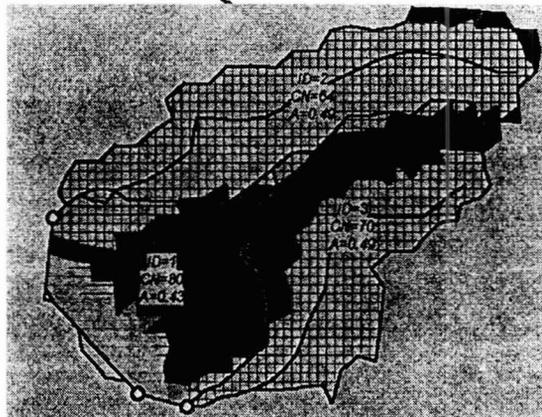
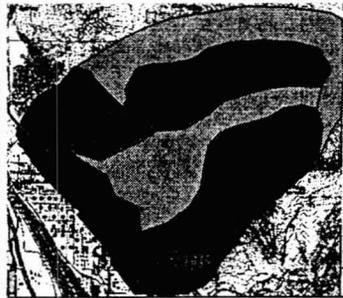
7, "Gravel streets", 76, 85, 89, 91

- Use with soil type coverage to define CN's of Green-Ampt parameters

Soil Type Coverage

- Assign a soil type or an ID to each polygon
 - hydrologic soil type A, B, C, D
 - polygons and their ID's can be imported from DXF or shape files
- Use with land use coverage to define CN's or Green-Ampt parameters
- Use with soil type ID's and a runoff coefficient table to compute runoff coefficients

Composite CN's



Runoff Coefficient Coverage

- Used to compute composite basin Runoff Coefficients
- A runoff coefficient is assigned to each polygon in the coverage
- When a runoff coefficient is assigned to a polygon, the basin runoff coefficients are automatically re-computed for the TIN

Others

- Generic
 - generally only an intermediate value
 - editing performed so conversion to a drainage coverage would be possible
- CASC2D
 - Used for definition of 2-D research models

Creating Feature Objects

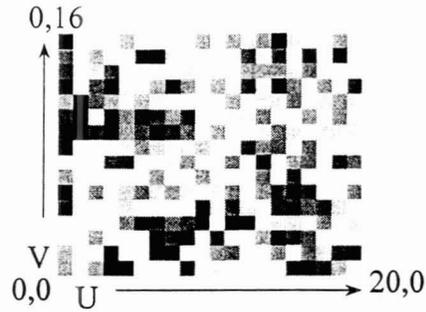
- On-screen Digitizing (point and click)
 - images - scanned topo maps
 - hill shaded DEMs - best if saved as an image first
 - contours
- Import (DXF, shape files, DLG)
 - shape files
 - DXF
 - DLG

Images

- TIFF Image
 - desktop scanner
 - DRG files developed by the USGS
 - private companies
- Must register the image using three points
- Used as a background map to digitize streams, land use polygons, etc.

Image Definition

- Pixel coordinates

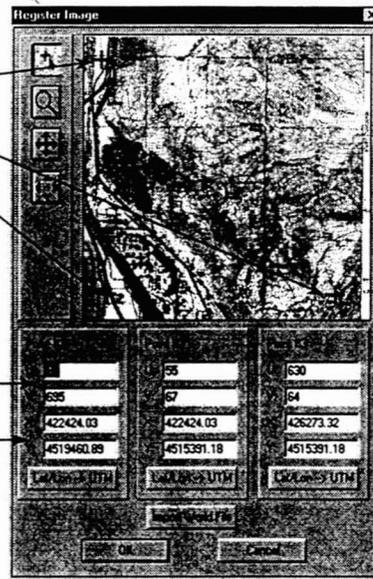


Registering Images

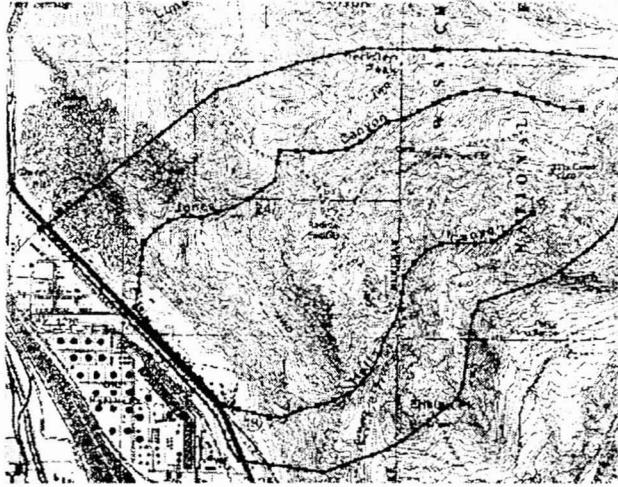
Registration Tics

Pixel Coordinates

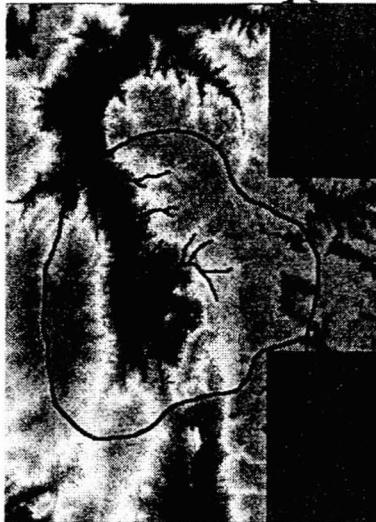
Real World Coordinates



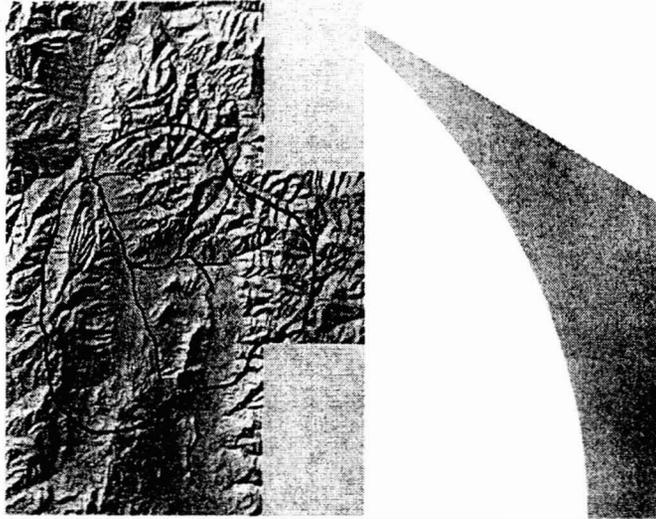
Using Images to Create Feature Objects



Using Contours to Create Feature Objects



Using a Shaded DEM to Create Feature Objects



Importing Existing Data to Create Feature Objects

- ArcView Shape files
- DXF Files
- DLG Files

ArcView Shape Files

- Can be created from ArcView or ARC/INFO
- Attributes imported depend on coverage type
- The ARCSHAPE command can be used to convert and Arc/Info Coverage to a shape file.
- Once a shape file has been imported you will want to convert to a WMS map file.

DXF Files

- Used as a background picture to digitize feature object on screen
- Converted directly to feature objects
 - Points
 - Lines
 - Polylines

Basic Feature Object Manipulation

The Map Module is at the heart of most operations in *WMS*, so it is important to gain a good understanding of how to create, edit, and apply feature objects. Within *WMS*, feature object data can be used for drainage characterization, land use, soil types, time of travel calculations, and many other applications. Often, the data you will want to use already exists in some other formats. This tutorial session will focus on the basic concepts behind manipulating feature objects and coverages. Also, this tutorial will discuss creating feature objects from background images.

2.1 Feature Object Definitions

Feature objects in *WMS* have been patterned after Geographic Information Systems (GIS) data objects and include points, nodes, arcs, and polygons (Figure 2-1).

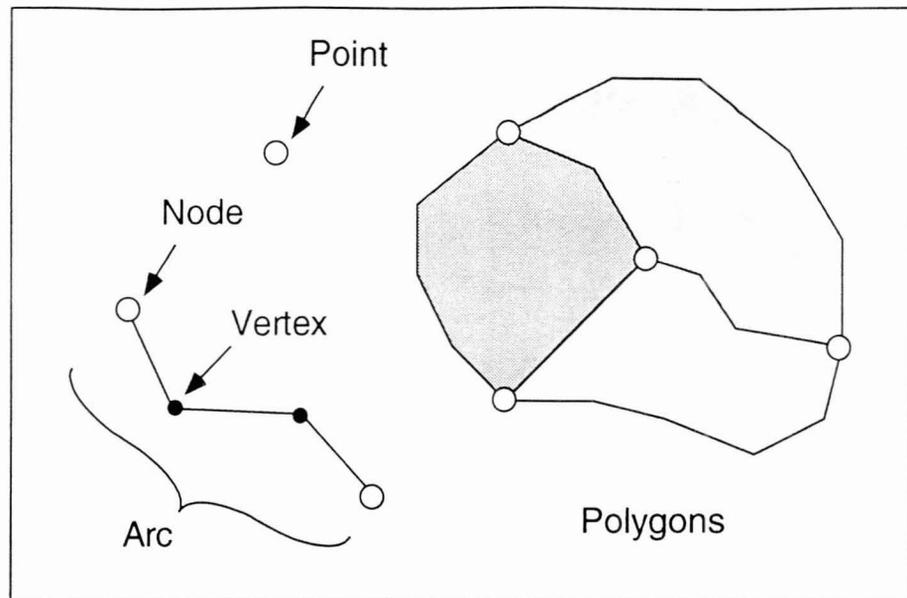


Figure 2-1 Feature Objects.

Feature objects can be grouped into coverages. Each coverage represents a particular set or group of data. The following coverage types can be used in WMS:

1. Drainage - defines basin boundaries and stream networks.
2. Land Use - defines land use type polygons
3. Soil - defines soil type polygons
4. Time Computation - used for determining lag times and times of concentration.
5. Generic
6. Other model specific coverages

In this tutorial you will learn how to create points, arcs, and polygons to develop these coverage types.

2.2 Display and Manipulation of Coverages

Because there are several different data layers represented by feature objects, it is important to understand a few rules about how they are created and edited. There is always one and only one active coverage at a time in WMS. Any time you wish to edit points, arcs, or polygons in a coverage, that coverage must be designated as the active coverage. Also, any new feature objects that are created are added to the active coverage. The active coverage is the only

one displayed in full colors. All feature objects of the inactive coverages are displayed in a single color (light gray by default).

In this section you will learn how to open a file and work with multiple coverages.

1. Select the Map module icon .
2. Open *asptutor.map*.
3. Select the *Coverages...* command from the *Feature Objects* menu.
4. Notice that for each coverage, the name, attribute set, active status, and visible status can be changed. To change the attributes for a coverage, select the coverage name from the text window and set its attributes.

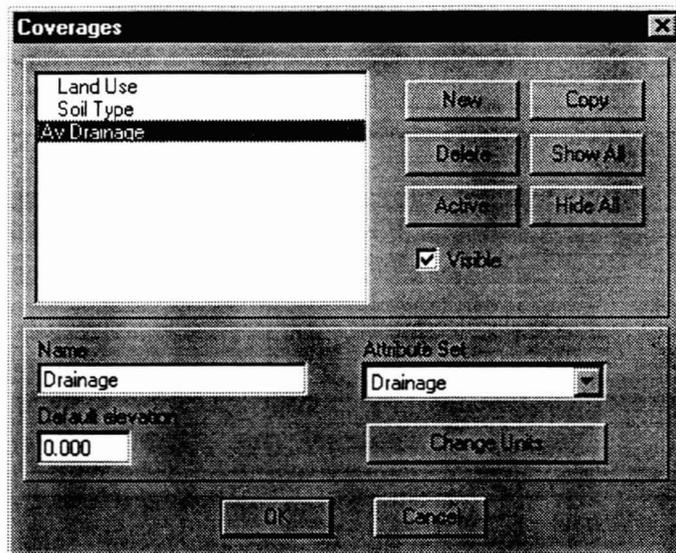


Figure 2-2 The Coverages Dialog.

5. Select the *Soil Type* coverage from the text window and turn off the *Visible* option. Then, select the *Land Use* coverage from the text window and turn off the *Visible* option. Select the *OK* button to exit the *Coverages* dialog. Notice that the only coverage displayed is the drainage coverage.
6. Select the *Coverages...* command from the *Feature Objects* menu. Set the *Soil Type* and *Land Use* coverages to visible. Select the *Land Use* coverage and select the *Active* button. Select the *OK* button to exit the *Coverages* dialog. Notice how the *Land Use* coverage is now drawn in different colors, while the other coverages are drawn in the inactive coverage color.

7. Select the select polygon tool , select one of the polygons in the *Land Use* coverage, and select the *Attributes...* command from the *Feature Objects* menu. Notice that the *Land Use Mapping* dialog comes up. This dialog can be used to assign land use ID's to selected land use polygons. The *Land Use Mapping* dialog can also be used to change the attributes assigned to each land use ID and to create new land use ID's. Select *Close* to exit the *Land Use Mapping* dialog.
8. Select the *Display Options* command from the *Display* menu.
9. Select the *Map* tab.
10. Turn on the *Land use legend* option and select the *OK* button. Notice that a land use legend is displayed on the screen.
11. The upper left corner of the edit window displays the active coverage and the drop-down combo box allows you to change the active coverage without invoking the *Coverages* dialog as demonstrated previously. Change the active coverage to the *Drainage* coverage by choosing it from the active coverages drop-down combo box in the *Edit* window.
12. Select the select polygon tool, select one of the polygons in the *Drainage* coverage, and select the *Attributes...* command from the *Feature Objects* menu. Notice that the *Drainage Feature Polygon Type* dialog comes up.
13. The *Drainage Feature Polygon Type* dialog can be used to change the type of a drainage polygon to a Generic, Drainage, or a Lake/Reservoir polygon. The polygon (as well as the point and arc) attribute dialog changes are based on the type of the active coverage and the selected tool. For example, if the active coverage is a land use coverage and the select polygon tool is selected, the *Land Use Mapping* dialog will come up. However, if the active coverage is a drainage coverage and the select polygon tool is selected, the *Drainage Feature Polygon Type* dialog comes up. Select the *Cancel* button to close the *Drainage Feature Polygon Type* dialog.
14. Select the *Display Options* command from the *Display* menu.
15. Select the *Map* tab.
16. Select the *Land Use* coverage from the drop-down list at the top of the dialog.
17. Toggle the *Land use legend* option off and select *OK*.
18. Select the *New* command from the *File* menu to delete all your data.

2.3 Using TIFF Images to Create Feature Objects

An image is an excellent way to provide a “background” picture from which streams or other features may be digitized directly from the screen. However, before an image can be used to define feature objects with accurate coordinates, it must be properly registered. All of the tools necessary to register and use images are available within the *WMS* interface.

There are three basic scenarios for registering images:

1. You must determine the correct coordinates and enter them manually in the image registration dialog. This is what is demonstrated in this tutorial and generally is applicable if you have scanned the image yourself using a desktop scanner.
2. Your image file (.tif) contains a corresponding tiff world file (.tfw). The tiff world file will have the correct registration information for the image and can be imported into *WMS*. The USGS digital raster graphics files (DRGs) have the .tfw files and most organizations that distribute them include the world files as well.
3. Many of the newer tiff files contain the registration information within the image file (.tif) itself. This is referred to as a geotiff file. When *WMS* reads a geotiff file it will bypass the registration window. The image will be registered automatically.

You can obtain TIFF images using a standard desktop scanner, by purchasing from a digital data provider, or from some locations on the internet (see <http://www.emrl.byu.edu/gishydrodata/images.htm> for a list of some of these sites). The image used for this tutorial was obtained from a USGS digital raster graphics (DRG) file.

2.3.1 Importing and Registering a TIFF Image

To import and register the TIFF image being used in this tutorial do the following:

1. Select the *Map* module icon .
2. Select the *Import* command from the *Images* menu.
3. Use the file browser to locate and open the file named “*jonescyn.tif*”. By default this file is found in the tutorial directory.

After a few seconds of reading the file an image of a portion of USGS quad map will appear in the *Register Image* dialog as shown in Figure 2-3.

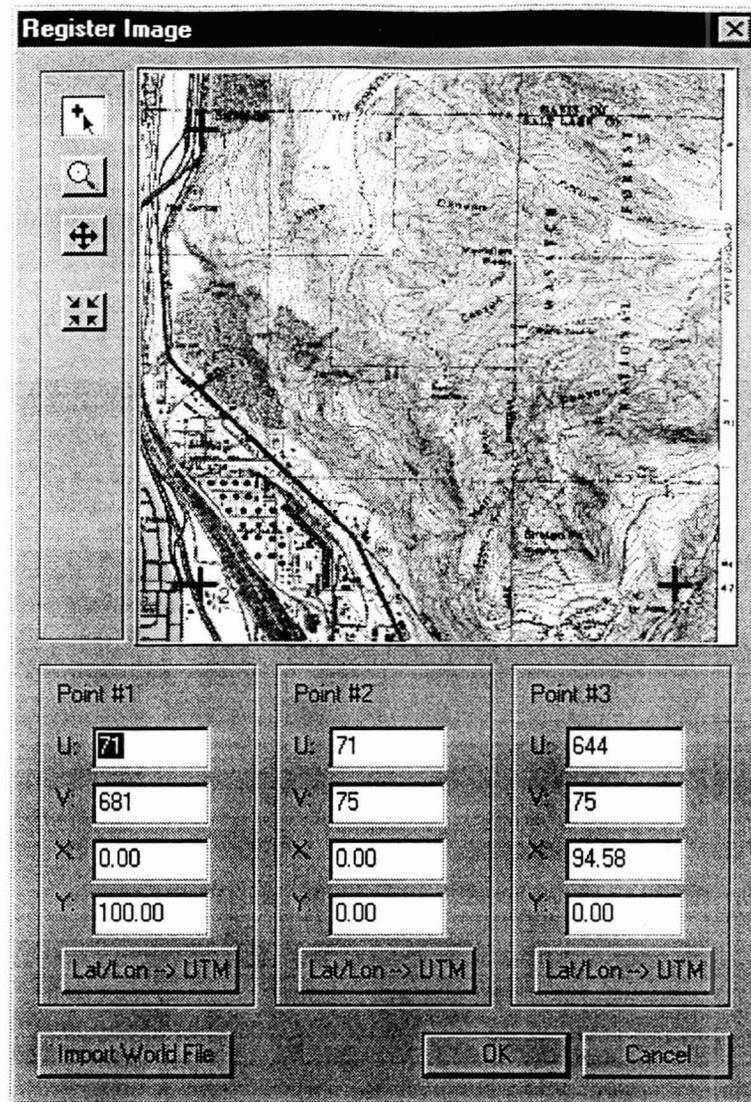


Figure 2-3 Image Registration Dialog.

Pixel coordinates of the image must be registered to proper ground locations so that feature objects created by on-screen digitization will have the proper coordinates.

- Set the values for the u,v and x,y pairs of each point according to the table below and select the *OK* button:

Registration Points

	Point 1	Point 2	Point 3
U	0	0	715
V	756	0	0
X	422424	422424	426273
Y	4519461	4515391	4515391

The image will be resampled so that as much of the image as possible is fit into the Graphics window (an image will often be of a higher resolution than the screen). After a few seconds you should see the image in the Graphics window.

2.3.2 Displaying and Manipulating Images

1. Select the zoom tool .
2. Zoom in on any portion of the image.

The image will become blurry since there are now more screen pixels than there are pixels in the portion of the image being displayed.

1. Select the pan tool .
2. Pan to a different portion of the image by dragging the cursor in the Graphics window.
3. Select the *Resample* command from the *Image* menu.

The image will become clearer after it is resampled.

1. Pan the image by dragging the cursor in the Graphics window.

The image will not cover the entire screen until another resample command is given. To summarize how resampling works with the zoom and pan tool:

- If you zoom in on an image it will become more blurred (unless your image has a lower resolution than the screen, or you have zoomed in and resampled to the point that the region of the image you are looking at has a lower resolution than the screen).
- You can pan an image if you zoom in without resampling.
- After resampling, the image will become as clear as possible, and fit the Graphics window.

After any sequence of zooming, panning, and resampling you can restore the original image by doing the following:

1. Select the *Fit Entire Image* command from the *Image* menu.

2. Select the *Resample* command from the *Image* menu.
3. Select the frame macro .

2.3.3 Using an Image to Define Feature Objects

As mentioned previously, the TIFF image you have imported is a scanned image of a USGS quadrangle map. This particular map corresponds to the northwest corner of Salt Lake City, Utah.

1. Select the *Create Point* tool .
2. Click on the point labeled as the beginning boundary point in Figure 2-4.
3. Select the *Create Arc* tool .
4. Select the *Attributes* command from the *Feature Objects* menu.
5. Choose the *Generic* arc type and select the *OK* button.
6. Define a single arc, which will serve as the boundary, by clicking on the feature point already created and then along several other points of the polygon shown in Figure 2-4 until you close the arc on itself by clicking on the first point a second time.
7. Select the *Attributes* command from the *Feature Objects* menu.
8. Choose the *stream arc* type and select the *OK* button.
9. Using the streams/road from the image in Figure 2-4 as a guide enter the two stream branches starting from the point marked "outlet" in Figure 2-4. Be sure to double-click to end the entry of a stream segment.

You may need to zoom and/or pan and resample the image in order to see features more clearly.

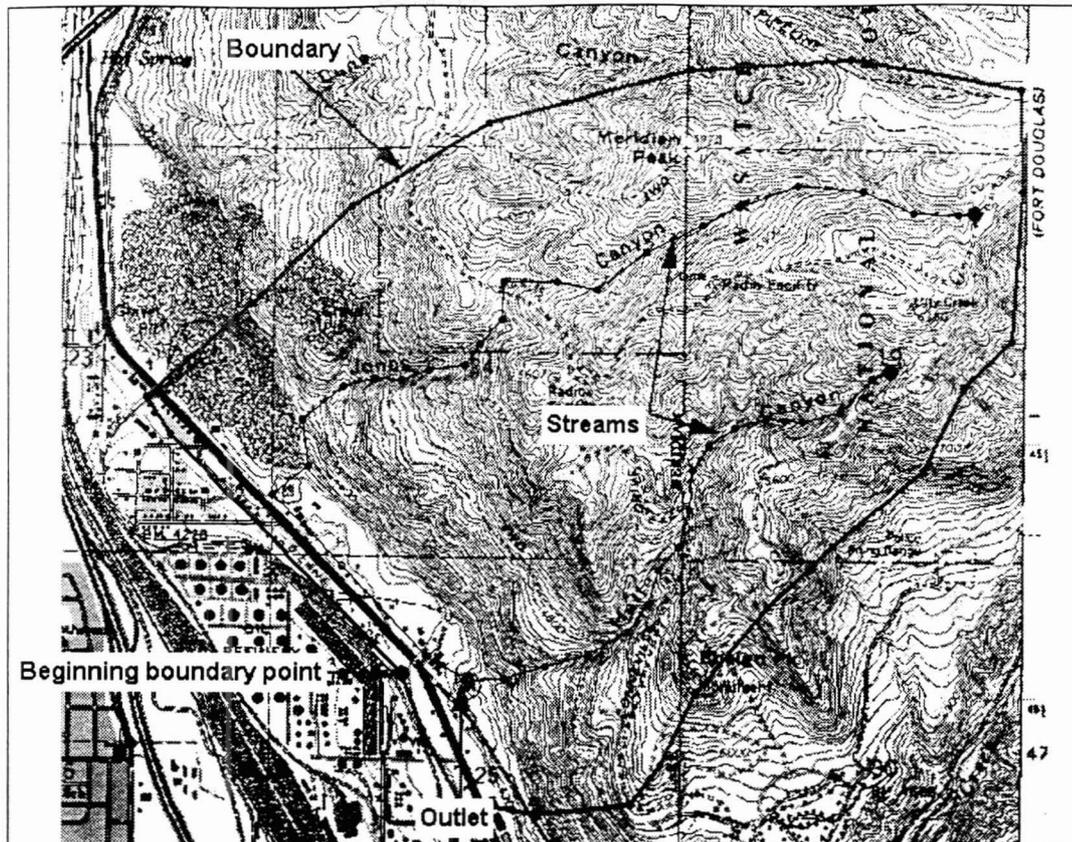


Figure 2-4 Completed Feature Objects for Jones Canyon Watershed.

If you have troubles defining the feature objects you can read in the map file named "jonescyn.map" to see what the final boundary and streams should look like.

2.3.4 Saving a File and Deleting the Feature Objects

If you would like to save your feature objects for later use in creating a TIN:

1. Select *Save As...* from the *File* menu.
2. Click on the *Maps* toggle box.
3. Turn off any other toggle box that may be on, including the super file, display options file, tree file, and image registration file. Since the image file has already been saved, you only need to save the feature object data at this time.
4. Click on the directory button and specify a directory where you want to save your files.
5. Enter "jones1.map" for the filename.

6. Hit *Save* to exit the *Save Files* dialog and save the map file.

Now that you have saved this map file you may delete these feature objects by doing the following commands:

1. Select *New* from the *File* menu.
2. Select *OK* to confirm that you want to delete all data.

2.4 Conclusion

This concludes the *Basic Feature Object Manipulation* tutorial. In this tutorial, you have learned some of the basic concepts behind manipulating feature objects and coverages. You have also learned how to digitize feature objects from background images.

Advanced Feature Object Manipulation

The previous tutorial focused on some basic concepts needed for creating and editing feature objects. This tutorial will focus on more advanced topics. First, you will learn about tools for cleaning and editing feature objects. Then, you will learn how to import and use data in ArcView® Shapefile and DXF format as feature objects in WMS.

3.1 Cleaning and Editing Feature Objects

Two scenarios can exist when using feature objects in WMS. The first is that you have feature objects in another format that you have read into WMS. The second is that you use WMS to develop your data. In this section, you will first learn to import and clean existing data. Then, you will learn some techniques for creating feature object data in WMS.

3.1.1 Importing and Cleaning Feature Object Data

This section will demonstrate various options for cleaning feature objects in WMS. Options to intersect selected arcs, snap selected nodes, and remove dangling arcs will be discussed. Then, the arcs will be converted to streams for input into a watershed model.

Importing the Feature Object Data

1. Select the *Open* command from the *File* menu.

2. Find and open “*arco.sup*”. It may take a few seconds for the superfile to read in.
3. Switch to the *Map* module by clicking on the *Map* icon  at the top of the *Toolbox*.
4. Select the *Resample* command from the *Images* menu to resample your image to the size of the screen.
5. Select the *Import* command from the *File* menu.
6. Select the *Shape file->Feature objects* option and select the *OK* button.
7. In the *Import Shapefile Data* window, select the *arcs* file browser button.
8. Find and open “*arco.shp*”.

HINT: In the arc attribute text window, notice that the *DRAINTYPE* database field is automatically mapped to the *Drainage Arc type* attribute in *WMS*. This occurred only because the attribute name in the stream file was *DRAINTYPE*. If the drainage type attribute name is anything other than *DRAINTYPE* it can still be assigned to the *WMS* Drainage Arc type variable manually using the *Attribute mapping...* button.

9. Select *OK* and wait a few seconds while the stream shapefile for the Arco watershed is read in and displayed.

Intersecting Selected Arcs

There are two arcs that overlap each other in this model. Overlapping arcs are not allowed in *WMS*, so you should intersect these arcs.

1. Select the *zoom* tool  and drag a box around the location of the intersecting arcs. The intersecting arcs are marked with an arrow that is labeled “*Intersect*”, as in Figure 3-1.
2. Select the *Resample* command from the *Images* menu to resample your image to the size of the screen.
3. Select the *Select Arcs* tool  in the toolbox.
4. Find and use *SHIFT-select* to select the two intersecting arcs (See Figure 3-1).

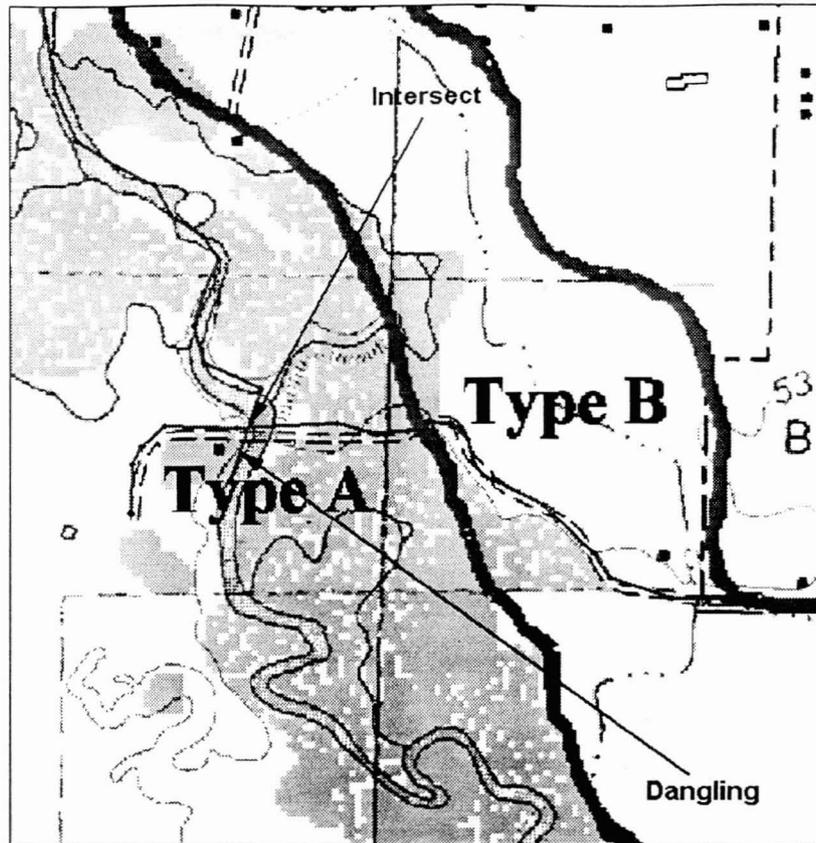


Figure 3-1 Cleaning Intersections and Dangling Arcs.

5. Select the *Clean...* command from the *Feature Objects* menu.
6. Make sure the *Intersect selected arcs* option is selected and select the OK button. The two selected arcs will be intersected and a node will be created at their intersection point.

Removing Dangling Arcs

After intersecting the overlapping arcs, a small dangling arc was created. Deleting a dangling arc is easy... simply select the arc and delete it.

1. Find and select the dangling arc (See Figure 3-1).
2. Select the *Delete* command from the *Edit* menu. If prompted, select *OK* to delete the selected arc.

Snapping Selected Nodes

Occasionally, arcs or nodes on arcs that should be connected are not. Sometimes, these problems can be fixed by intersecting all the overlapping arcs and removing the dangling arcs using the automatic clean functions in

WMS. Other times, you need to snap a node or vertex on one arc to a node or vertex on another arc.

1. Select the *Frame Image* macro .
2. Select the *zoom* tool  and drag a box around the location of the nodes that need to be snapped together. The nodes that need to be snapped are marked with an arrow that is labeled "Snap", as in Figure 3-2. You may have to zoom in very close to see that the nodes are not connected to each other.
3. Select the *Resample* command from the *Images* menu to resample your image to the size of the screen.
4. Select the *Select Point/Node* tool .
5. Select the node at the end of the arc to snap (See Figure 3-2).

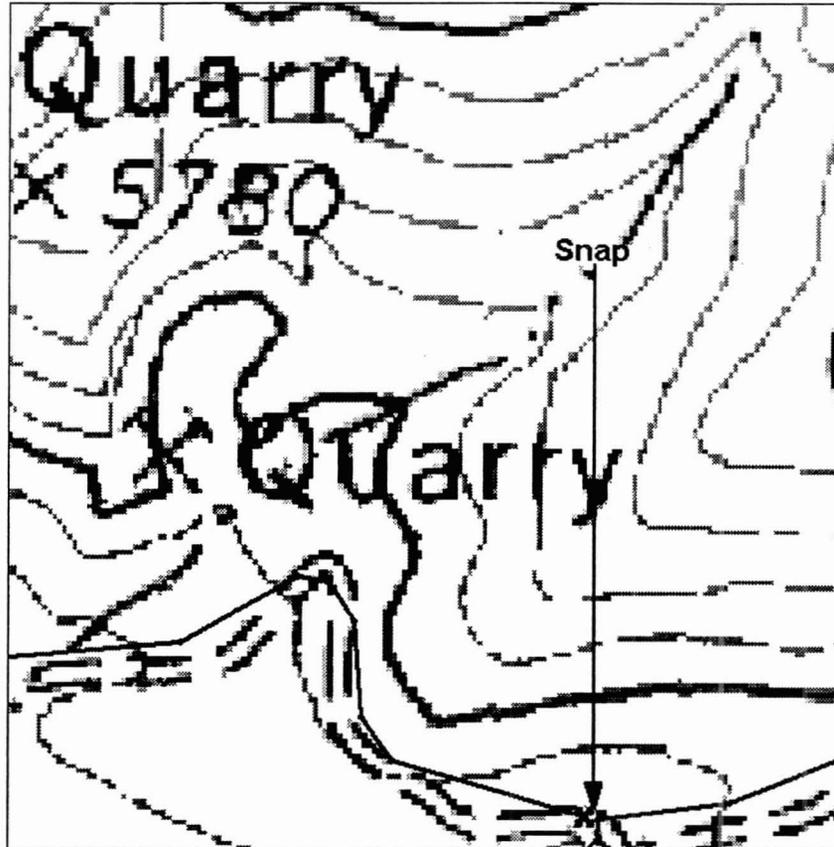


Figure 3-2 Snapping Selected Nodes

6. Select the *Clean...* command from the *Feature Objects* menu.

7. Make sure the *Snap selected nodes* option is selected and select the OK button. You will be prompted to select a snapping point. Select a vertex on the arc adjacent to the selected node. The two points will be snapped together and a node will be created at the intersection of the arcs.

Creating Streams from Existing Feature Objects

Now that the arcs are cleaned, you are ready to convert your generic arcs to stream arcs.

1. Select the *Frame Image* macro .
2. Select the *Resample* command from the *Images* menu to resample your image to the size of the screen.
3. Select the *zoom* tool  and drag a box around the location of the downstream point. The downstream point is marked with an arrow that is labeled "Downstream point", as in Figure 3-3.

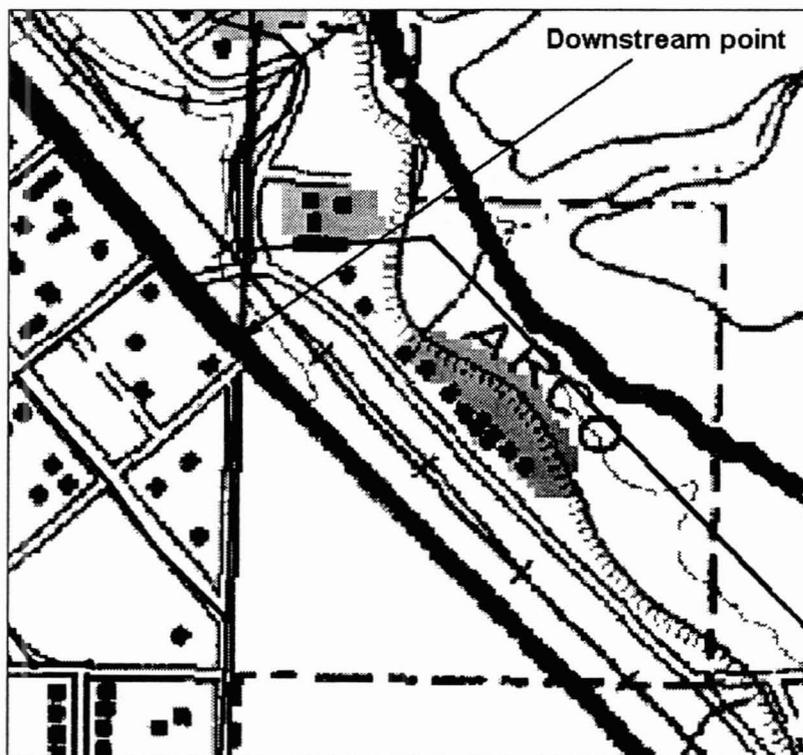


Figure 3-3 Reordering Streams.

4. Select the *Resample* command from the *Images* menu to resample your image to the size of the screen.

5. Select the *Select Point/Node* tool .
6. Select the downstream node (See Figure 3-3).
7. Select the *Reorder Streams* command from the *Feature Objects* menu.
8. Select the *Frame Image* macro .
9. Select the *Resample* command from the *Images* menu to resample your image to the size of the screen.
10. Select the *Select Stream Network* tool .
11. Select any location on the arc network.
12. Select the *Attributes...* command from the *Feature Objects* menu. Change the current feature arc type to *Stream* and select the *OK* button. The stream network will be generated and linked to the hydrologic modeling tree.

3.1.2 Creating Feature Objects from Images

Sometimes, you may have existing GIS data that you have been able to read into *WMS*. But the data may not have all the attributes required for your model. In addition, some data may not be in a format usable by *WMS*. Some data, such as land use data or soil type data, must be digitized in *WMS* as feature objects before it can be put to use.

Creating and Removing Attributes

In this section, you will learn how to convert a vertex to a node and how to assign an attribute to that node. Recall that each arc consists of two nodes and any number of vertices. The vertices define the shape of the arc and the nodes define the arc's endpoints. Nodes can have attributes, but vertices cannot.

1. Select the *Select Feature Vertex* tool .
2. Use **SHIFT**-select to select the two vertices labeled as outlet points.

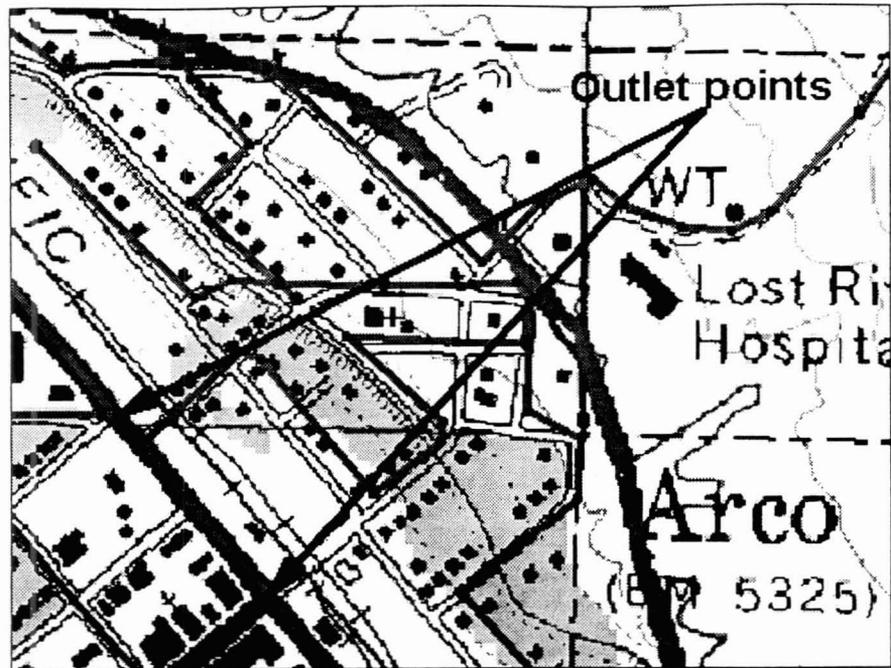


Figure 3-4 Creating Outlet Points.

3. Select the *Vertex <-> Node* command from the *Feature Objects* menu.
4. Select the *Select Point/Node* tool .
5. Use SHIFT-select to select the two nodes labeled as outlet points.
6. Select the *Attributes...* command from the *Feature Objects* menu.
7. Change the attributes of the selected nodes to *Drainage outlet* and select the *OK* button. The newly created outlets will be linked to the hydrologic modeling tree.

Creating Soils Data from Image Data

To compute composite curve numbers or runoff coefficients, you need a soil coverage defining the boundaries of each soil in your watershed. In the Arco watershed model, the boundaries of each soil are defined in the image, but not as a coverage.

1. Select the *Coverages...* command from the *Feature Objects* menu.
2. Select the *New* button in the *Coverages* dialog.
3. Change the name of the new coverage to *Soil Type* and change the coverage's *Attribute set* to *Soil Type*.
4. Select the *OK* button in the *Coverages* window.

5. There are now two coverages: The drainage coverage and the soils coverage. Your next task will be to create arcs and polygons defining the soils coverage. Select the *Create Feature Arc* tool .
6. Create a rectangular boundary around your entire model by selecting one corner. Then select each of the remaining corners and create a single closed arc around your entire model, as shown in Figure 3-5.

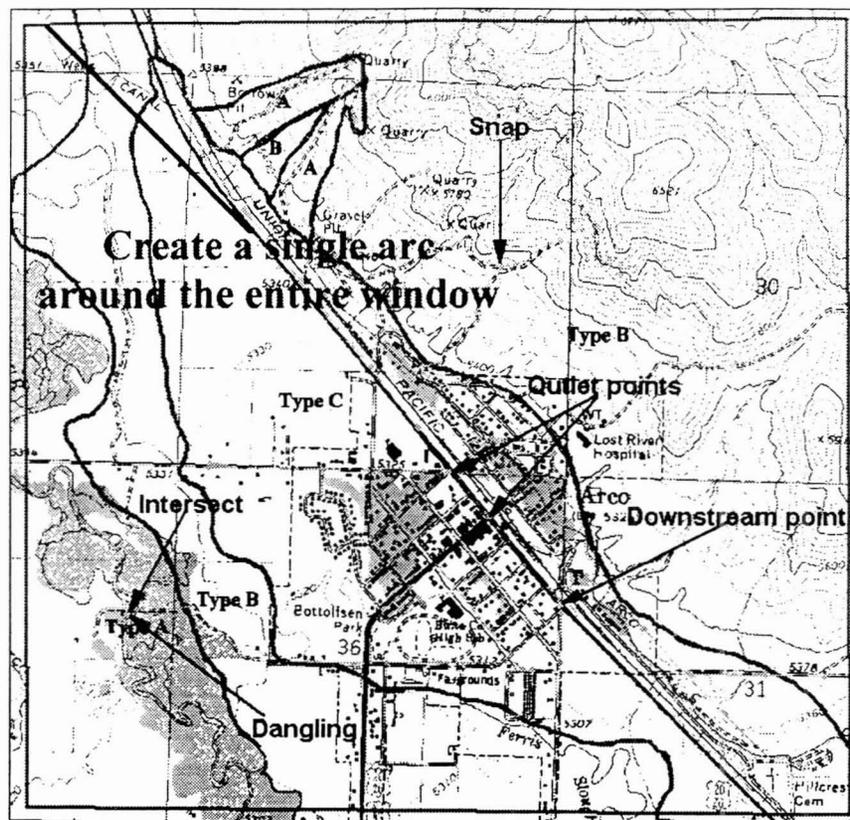


Figure 3-5 Step 1—Create a Single Arc around the Entire Window.

7. The pink lines in the image represent soil type boundaries. Select a point on the rectangular boundary that intersects with one of the pink lines and create an arc along the pink line until another boundary line is reached.

HINT: If you click close to an existing arc when creating an arc, the arc you are creating will automatically snap to that existing arc. All the arcs you create in a soil type coverage must be connected to each other and form closed polygons. If closed polygons are not created, WMS will not be able to determine the type of soil for each point in your watershed model.

8. Create arcs from the other soil boundary lines, as shown in Figure 3-6. Be sure all the soil arcs form closed polygons. You don't have to be too accurate when creating the soil boundary arcs. If you are creating a real model, of course, you should try to be more accurate.

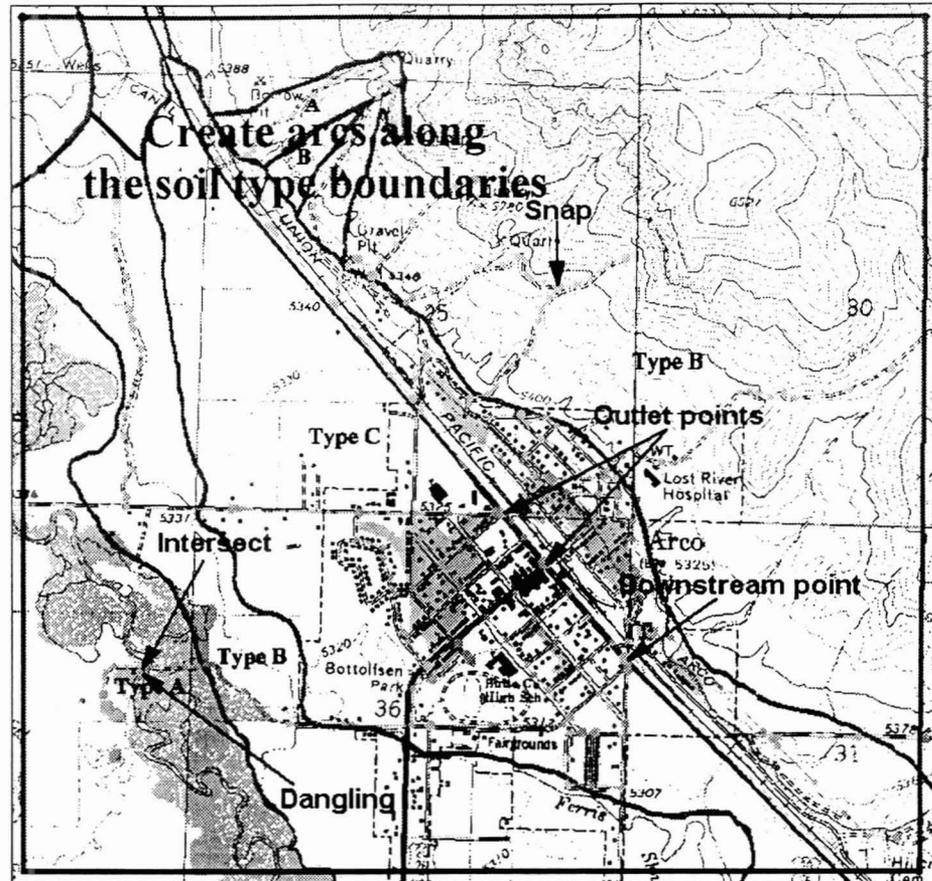


Figure 3-6 Step 2—Create Arcs Along the Soil Type Boundaries.

3.1.3 Building Polygons

Now that you have defined the boundaries for the soils in your coverage, you are ready to build polygons and assign soil attributes to each polygon. Polygons must be built to assign soil types to areas in the soil coverage.

1. Select the *Build Polygon* command from the *Feature Objects* menu.
2. When prompted if you want to use all arcs to build the polygons, select *OK* to build the polygons. If you built the soil arcs correctly, polygons will be generated from your soil arcs.
3. Select the *Select Polygon* tool . This tool must be selected to access the polygon attributes for the land use coverage.

4. Notice that the type of each soil is written on the background image. Use SHIFT-select to select all the soil type A polygons and select the *Attributes...* command from the *Feature Objects* menu.
5. Note in the *Soil type mapping* dialog (Figure 3-7) that SCS soil type A is the default soil type when a soil polygon is created. Select the *Apply* button to assign this default value to all the selected polygons.

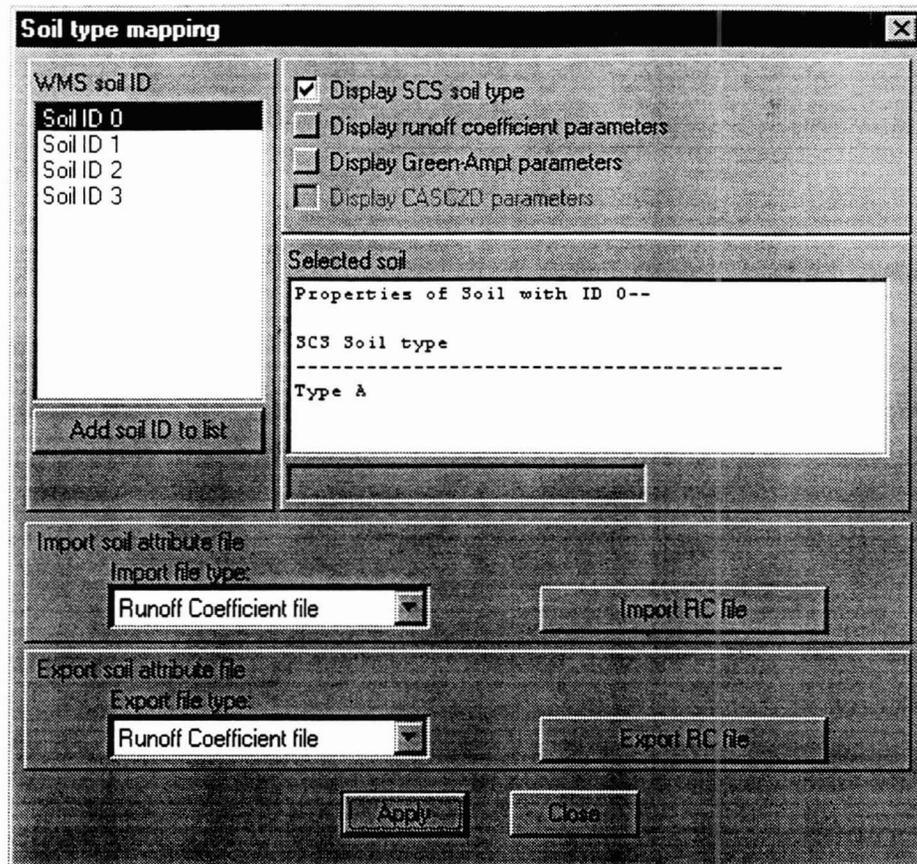


Figure 3-7 The Soil Type Mapping Dialog.

6. Perform steps 4 and 5 for soil types B, and C. For soil type B, you will need to change the *WMS soil ID* to *Soil ID 1*. For soil type C, you need to change the *WMS soil ID* to *Soil ID 2*.
7. Select the *Display Options...* command from the *Display* menu.
8. Toggle the *Color fill polygons* option on and select the *OK* button.
9. Select the *New* command from the *File* menu.
10. Select *OK* to confirm that you want to delete all data.

3.2 Importing Feature Objects from Arc/Info (ArcView) Shape Files

You have now learned how to create watershed boundaries and streams in the map module from digital image data and digital elevation data. What if you already have watershed boundaries, streams, or both in a Geographic Information System (GIS) and you want to read this data into WMS? Or what if you have soil type and land use information stored in a GIS?

WMS provides tools to import and export ArcView® GIS shapefiles. Some basic functions of these tools will be discussed in this section. For more information on where you can download these data see: www.emrl.byu.edu/gishydrodata.

3.2.1 Reading a WMS Map Drainage File

First, you will read in a map file containing basins and streams in the Aspen Grove, Utah watershed.

1. Select the *Open...* command from the *File* menu.
2. Find and open *aspencn.map*, a map drainage file of a watershed containing streams, basin boundaries, and outlet points.

3.2.2 Importing a Soil Type Shape File

In this section, you will perform the next step in computing a composite curve number—importing a soil type shapefile.

1. Select the *Import...* command from the *File* menu.
2. Select the *Shape file->Feature objects* option and select the *OK* button.

HINT: It is extremely important that you create a new soil type coverage when importing a soil type shapefile or any other type of attribute shapefile. After creating the new coverage, you must specify the coverage attribute type as explained below.

3. Select the *Coverage options...* button in the *Import Shapefile Data* dialog.
4. Select the *New* button in the *Coverages* window.
5. Change the name of the coverage to *Soil Type* and change the coverage's *Attribute set* to *Soil Type*.

6. Select the *OK* button in the *Coverages* window.
7. In the *Import Shapefile Data* window, select the *polygons* file browser button (see Figure 3-8).

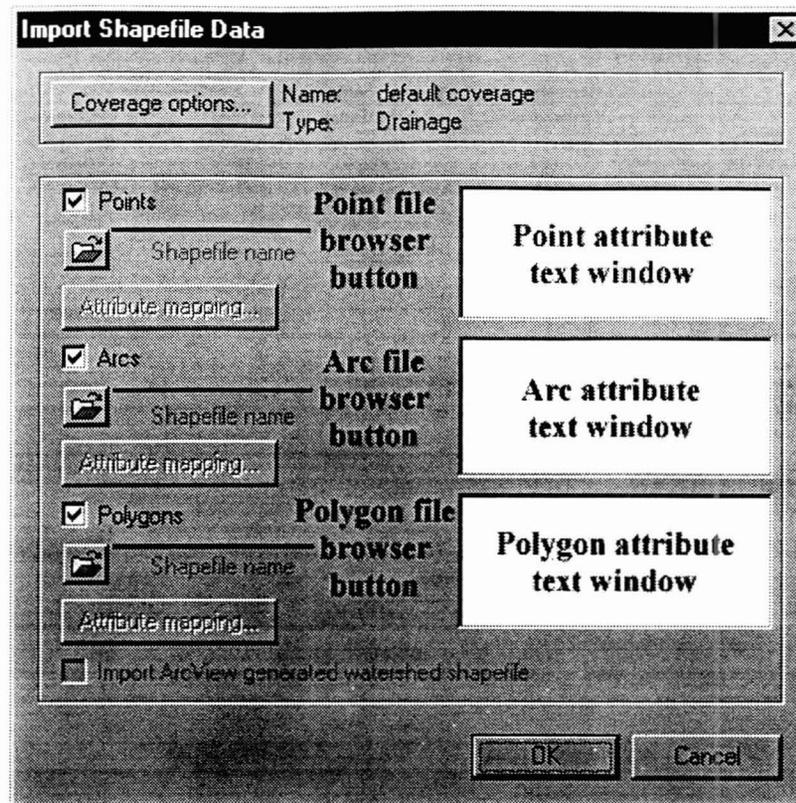


Figure 3-8 The *Import Shapefile* Dialog.

8. Find and open "*aspenso.shp*".
9. In the polygon attribute text window (see Figure 3-8), notice that the *HYDGRP* database field is automatically mapped to the *SCS soil type* attribute in WMS. This occurred only because the attribute name in the soil type file was *HYDGRP*. If the attribute name is anything other than *HYDGRP* it can still be assigned to the *WMS SCS soil type* variable manually using the *Attribute mapping...* button.
10. Select *OK* and wait a few seconds while the soil type shapefile for the Aspen Grove watershed is read in and displayed.

3.2.3 Importing a Land Use Shape File

After importing or creating your watershed model, you can import other GIS data to compute hydrologic parameters such as curve numbers or runoff

coefficients. To compute a curve number, for example, you need land use and soil type coverages. You will now import a land use shapefile.

1. Select the *Import...* command from the *File* menu.
2. Select the *Shape file->Feature objects* option and select the *OK* button.

HINT: It is extremely important that you create a new land use coverage when importing a land use shapefile or any other type of attribute shapefile. After creating the new coverage, you must specify the coverage attribute type as explained below.

3. Select the *Coverage options...* button in the *Import Shapefile Data* dialog.
4. Select the *New* button in the *Coverages* window.
5. Change the name of the coverage to *Land Use* and change the coverage's *Attribute set* to *Land Use*.
6. Select the *OK* button in the *Coverages* window.
7. In the *Import Shapefile Data* window, select the *polygons* file browser button.
8. Find and open "*aspenlu.shp*".
9. In the polygon attribute text window, notice that the *LU_CODE* database field is automatically mapped to the *Land use* attribute in *WMS*. This occurred only because the attribute name in the land use file was *LU_CODE*. If the attribute name is anything other than *LU_CODE* it can still be assigned to the *WMS Land Use* variable manually using the *Attribute mapping...* button.
10. Select *OK* and wait a few seconds while the land use shapefile for the Aspen Grove watershed is read in and displayed.

3.2.4 Assigning Attributes to Feature Objects

When reading in your land use and soil type shapefiles, an ID is read in for each land use and soil type polygon. However, you still need to be able to compute a curve number for each basin in the Aspen Grove watershed. But how do you relate the soil type ID and land use ID in each of these polygons to a curve number?

WMS stores a separate table for each land use ID. This table relates each land use ID to a land use name, a curve number for each SCS soil type (A, B, C,

and D), and other land use properties. When *WMS* computes the curve number for each basin, it determines the soil type and land use for several locations in the sub-basin. *WMS* then determines the curve number for each of these locations using the land use table. Fortunately, *WMS* provides an easy method of setting up this land use table in the land use polygon attributes dialog.

In this section, you will learn how to set up a land use table using the land use polygon attributes dialog.

1. Switch to the *Map* module by clicking on the *Map* icon  at the top of the *Toolbox*.
2. In the *Coverage* box in the *Edit Window*, make sure the "Land Use" coverage is the active coverage.
3. Select the *Display Options...* command from the *Display* menu.
4. Toggle the *Land use legend* option on and select the *OK* button to close the *Land Use Feature Object Display Options* dialog. Notice that a land use legend appears on your graphics window (see Figure 3-9). Three land use ID's are shown on the legend. You must assign data to each of these land use ID's.

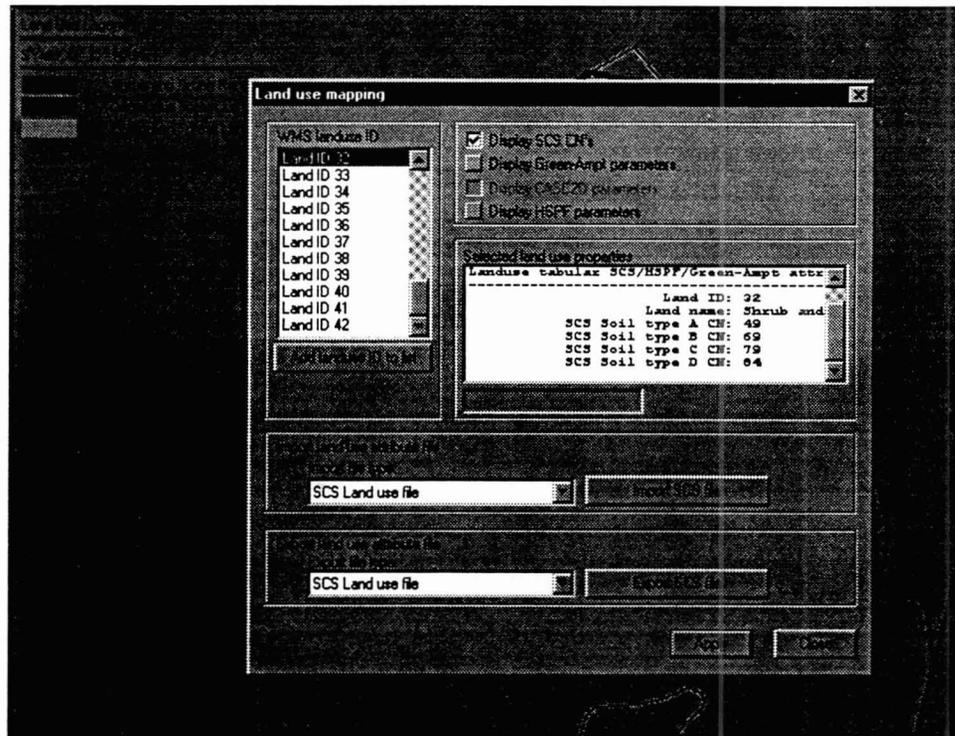


Figure 3-9 The Land Use Polygon Attributes (Land Use Mapping) Dialog.

5. Select the *Select Polygon* tool . This tool must be selected to access the polygon attributes for the land use coverage.
6. Select the *Attributes...* command from the *Feature Objects* menu.
7. In the *Land use mapping* dialog window, select Land ID 32 from the *WMS landuse ID* window. In the *Selected land use properties* window, select the *Land name* row and change the name of the land use to "Shrub and Brush Rangeland", as in Figure 3-9.
8. In the *Selected land use properties* window, select the *SCS Soil type A CN* row and change the Type A CN Value to 49. Select the *SCS Soil type B CN* row and change the Type B CN Value to 69. Select the *SCS Soil type C CN* row and change the Type C CN Value to 79. Select the *SCS Soil type D CN* row and change the Type D CN Value to 84.
9. Repeat steps 8 and 9 for soil types 41 and 42 with the following Curve number and name values:

<i>Property</i>	<i>Land use ID 41</i>	<i>Land use ID 42</i>
<i>Name</i>	<i>Deciduous Forest Land</i>	<i>Evergreen Forest Land</i>
Type A CN	45	36
Type B CN	66	60
Type C CN	77	73
Type D CN	83	79

10. In the *Export file type* field, select the option to export an SCS Land use file. Select the *Export SCS file* button, go to the directory where you want to save the land use file, and call the SCS land use file "aspencn.tbl". After exporting, this file can be used later without having to set it up again.
11. Select the *Close* button to close the *Land use mapping* dialog.
12. In the *Coverage* box in the *Edit Window*, make sure the "Drainage" coverage is the active coverage.
13. Switch to the *Tree* module by clicking on the *Hydrologic Modeling* module icon  at the top of the *Toolbox*.
14. Select the *Composite Runoff Coeff/CN...* command from the *Calculators* menu.
15. Select *SCS Curve numbers* for the *Computation type* and select the *OK* button to compute the composite Curve numbers for each basin. *WMS* will take a few seconds while the curve numbers for each basin are computed. If you want to display the curve numbers for each sub-

basin, you can turn them on from the *Display Options* dialog under the *Drainage* tab.

16. Switch to the *Map* module by clicking on the *Map* icon  at the top of the *Toolbox*.
17. In the *Coverage* box in the *Edit Window*, select the "Land Use" coverage to make it active.
18. Select the *Display Options...* command from the *Display* menu.
19. Toggle the *Display land use legend* option off and select the *OK* button to close the *Land Use Feature Object Display Options* dialog.
20. Delete all your data by selecting the *New* option from the *File* menu. Select *OK* to delete all your data.

3.3 Creating Feature Objects from DXF Data

DXF data files can be imported and used in WMS for creating streams or other feature object data in two different ways: 1) direct conversion, or 2) using them as a background map. DXF files from any application can be used, but must be in R-12 format. See the user manual for help in converting to R-12 format if you have problems reading a particular DXF file.

3.3.1 Creating a Boundary Polygon

Although the DXF file can contain data for the boundary of the watershed, the file that you will import contains data for the stream. The boundary that will be used in this portion of the tutorial will be read from an existing map file (".map"). To read the file:

1. Select *Open* from the *File* menu.
2. Using the file browser, find and open "*aspenbnd.map*."
3. Select *YES* if prompted to delete the current feature objects.

3.3.2 Importing a DXF File

To import the DXF file:

1. Select *Import* from the *DXF* menu.
2. Find and *Open* the file named "*aspen.dxf*."

The file will be imported and then displayed in the main window. The next two sections illustrate the two different ways DXF data can be used to create feature objects.

3.3.3 Digitizing Feature Objects From DXF Data

The first way DXF data can be used to create feature objects is by simply digitizing new arcs/polygons over the top of the existing data.

1. Select the *Create Arc* tool .
2. Select the *Attributes...* command from the *Feature Objects* menu.
3. Change the arc attribute type to *Stream* and select *OK*.
4. Create the arc along the main stream by clicking on the outlet point (lower right) and then clicking on additional points along the DXF stream data. Be sure to click (create a vertex) at locations along the main stream at branching locations.
5. Finish the stream arc by double clicking on the last point.

Repeat steps 4 and 5 for each of the stream branches. If you missed placing a vertex at one of the stream branches you can:

1. Select the *Create Feature Vertex* tool .
2. Click on any locations along the stream arc where the stream branches.
3. Select the *Create Feature Arc* tool .

If you wish to save the completed feature objects to a map file you may do so, but you must delete the feature objects before proceeding to the next section.

1. Select the *Delete* command from the *Feature Objects* menu.
2. Select *OK* to confirm that you want to delete the feature objects.

3.3.4 Converting DXF Data to Feature Objects

A more direct approach, especially for large data files, is to convert the DXF data to Feature Objects. This is done using the following steps:

1. Select the *DXF->Feature Objects* command from the *DXF* menu.

2. Turn on the toggle to convert arcs from dxf lines and turn off the toggle to convert points from dxf points (nodes for arcs will automatically be placed at the beginning and ending locations of the dxf lines).
3. Select the option to *Add to currently active coverage*.
4. Select *OK*.

Since the DXF line segments may not be ordered properly for stream arcs (connected from downstream to upstream) they are defined as generic arcs. You must now reorder the arcs for streams, and then convert them to stream type.

1. Select the *Select Points/Nodes* tool .
2. Select the downstream most point in the lower right of the main window.
3. Select the *Reorder Streams* command from the *Feature Objects* menu.
4. Select the *Select Network* tool .
5. Select any one of the stream arcs.
6. Select the *Attributes* command from the *Feature Objects* menu.
7. Choose the stream type option and select *OK*.

3.3.5 Saving the Feature Objects to a Map File

Now that you have created feature objects you can save them to a map file if you want to use them later.

1. Select *Save As* from the *File* menu.
2. Turn on the *Map* toggle box.
3. Turn off any other toggle box that may be on.
4. Specify the directory where you want to save your files.
5. Enter "*aspen3.map*" for the filename.
6. Select *Save* to exit the *Save Files* dialog and save the map file.
7. Select the *New* command from the *File* menu.

8. Select *OK* to confirm that you want all data deleted.

3.4 Conclusion

This concludes the *Advanced Feature Object Manipulation* tutorial. In this tutorial, you have learned about some of the tools in WMS for cleaning and editing feature objects. You have also learned how to assign some attributes to feature objects. Finally you have learned how to import and use data in ArcView® Shapefile and DXF format as feature objects in WMS.

Map

Supplementary Exercise for Map Module Basics-Session 2

For this exercise, a set of dems is provided. You will also need to use Paint Shop Pro.

All files for this exercise are located in the *Supplimentary 2* directory.

1. Digitizing from Contours

Since the DEM is ultimately used to map elevations it is best to create the streams so that they correspond to the DEM. One of the best ways to do this is by on-screen digitizing using the contours as a guide.

- a) Import the following USGS DEM files:
 wallsrdg.dem
 bridal.dem
- b) Turn on color-filled contouring.
- c) Read in the following map file:
 sfpoint.map
- d) Create the major streams from this point up. Don't worry about a boundary polygon at this point, you will do that in the next exercise.
- e) Save your map as sf.map

2. Digitizing from a hillshade image

An appropriately shaded DEM (hillshade) can be useful in identifying key features, particularly the rough boundary.

- a) Import the following USGS DEM files:
 wallsrdg.dem
 bridal.dem
- b) Set the display step to 1 and shade the DEM.
- c) Move the cursor to the four corner points of the DEM and record their values.
- d) Using Paint Shop Pro, capture the shaded DEM. Save the captured image as a TIFF file in the extra directory.
- e) Import and geo-reference the TIFF image you just created
- f) Use the TIFF image to create a "rough" watershed boundary.
- g) Save your map file as sf.map.

3. Digitizing from DLG data

An appropriately shaded DEM (hillshade) can be useful in identifying key features, particularly the rough boundary.

- a) Import the following USGS DEM files:
 slc_e.dem

s1c_w.dem

- b) Turn on color-filled contouring.
- c) Import the following DLG files:
 - s15.dlg
 - s17.dlg
 - s18.dlg
- d) Use the DLG points to create a set of streams.



Watershed Delineation

- Traditionally done from paper maps
- WMS provides 3 different ways
 - From feature objects (semi-automated)
 - From DEMs (automated)
 - From TINs (automated)

Watershed Delineation

Three different options

- TIN-based model
- DEM-based model
- Map/GIS-based model



The Basic Goal

- Create a digital representation of a watershed
 - automatically generate a topologic tree
- Compute geometric parameters
- Run Hydrologic model
 - HEC-1, TR20, TR-55, Rational, NFF, others

Watershed Delineation - Maps

- Hydrologic models can be created from feature object geometry and attributes
- Two basic methods are used in WMS
 - create from scratch
 - import existing data from a GIS/DXF file

Creating From Scratch

- Use a scanned image
- Use contours
- Hill shaded image
- Or just a schematic “representation” which is not to scale

Stream Direction Reminder!

- Arcs have direction
- WMS always assumes that a stream is defined from “downstream” to “upstream”
- Whenever creating stream arcs from scratch you should use this convention

Importing

- Arc/Info
 - through shape files
 - special tools for version 1.1 of ArcView spatial analyst
- Others
 - DXF
 - DLG

Arc/Info

- Applies to all methods of importing, but WMS development has been patterned after the ARC/INFO model
- Three layers of interest
 - Points - Outlets
 - Arcs - Streams and basin boundaries
 - Polygons - Basins

Which Data Are Available?

- Outlets, Streams, and Basins
- Streams and Basins
- Only Basins
- Only Streams

Outlets, Streams and Basins

- Preferred method
- WMS could possibly import directly without any further editing
- Problems which may occur?
 - intersection of streams, basins, outlets may not coincide
 - stream arc direction may be incorrect
 - Vertices or nodes must be located at the intersection of streams, basins, and outlets

Importing without Editing

- The following conditions must be met
 - stream arcs must have the proper direction
 - drainage type field in all three layers
 - point draintype = 5 (outlet)
 - arc draintype = 3 (stream)
 - polygon draintype = 1 (drainage basin)
 - The location of the outlet points, a vertex or node on a stream arc, and a vertex or node on the basin boundary must be within a tolerance of 1.0 (feet or meters depending on base units).

Streams and Basins

- Some editing typically required
 - Import into a general coverage (Before you import, you must make sure the coverage is a general coverage)
 - snap stream junctions to basin outlets
 - reorder streams (to change direction of arcs)
 - reconstruct polygon topology

Basins Only

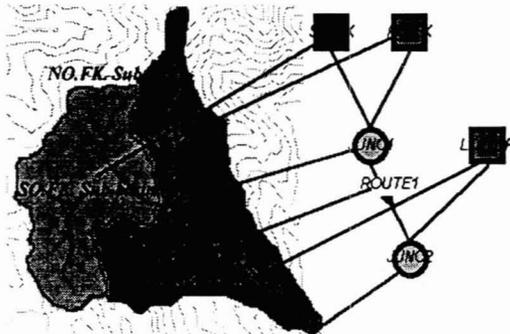
- Must add a stream from “scratch”
 - could be actual stream defined from image, contours, DLG file, etc.
 - could be a “fictitious” stream that only defines the connectivity between basins

Streams Only

- Must build basins from scratch
 - could be actual basins defined from image, contours, etc.
 - could be “fictitious” basins that only provide a representation

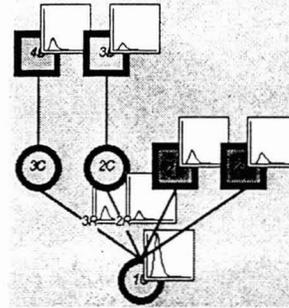
Hydrologic Model Definition

- Topologic tree structure corresponds to outlet/stream/basin feature objects
- Use the “Update basin data” command to compute areas and other data from polygons.



Run Hydrologic Model

- NFF
- HEC-1
- TR-20
- Rational Method
- Others...



Strengths of Using Feature Objects for Basin Delineation

- Hydrologic model can be constructed when no elevation data are available
 - nice graphical representation even if scale is not correct
- Existing data
 - you don't have to “re-invent the wheel”
- Overall interaction with GIS
 - often this is where data is stored/viewed

Weaknesses of Using Feature Objects for Basin Delineation

- Cannot readily compute geometric parameters
- Reservoirs not yet included as part of model

Hydrologic Models from Feature Objects

This tutorial demonstrates how feature objects alone can be used to define hydrologic models. Feature objects may be digitized on-screen using a registered image, or imported as a shape file from an Arc/Info or ArcView layer. Using the drainage coverage type, features can be converted to streams, outlets, and basins to represent the watershed being modeled. If GIS layers are imported as shape files, their corresponding attributes (i.e. area, curve number, time of concentration, etc.) are also imported and assigned to the appropriate basins.

4.1 Launching WMS

To launch the PC version of *WMS*:

Double click on the *WMS* icon .

To launch the Unix version of *WMS*:

1. Bring up an xterm window.
2. Change to the directory containing the *WMS* executable or make sure the directory containing *WMS* is in your Unix path environment.
3. type **wms** at the command line.

4.2 Creating a Watershed from Scratch with Feature Objects

By using a combination of stream arcs, outlet nodes, and basin polygons, you can develop an entire watershed even without the use of a digital terrain model. The watershed can be to scale or a schematic. Of course, if it were not to scale, polygon areas and stream lengths would not be valid for your hydrologic model.

In this section of the tutorial you will create the aspen grove watershed from an image of a scanned paper map with clearly marked streams and basin boundaries.

1. Select the *Map* module icon .
2. Select the *Open* command from the *File* menu.
3. Find and open the file named "*aspentrc.img*".

You should see a portion of a USGS quad map with basin boundaries outlined in red and the stream network in black.

4.2.1 Creating Basin Boundaries

We will begin by creating the basin boundaries, but it does not matter whether the basins or streams are created first.

1. Select the *Create Arcs* tool .
2. Select the *Attributes...* command from the *Feature Objects* menu.
3. Make sure that the arc type is *generic* and select *OK*.
4. Beginning at the outlet point (lower right) trace out the entire watershed boundary. You do not need to follow every detail, take as much time as you want. End by *double-clicking* near the same point where you began.

Note: Whenever you create feature arcs in *WMS* a snapping tolerance is used to snap the new vertex to an existing node, vertex or arc. Therefore, you do not have to click "exactly" on the same point when ending. For the most part this is very useful because it avoids the necessity of creating vertices at all arc intersection/branching points the first time it is encountered (i.e. at a stream branching point, or where sub-basin boundaries join).

1. Now create each of the other three sub-basin boundary arcs on the interior of the watershed. Begin by clicking on a point near the junction in the center of the watershed and ending by double-clicking

near the intersection of the arc previously created for the exterior boundary.

4.2.2 Creating the Stream Network

The stream network is created in much the same way the basin boundaries were. The only thing to note is that the upper basin the basin boundary comes very close to the stream. You will need to zoom in on this region in order to avoid conflicts with the snapping tolerance.

1. Select the *Attributes...* command from the *Feature Objects* menu.
2. Choose the *stream* feature arc type.
3. Select *OK*.
4. Create the main channel from the outlet of the watershed to the outlet point for the two upper basins. Begin by clicking near enough to the boundary arc at the outlet so that it snaps to it and end by double-clicking on the basin junction point.
5. Create the two branches of the lower basin by clicking on a point near the stream arc just created and double-clicking at the most upstream point of the branches in the image.

NOTE: As you create new vertices on stream arcs you should always do so from downstream to upstream.

1. Select the *Zoom* tool .
2. Zoom in on the region shown in Figure 4-1.

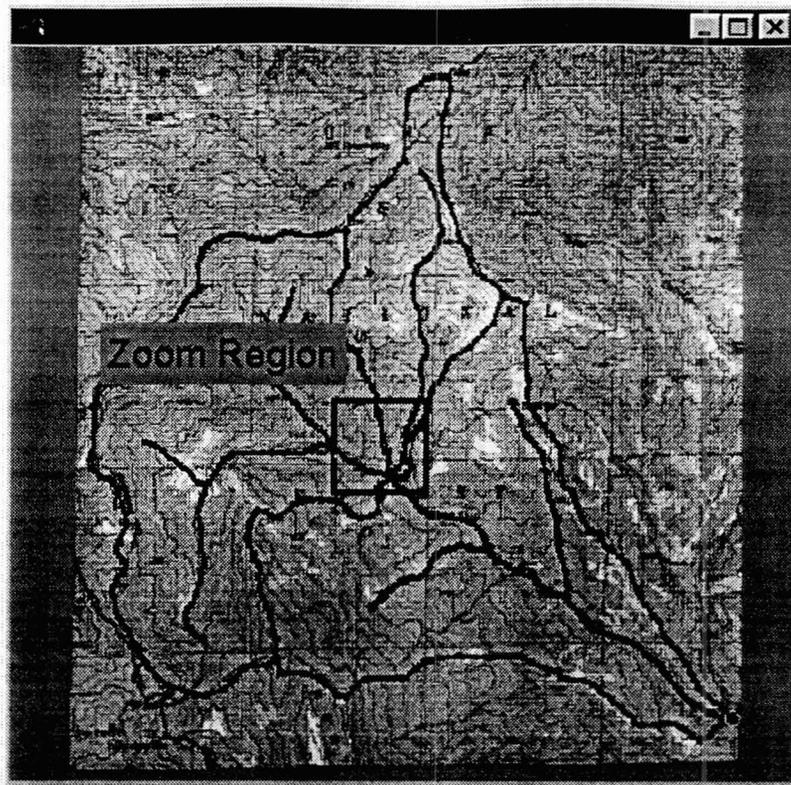


Figure 4-1 Junction of Main Channel in Aspen Grove Watershed.

1. Select the *Resample* command from the *Images* menu.
2. Select the *Create Arcs* tool .
3. Create the initial portion of each portion of the stream by clicking on the junction point (intersection of red boundary lines in the image) and going as far upstream as is possible on the zoomed image. End by double-clicking.

You needed to zoom in in order to avoid conflicts with the auto-snapping feature. If you do click too close to an existing arc you will get a message that the stream is illegal and you will need to try again.

You can end the stream at one location and then continue defining after zooming out by beginning at the point where you left off.

1. Select the *View Last* command from the *View* menu.
2. Select the *Resample* command from the *Images* menu.
3. Finish defining each branch. Begin the branch by clicking near the point you left off with and ending by double clicking at the terminal point of the stream.

In order to define separate basins at the junction point you will need to convert the node at the junction to an outlet node.

1. Select the *Select Feature Point/Node* tool .
2. Select the junction point in the center of the watershed corresponding to the intersection of the streams and the sub-basin boundary arcs that you just created.
3. Select the *Attributes* command from the *Feature Objects* menu.
4. Change the attribute to *Drainage outlet*.
5. Select *OK*.

4.2.3 Building Polygons

At this point the watershed boundaries are only arcs. In order for them to become polygons you must create the polygon topology.

1. Select the *Select Polygon* tool .
2. Select the *Attributes...* command from the *Feature Objects* menu.
3. Change the attribute to *Drainage boundary*. Select the *OK* button.
4. Select the *Select Arcs* tool .
5. Select the *Select All* command from the *Edit* menu.
6. Select the *Select Network* tool .
7. While holding down the *SHIFT* key, select any one of the stream arcs. This should un-select the stream arcs and leave the remainder selected for use in polygon generation.
8. Select the *Build Polygon* command from the *Feature Objects* menu.

4.2.4 Updating Geometric Parameters

1. Select *Display Options...* from the *Display* menu.
2. Turn on the *Color fill polygons* option.
3. Select *OK*.

In order to transfer the basin area and stream lengths, and to compute them in appropriate units for hydrologic modeling you need to compute the basin data.

This will make it possible to use the polygon area in any of the hydrologic modeling interfaces.

1. Select the *Update Basin Data* command from the *Feature Objects* menu. This command computes areas, perimeters, and centroids for each of the sub-basins and assigns these values to the hydrologic modeling tree.
2. Set the model units to meters (the base units were UTM meters).
3. Set the area units to sq. miles.
4. Set the distance units to feet.
5. Select the *OK* button to compute the sub-basin data.

4.3 Importing GIS Data to Create a Watershed

Many times stream networks and/or basin boundaries already exist as layers in a GIS. In such cases the data can be imported from a shape file (ArcView's native file format), and all applicable attributes inherited by the *WMS* feature objects for hydrologic modeling. If you have an ARC/INFO coverage you will need to convert it to a shape file first using the ARCSHAPE command. You will also want to pay attention to the keywords used for automatically mapping attributes, and some of the issues associated with importing GIS data as explained in Chapter 4 of the *WMS* Reference manual.

This part of the tutorial shows how GIS data developed with the end goal of importing into *WMS* can be used, the next three sections in the tutorial discuss how any watershed data layer(s) can be imported and then modified for use within *WMS*.

In order for *WMS* to be able to import GIS layers and automatically create a watershed for use in a hydrologic model, the following conditions must exist.

- You must have a polygon layer representing basin polygons with an attribute item named *draintype*, and the *draintype* value for each basin polygon needs to be 1.
- You must have an arc layer representing streams with an attribute item named *draintype*, and for each stream arc the value of *draintype* should be 3. Also the arcs must be ordered so that the from-node is always downstream of the to-node (if this is not the case it may be easier to follow the guidelines shown in the next section).
- You must have a point layer representing outlets with an attribute item named *draintype*, and for each outlet point the *draintype* should be 5.

A special extension in ArcView, called *WMSHydro*, has been developed which automatically creates these three layers. If you are not using this extension it may be more trouble to develop these three layers in the required format than it is to import them in one of the ways described in sections 4.4-4.6.

In addition to the required attributes, the shape file may also contain other hydrologic modeling parameters which can be mapped to corresponding values in *WMS*. Table 2.1 in the reference manual lists the attributes that can be mapped and the keywords used by *WMS* to automatically define the mapping. If your shape file contains the correct parameter, but does not use the specified keyword, you can still manually specify the mapping.

4.3.1 Importing Shape Files

1. Select the *New* command from the *File* menu to make sure that all existing data is removed. Select *OK* to delete everything.
2. Select the *Map* module icon  if it isn't active already.
3. Select the *Import...* command from the *File* menu.
4. Choose the *Shape file to Feature objects* option and select *OK*.

You should now see the Import Shapefile Data dialog. It allows you to define the shape files and mappings for points, arcs, and polygons. You can also access the coverage options dialog in case you need to change coverage types, create a new coverage, etc.

1. Click on the *open points shape file* button.
2. Find and open the point shape file named "*asppnts1.shp*".
3. Click on the *open arcs shape file* button.
4. Find and open the arc shape file named "*asparcs1.shp*".
5. Click on the *open polygons shape file* button.
6. Find and open the polygon shape file named "*asppoly1.shp*".

Attribute items are automatically mapped to *WMS* parameters if they have the appropriate keywords. Mappings are shown in the text window to the right of the shape file name. If an attribute name doesn't match a keyword as listed in Table 3.1 of the reference manual you can manually map it by doing the following:

1. Select the *Attribute Mapping...* button below the polygon shape file.
2. Select *AREA* in the *Database Fields* window.

3. Select *Basin area* in the *Coverage Attributes* window (you may have to use the scroll bar to find Basin area).
4. Select the *Map* button. If you want, scroll the *Mapped fields* window so that you can see that the area mapping has been added. An attribute could be similarly unmapped by selecting it in the *Mapped fields* window and selecting the *Unmap* button.
5. Select the *Done* button.
6. Select *OK*.

Your watershed data will be read in and all parameters that were mapped will be set inside of *WMS*. If you are running a licensed version of *WMS* you could very easily run an HEC-1 model as outlined in the next section. If you are running a demonstration version skip to section 4.4.

4.3.2 Running HEC-1 with Data Defined in a GIS

Since many of the required parameters to run the SCS options within HEC-1 were included in the shape files just imported, you can very quickly get to the point where you can run HEC-1 (this assumes of course that at some point these parameters were correctly computed and stored in the GIS). In order to complete the HEC-1 model definition do the following.

1. Select the *Hydrologic Modeling* icon .
2. Select *Job Control...* from the *HEC-1* menu.
3. Change the *Computational time interval* to 5 (minutes).
4. Change the *Number of ordinates* to 50.
5. Select *OK*.

You have specified your simulation to last for 5 times 50 minutes or 4 hours 10 minutes.

1. Select the basin named *3B* in the *Graphics* window.
2. Select *Edit HEC-1 Parameters...* from the *HEC-1* menu.
3. Select the *precipitation* button.
4. Choose the *Basin average* option.
5. Select the *Define Series* button.
6. Select the *Import* button.

7. Select and open the file "*scstabl.ser*".
8. Make sure that the *SCS-StdEmergencySpillway* is highlighted.
9. Select OK to the *XY Series Editor* and *Precipitation* dialogs.
10. Select basin *1B* from the *Hydrologic Modeling* window.
11. Repeat steps 3-9 for basin *1B* - be sure that you highlight the *SCS-StdEmergencySpillway* option in the *XY Series Editor* as it will not be the default.
12. Repeat for basin *2B*.
13. After defining precipitation for all three basins select the *Done* button on the *Edit HEC-1 Parameters* dialog.

You are now ready to run HEC-1 and review the results.

1. Select *Run HEC-1* from the *HEC-1* menu.
2. If you want to change the names of the files you can do so, but keeping the files named *untitled* is okay.
3. Select a directory where you can save your HEC-1 files using the directory browser button.
4. Select *OK*.
5. After HEC-1 completes the simulation close the window in which HEC-1 ran.
6. Select *Open* from the *Hydrograph* menu.
7. Find and open the *.sol* file created by HEC-1 (*untitled.sol* unless you changed the file names).

If you wish to examine the hydrographs in more detail:

1. Select *Show Hydrograph Window* from the *Display* menu.
2. Select any of the hydrograph icons in the *Hydrologic Modeling* window.

This completes the tutorial on using "ideal" GIS data to define hydrologic models. More often than not you will have one or more of the three required layers, and your data will need to be modified a bit after you read it in. The next three sections demonstrate some of the kinds of problems you might run into and what you can do about them.

4.4 Creating a Watershed with a GIS Stream and Basin Layers

In this section you will learn how to create the same watershed model done in the last two sections when you lack a point file defining outlets and/or the data does not meet the requirements for automatic creation as outlined at the beginning of section 4.3.

If you are continuing from the previous section you will want to delete your data before going on.

1. Select the *New* command from the *File* menu. Select OK to confirm that you want to delete everything.
2. Select the *Map* module icon .

4.4.1 Importing the Basin and Stream Layer

1. Select the *Import* command from the *File* menu.
2. Choose the *Shape file -> Feature objects* option and select *OK*.
3. Click on the *open arcs shape file* button.
4. Find and open the arc shape file named "*asparcs2.shp*".
5. Click on the *open polygons shape file* button.
6. Find and open the polygon shape file named "*asppoly2.shp*".
7. Select *OK*.

The data looks almost exactly like the shape files of the previous section. With a closer look though you will notice the following differences:

- The streams are not colored blue and the basin boundaries are not colored brown because the "Draintype" field is not present for these shape files. You may wish to turn off the color-filled option for polygons using the *Display Options* command from the *Display* menu.
- The watershed outlet and stream branching points do not connect to the basin boundaries.

4.4.2 Converting Generic Arcs to Stream Arcs

The first task will be to convert the existing set of "generic" streams to stream arcs.

1. Select the *Zoom* tool .

2. Zoom in around the watershed outlet (lower right).
3. Select the *Select Points/Nodes* tool .
4. Select the last node (the outlet point) on the stream network.
5. Select the *Reorder Stream* command from the *Feature Objects* menu.

This will insure that the arcs connected to the selected point will be ordered from downstream to upstream (from-node at the downstream end and to-node at the upstream end). This **MUST** be done before the stream arcs can be assigned a stream attribute type.

1. Select the *Stream Network* tool .
2. Select an arc on the stream network (this will actually select all arcs connected to this network).
3. Select the *Attributes...* command from the *Feature Objects* menu.
4. Choose the *Stream* attribute type.
5. Select *OK*.

Now you need to “snap” the stream outlet point to a point on the boundary of the watershed polygon.

1. Select the *Select Points/Nodes* tool .
2. Select the downstream most point on the stream network.
3. Select the *Clean...* command from the *Feature Objects* menu.
4. Make sure only the *Snap selected nodes* option is on.
5. Select *OK*.
6. You will now be prompted to select a point to snap to (red text in the help window). Click on a point on the polygon near the stream outlet point.
7. Select the *Frame Image* macro .
8. Select the *Zoom* tool .
9. Zoom in around the outlet point at the junction of the streams from the two upper basins.

If you have zoomed in around the right location your display should look something like Figure 4-2. You will notice that the outlet point does not coincide with the intersection of the three basins. You will now need to snap it in the same way you did for the watershed outlet.

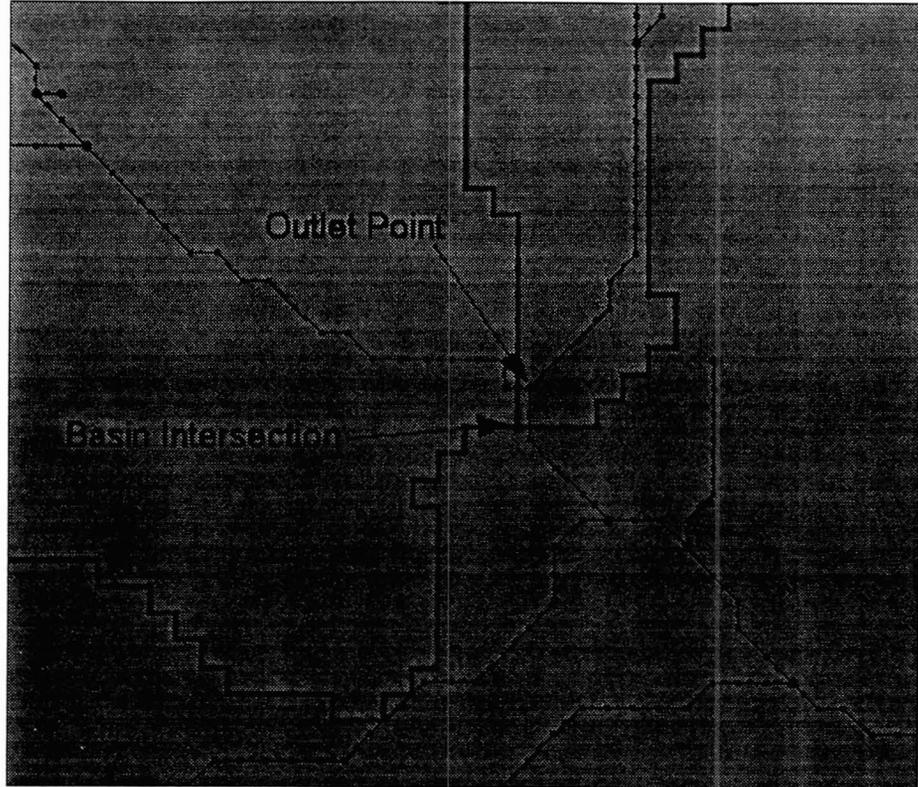


Figure 4-2 Snapping Nodes Together.

1. Select the *Select Points/Nodes* tool .
2. Select the outlet junction point on the stream network.
3. Select the *Clean...* command from the *Feature Objects* menu.
4. Make sure only the *Snap selected nodes* option is on.
5. Select *OK*.
6. Click on a point on the basin intersection point.
7. Select the newly snapped node (the intersection of the basins and stream network).
8. Select the *Attributes...* command from the *Feature Objects* menu.
9. Choose the *Drainage outlet* option.

10. Select *OK*.
11. Select the *Frame Image* macro .

4.4.3 Converting Generic Polygons to Basin Boundary Polygons

In order to “tie” the polygons to the topologic tree you must change them from *generic* to *Drainage boundary*.

1. Select the *Select Polygon* tool .
2. While holding the left mouse button down, drag a box around all three polygons (you could also multi-select the polygons by clicking insided each while holding down the *SHIFT* key or by using the *Select All* command fom the *Edit* menu).
3. Select the *Attributes...* command from the *Feature Objects* menu.
4. Choose the *Drainage boundary* option.
5. Select *OK*.
6. Select the *Update Basin Data* command from the *Feature Objects* menu.
7. Choose *Meters for Model Units* and select *OK*.

This will insure that the units for polygon area and arc length are converted/stored for hydrologic modeling.

You have now converted your stream and polygon shape file to a watershed model that is ready for use with any of the hydrologic model interfaces. Notice how if you select a single basin icon, the corresponding polygon is highlighted in red.

4.5 Creating a Watershed with a GIS Basin Layer

In this section, using the same watershed boundary data as the previous section, you will learn how to create a watershed model if you only have a GIS layer for basin boundaries. In such cases you will need to create the stream network within WMS. The stream network does not have to correspond to the actual network, but is primarily used to determined how flow is combined and routed from the different basins. If you have a scanned map, or other source of stream data you can create the actual stream, but it is not required. However, keep in mind that unless you create streams which represent the actual channel, stream length information will be incorrect.

If you are continuing from the previous section you will want to delete your data before going on.

1. Select the *New* command from the *File* menu.
2. Select the *Map* module icon .

4.5.1 Importing the Basin Layer

1. Select the *Import* command from the *File* menu.
2. Choose the *Shape file -> Feature objects* option and select *OK*.
3. Click on the *open polygons shape file* button.
4. Find and open the polygon shape file named "*asppoly2.shp*".
5. Select *OK*.

You should now have the basin file read in with three different sub-basins. There is no stream network associated with these basins and so you must define it.

4.5.2 Creating a Stream Network

1. Select the *Zoom* tool .
2. Zoom in around the intersection of the three sub-basins near the center of the watershed.
3. Select the *Create Arcs* tool .
4. Select the *Attributes...* command from the *Feature Objects* menu.
5. Choose the *Stream* attribute type.
6. Select *OK*.

Use Figure 4-3 as a guide in the next few steps.

1. Click on the intersection point of the polygons to begin a new stream segment.
2. Double-click on an interior point of the upper-left basin.
3. Repeat the last two steps for the upper-right basin.

4. For the lower basin reverse the two steps by first clicking on an interior point and then double-clicking on the intersection point.

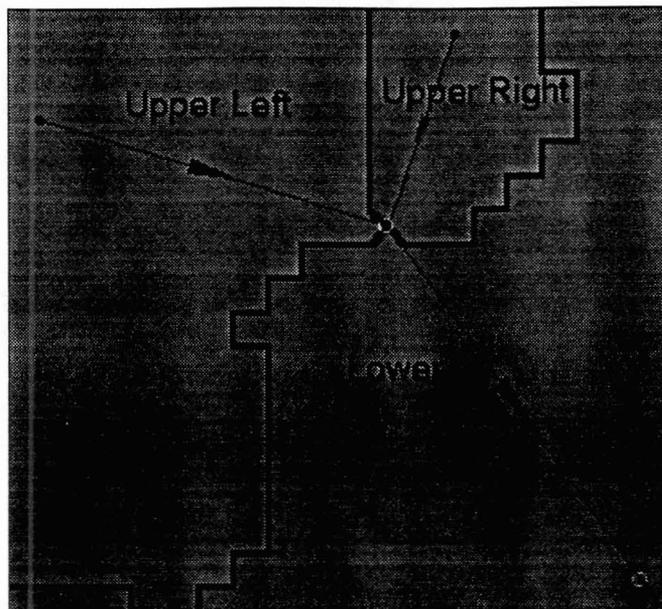


Figure 4-3 Creating Streams for Basin Polygons.

1. Select the *Frame Image* macro .
2. Click on a location on the boundary of the basin polygons near the watershed outlet (lower right portion of the screen), and then double-click on the stream point you just created in the middle of the lower sub-basin. See Figure 4-4 as a guide.

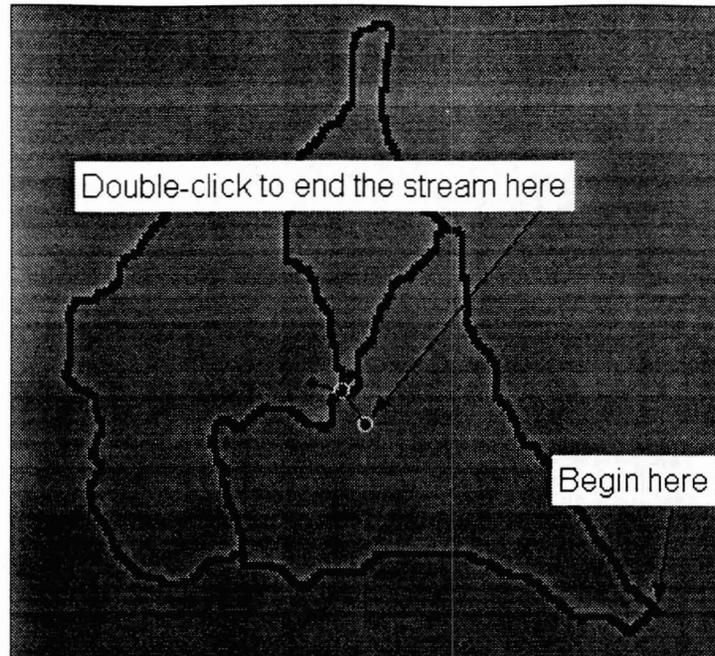


Figure 4-4 Finishing the Stream.

Now you will want to eliminate the outlet point in the interior of the lower basin polygon.

1. Select the *Select Point/Node* tool .
2. Select the node on the stream arc where you just double-clicked to join the outlet to the stream network.
3. Select the *Vertex<->Node* command in the *Feature Objects* menu.

4.5.3 Converting Polygons to Basins

In order to “tie” the polygons to the topologic tree you must change them from *generic* to *Drainage boundary*.

1. Select the *Select Polygon* tool .
2. While holding the left mouse button down, drag a box around all three polygons (you could also multi-select the polygons by clicking insided each while holding down the *SHIFT* key or by using the *Select All* command fom the *Edit* menu).
3. Select the *Attributes...* command from the *Feature Objects* menu.
4. Choose the *Drainage boundary* option.
5. Select *OK*.

6. Select the *Update Basin Data* command from the *Feature Objects* menu.
7. Choose *Meters* for *Model Units* and select *OK*.

This will insure that the units for polygon area and arc length are converted/stored for hydrologic modeling.

4.6 Creating a Watershed with a GIS Stream Layer

In this section, using the same stream data as section 4.4, you will learn how to create a watershed model if you only have a GIS layer for streams. In such cases you will need to create within WMS the basin boundaries. The boundary polygons do not have to correspond to the actual watershed/sub-basin boundaries, but they are primarily used to identify the watershed and sub-basins of a watershed model. If you have a scanned map, or other source of drainage boundary data you can create the actual basin polygons, but it is not required. However, keep in mind that unless you create polygons which represent the actual basin boundaries, polygon (basin) areas will be incorrect.

If you are continuing from the previous section you will want to delete your data before going on.

1. Select the *New* command from the *File* menu.
2. Select the *Map* module icon .

4.6.1 Importing the Stream Layer

1. Select the *Import* command from the *File* menu.
2. Choose the *Shape file to Feature objects* option and select *OK*.
3. Click on the *open arcs shape file* button.
4. Find and open the arc shape file named "*asparcs1.shp*".
5. Select *OK*.

You should now have the stream file. There are no basin polygons associated with the stream this network and so you must define them.

4.6.2 Creating Basin Polygons

Figure 4-5 is an example of basins created for the imported stream layer. Refer to it as you create your own basin polygons. In particular you will want to note the point on the stream where the three sub-basins intersect.

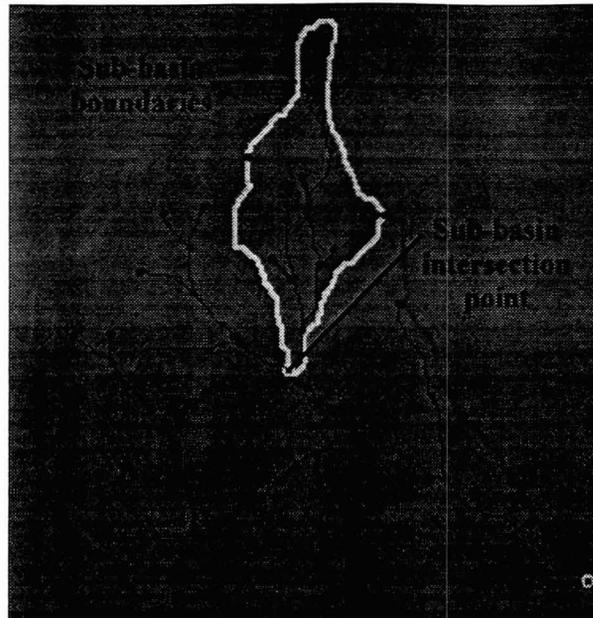


Figure 4-5 Sub-basins of Stream Network.

1. Select the *Select Point/Node* tool .
2. Double click on the node at the intersection point shown in Figure 4-5.
3. Set the attribute type to *Drainage Outlet* and select *OK*.
4. Select the *Create Arcs* tool .
5. Select the *Attributes...* command from the *Feature Objects* menu.
6. Select the *Generic* option.
7. Select *OK*.
8. Create a bounding polygon for the watershed. Begin by clicking on the outlet point, continue by creating additional points to define a polygon around the entire stream network, and end by double-clicking on the outlet point.
9. Select the *Zoom* tool .
10. Zoom in around the point on the stream where you will create the intersection of the three sub-basins (see Figure 4-5).
11. Select the *Create Arcs* tool .
12. Create the three arcs shown in Figure 4-6 that will define the boundaries of the three basins at the intersection point. For each arc

begin by clicking on the intersection point in the stream and end by double-clicking.

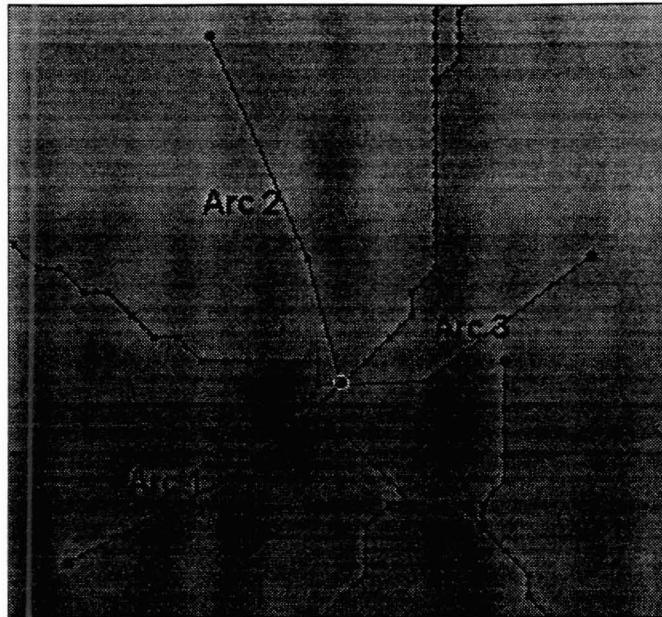


Figure 4-6 Interior Boundary Arcs.

1. Select the *Frame Image* macro .
2. Finish defining each of the three boundary arcs started at the intersection point by clicking on the end point of the arc to begin and double-clicking on the watershed boundary arc to end.

At this point you have all the necessary arcs to define the basin polygons, but you need to convert them to polygons.

1. Select the *Select Network* tool .
2. Select any of the arcs that will be used for the basin boundaries (this should cause all of the basin boundary arcs to become selected).
3. Select the *Build Polygons* command from the *Feature Objects* menu.
4. Select the *Select Polygons* tool .
5. Select all the polygons by dragging a box around them, or selecting each while holding down the *SHIFT* key.
6. Select the *Attributes...* command from the *Feature Objects* menu.
7. Make sure that *Drainage boundary* is chosen as the polygon type (if you are continuing from another part of the tutorial the polygons are

likely already set to *Drainage boundary*, but in many cases they will by default be *Generic* and you will need to change them).

8. Select *OK*.

You now have defined basins for each part of the watershed. The boundaries are not actual boundaries, but rather an approximation. Still, the hydrologic modeling tree can now be used to finish defining input for the watershed model.

Practically speaking, the only time you will want to define a watershed model in this fashion is if you already know areas for the sub-basin boundaries.

4.7 Conclusions

In this tutorial you have learned how to create watershed models using the map module objects. In particular, you have learned how to import shape files representing streams and boundaries and use them for setting up a watershed model.

Importing GIS Watershed Data

Supplementary Exercise for Basin Delineation using the Map Module- Session 5

In this exercise, you will import watershed boundary and stream data from a GIS and convert the data to a format usable by the WMS drainage coverage.

You have watershed boundaries and streams defined in Arc/Info® and you want to import your streams and boundaries into WMS to define your watershed model and generate a hydrograph. Your task in this exercise is to import your watershed boundaries and streams into WMS and edit your data so a drainage coverage can be generated.

For this exercise, you will need the following files in the *supplementary 5* directory:

- *streams.shp*-An ArcView® shape file containing the streams in your watershed. This file was generated in ArcView®. The appropriate database field was added to identify these arcs as stream arcs.
- *boundaries.shp*-An ArcView® shape file containing basin and sub-basin boundaries in your watershed. This file was generated in ArcView®. The appropriate database field was added to identify these polygons as boundary polygons.

Here's some helpful steps on how to convert your GIS shape file data to a WMS Drainage feature coverage:

1. Import the stream and boundary shape files into a WMS *General* feature coverage. If the coverage type is not *general*, you will have problems.
2. Clean your feature objects. For this model, you can select the *Clean* command from the *Feature Objects* menu to accomplish this step. This step will join the streams with the basin boundaries at the outlet points. Note that other tools are available for cleaning individual feature objects in WMS.
3. Re-build your polygons. The clean process made any existing polygons defunct.
4. Change your coverage type to a drainage coverage.
5. Select all the polygons and change their attributes to a drainage boundary attribute.
6. Convert the feature nodes at the sub-basin outlet points to outlet points.
7. Update the drainage data.
8. You are now done and ready to run your HEC-1 model (After defining all hydrologic parameters).



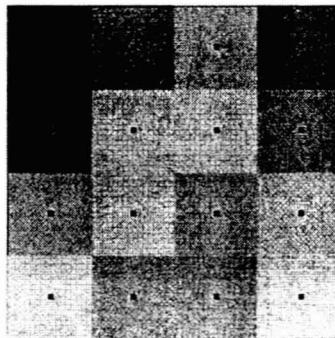
DEMs

Digital Elevation Models or DEMs are the most common digital elevation source. They can be used in WMS for automated basin delineation or as a background elevations source for creating TINs.

Structure of a DEM

- Rows and columns of elevation points
- The spacing between elevation points in x and y is referred to as the resolution.

resolution {



Supported Formats

- USGS
 - 3 arc second
 - 7.5 minute
- SDTS
- Arc/Info
- GRASS
- WMS

```
ORIGIN xlowerleft ylowerleft
DELTA X xresolution
DELTA Y yresolution
NCOLS
NROWS
ELEVATIONS
Z11
Z21
Z31
```

USGS - 3" DEM

- Covers US at 90 meter (3") resolution

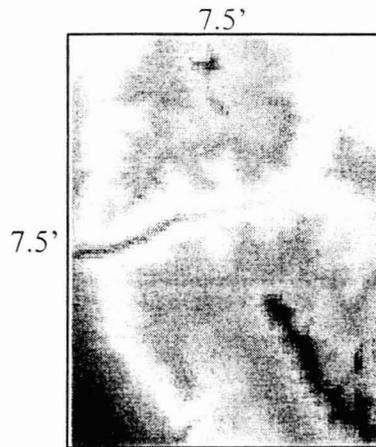


W

E

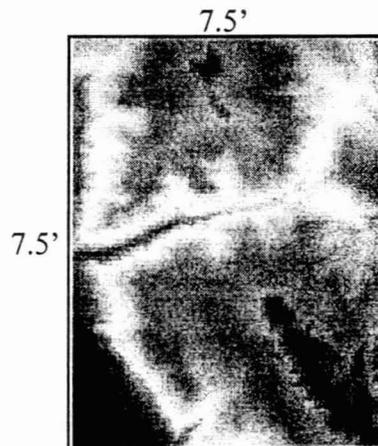
USGS - 7.5' DEMs

- Covers US at 30 meter resolution



USGS - SDTS format

- Spatial Data Transfer Standard



ARC/INFO DEM Format

- Used commonly by data publishers
- The GRIDASCII command can be used from ARC to export a DEM in this format

```
ncols 128
nrows 136
xllcorner 422415
yllcorner 4515405
cellsize 30
NODATA_value -9999
1287 1286 1286 1288 1289 1291 1291 1292 1292 1295 1300 1302 1304 1304 1304 1305 1309 1314 1323 1336 1354
1375 1395 1411 1426 1441 1453 1464 1472 1481 1491 1501 1510 1514 1521 1530 1536 1542 1549 1555 1560 1563
1565 1566 1569 1572 1574 1575 1576 1576 1577 1577 1578 1580 1581 1583 1585 1586 1588 1589 1590 1592
1596 1602 1609 1619 1627 1633 1640 1646 1648 1646 1641 1636 1632 1632 1636 1641 1647 1655 1665 1675 1684
1692 1701 1711 1726 1744 1750 1753 1751 1746 1740 1735 1729 1725 1720 1715 1709 1704 1700 1696 1694 1698
1701 1706 1708 1709 1706 1701 1694 1691 1688 1681 1680 1683 1686 1691 1699 1709 1719 1730 1740 1750 1758
1764 -9999
```

GRASS DEM Format

- GRASS is a GIS used primarily by the DOD
- Simple format, not as common as ARC/INFO

```
north: 2160
south: 0
east: 3060
west: 0
rows: 72
cols: 102
.00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 1.00000 1.00000 145.50000 142.20000
139.80000 136.30000 130.60000 132.40000 133.20000 136.20000 135.90000 131.60000 128.60000 135.70000
135.10000 135.10000 139.30000 135.10000 125.80000 117.60000 112.80000 116.40000 119.20000 119.80000
119.00000 112.80000 107.00000 105.30000 105.80000 102.20000 94.00000 95.70000 96.00000 89.20000
82.40000 77.80000 75.20000 74.80000 78.20000 84.60000 90.50000 95.10000 99.70000 107.40000 107.50000
107.80000 111.50000 113.50000 105.60000 95.30000 88.20000 79.40000 73.20000 69.60000 73.00000
71.00000 60.60000 55.60000 57.00000 54.60000 60.20000 65.40000 68.00000 63.60000 67.90000 74.60000
72.20000 68.80000 76.20000 76.70000 78.60000 85.00000 87.00000 87.20000 85.80000 80.20000 80.80000
84.00000 79.80000 74.60000 74.20000 70.10000 61.90000 54.60000 49.00000 48.50000 54.30000 51.80000
44.30000 41.10000 43.00000 49.00000 57.00000 1.00000
```

WMS DEM Format

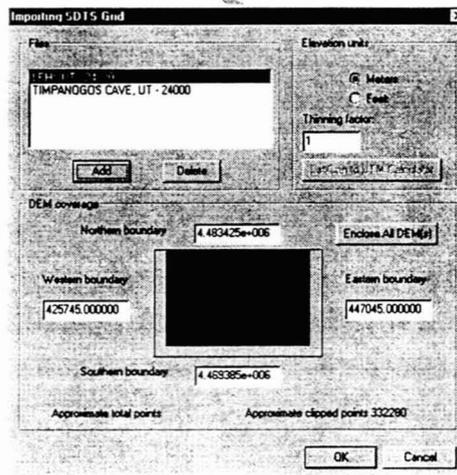
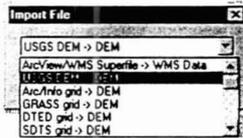
- Data from any source can be exported to the WMS format

```
DEM
ORDER 0
ORIGIN xlowerleft ylowerleft
DELTAx xresolution
DELTAy yresolution
ELEVATIONS ncols nrows
Z11
Z21
Z31
.
.
Zncol.nrow
```

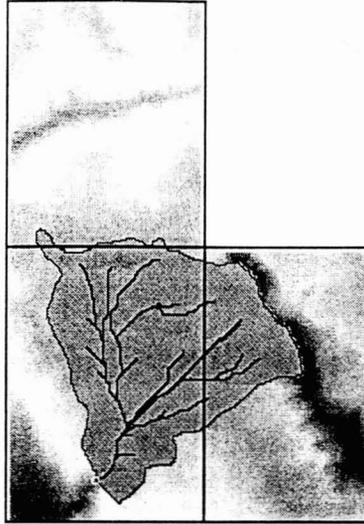
- ORDER 0 - Starts in upper left of grid, row by row
- ORDER 1 - Starts in upper left of grid, column by column
- ORDER 2 - Starts in lower left of grid, row by row
- ORDER 3 - Starts in lower left of grid, column by column

Importing DEMs into WMS

USGS, SDTS, ARC/INFO, GRASS

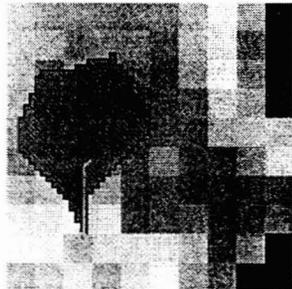


Tiling DEMS



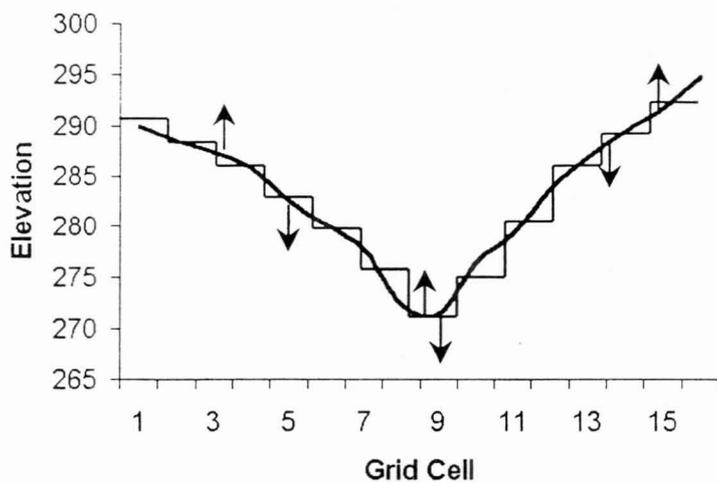
Resolution

What is the “right” resolution??

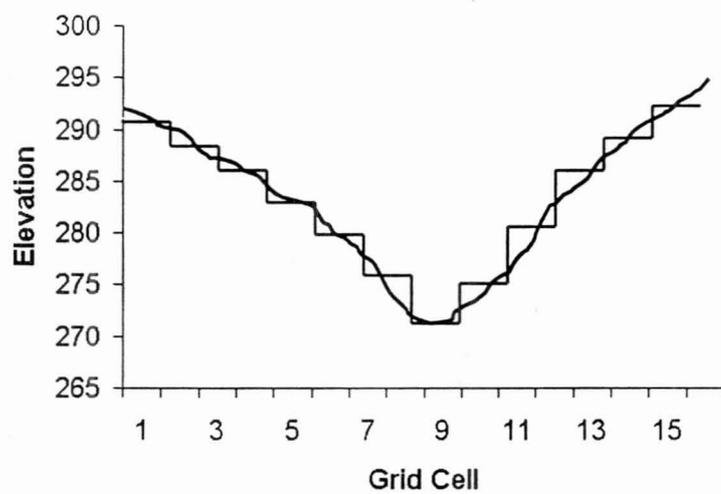


Too coarse

Vertical Error

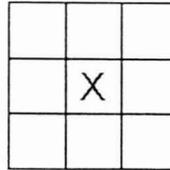


Smoothing

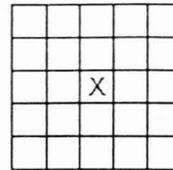


Smoothing Options

❖ Filter size



Or



❖ Number of iterations

❖ Maximum change in elevation

❖ Filter ratio

10	10	10
10	20	10
10	10	10

5	5	5
5	60	5
5	5	5

Smoothing Results



DEM Basics

Estimated Time: 15 minutes

A DEM is a file containing x, y, and elevation data (in UTM coordinates) for a portion of the earth's surface. While DEM data can be obtained from the USGS, it is often supplied by state, county or other local agencies involved in GIS work. For a listing of web sites where DEMs can be obtained see www.emrl.byu.edu/gishydrodata/dem.htm. WMS can use DEMs to directly delineate watersheds, or as background elevation maps when constructing TINs.

In this tutorial you will learn about WMS's DEM importing and visualization tools. After you have completed this tutorial you should be able to:

- Import DEMs into WMS.
- Smooth elevations to eliminate roundoff error.
- Use WMS's three dimensional viewing tools to visualize DEMs.
- Use WMS's elevation contouring options.
- Display a hill shade of the DEM.

5.1 Importing a USGS DEM

The process of importing a USGS DEM can be subdivided into three steps.

- Opening the DEM file.

- Selecting the portion of the DEM that you would like to import.
- Importing the desired portion of the DEM.

The DEM that you will be importing in this tutorial contains data for Aspen Grove, Utah. Aspen Grove is located about 50 miles south of Salt Lake City, Utah. The Aspen Grove DEM was used in previous tutorial for creating feature object data and will be used by later tutorials to construct a TIN representing the watershed located near Aspen Grove.

5.1.1 Opening the File

1. Switch to the *DEM module* by selecting the *DEM module* icon  from the top of the *Toolbox*.
2. Select the *Import* command from the *File* menu.
3. Choose the *USGS DEM->DEM* option.
4. Select *OK*.

The window at the top of the *Import USGS DEMs* dialog shows a list of the DEM files that will be imported. Since there are none in the list to begin with, you will need to add some.

1. Click on the *ADD* button.
2. Use the file browser to find and open the file "*aspen.dem*". This file should be found in the tutorial directory. This will add the DEM contained in the file to the list.
3. Select the *OK* button to import the DEM into *WMS*.

5.2 Selecting an Active Area

Whether you are using a single DEM, or several tiled together, it is likely that your watershed will not cover the entire area of the DEM. You can inactivate the portions of the DEM that you know are outside of your watershed and then delete them from memory in order to conserve RAM. There are actually several ways to do this, but the easiest and most frequently used one is demonstrated here.

1. Select the *Open* command from the *File* menu.
2. Use the file browser to find and open the file "*aspenrgn.map*." This file should be found in the tutorial directory.

After reading this file you should see a bounding box around the portion of the aspen grove DEM that contains the watershed we will be working with. You now need to eliminate the DEM outside of this bounding box.

1. Select the *select DEM points* tool .
2. Drag a rectangle that encompasses the black box defined in aspenrgn.map (see Figure 5-1).
3. Select the *Set Active Region* command from the *DEMs* menu.

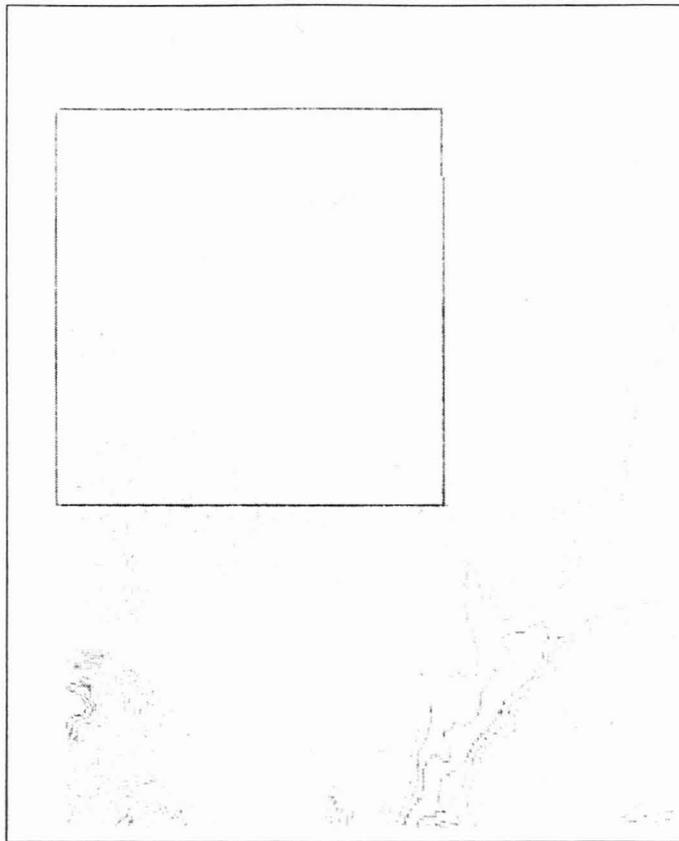


Figure 5-1 DEM Region Selection Box.

It appears that portion of the DEM outside the box is gone, but actually these elevations have just been toggled to inactive. You could bring them back by selecting the *Activate All* command, but in this instance you won't be needing these data so go ahead and delete them.

1. Select the *Display Options...* command from the *Display* menu.
2. Turn on the display of *Inactive cells* and select *OK*.

The inactive portion of the DEM will now be displayed in red. Since you won't need this part of the DEM you can go ahead and remove it altogether.

1. Select the *Delete Inactive* command from the *DEMs* menu.

5.3 Viewing the DEM

WMS features many options for viewing DEMs. These options include:

1. Elevation contours of the DEM.
2. Hill Shade display.

These tools will allow you to better visualize the topography of the region represented by the DEM.

5.3.1 Displaying Contours

WMS has the capability of displaying elevation contours on DEMs. Elevation contours are used to determine where the peaks, canyons, and valleys are in the DEM. Contours are especially useful while digitizing feature lines from the display window. WMS can display either normal linear contours or color filled contours. By default linear contours are on and you have already seen this options.

1. Select *Contour Options* in the *Display* menu.
2. To display color contours, select the *Contours...* button.
3. Turn on the *Color fill between contours* radio button.
4. Select *OK*.

Note: The escape key can be used to abort drawing at any time if it is taking to long to refresh the display.

The color legend in the top left corner of the display indicates that the blue regions are of high elevation while the red regions are of low elevation.

5.3.2 Display Speed

Depending upon the type of computer you are using, the display of the color filled contours may have been slow. To speed up the display process you can change the *Point display step*.

1. Select *Display Options...* from the *Display* menu.
2. Change the *Point display step* to 5.
3. Select *OK*.

With a larger *Point display step*, the display is faster but the resulting contours are not as smooth.

5.3.3 Displaying a Hill Shade View

By creating a hill shade view of the DEM you will get a nice image depicting the overall topographic layout. A light source is used to determine the direction of the sun, and therefore the “shadow” or shaded areas.

1. Select the *Shade* command from the *Display* menu.

With color filled contours on, the hill shade uses the contour values for shading the DEM. You can also shade the DEM in a single color (the DEM point display color) by turning off DEM contouring.

1. Select the *Display Options...* command from the *Display* menu.
2. Turn off *contours*.
3. Select *OK*.
4. Select the *Shade* command from the *Display* menu.

The lighting direction can be changed to highlight the DEM from a different angle.

1. Select the *Light Angle* command from the *Display* menu.
2. Click on a location on the sphere to indicate a new direction for the light source (the white regions on the sphere indicate the current light source direction).
3. Select the *Shade* command again from the *Display* menu.

While a true hidden surface algorithm is not used to shade the DEM, an oblique view can be used. Some angles will be more effective/accurate than others, so you may wish to experiment a little with the Rotate tool .

1. Select the *Oblique View* command from the *View* menu.
2. Select the *Shade* command from the *Display* menu.
3. Select *Plan View* from the *View* menu.

5.4 Importing ASCII Grid Files

Besides standard USGS DEMs, gridded elevation data from the ARC/INFO or GRASS GIS files can be imported and used as a DEM. WMS also has it's

own data file for gridded elevation, which can be used when trying to convert a file from other non-supported formats. Refer to the reference manual for a description of the DEM file format.

5.4.1 Importing an ARC/INFO ASCII Grid

For ARC/INFO and GRASS formatted grid files the following sequence can be used to import an ASCII file.

1. Select the *Map* module icon .
2. Select the *Delete All* command from the *Edit* menu.
3. Select the *DEM* module icon .
4. Select *Contour Options...* from the *Display* menu.
5. Turn on the *Normal linear contours* radio button in the upper right portion of the dialog.
6. Select *OK*.
7. Select the *Import* command from the *File* menu.
8. Choose the *Arc/Info grid -> DEM* option.
9. Select *OK*.
10. Select *OK* to delete the existing DEM.
11. Select the *Add* button in the *Import DEM* dialog.
12. Find and *Open* the file named "*jonescyn.grd*".
13. Select *OK*.

This DEM represents a small hill with a couple of small canyons that empty into a valley region. DEM elevations are typically rounded to the nearest meter. For areas of high relief this rounding has little noticeable affect. However, in valley areas it may cause artificial flat areas to appear. It is important to remove these flat areas before creating a TIN for drainage basin delineation. In order to see the flat areas:

1. Select the *Display Options...* command from the *Display* menu.
2. Turn everything off but contours and flat dem cells.
3. Change the *Point display step* to 1.
4. Select *OK*.

The regions colored in red are locations in the DEM where four elevation points, defining a square, all have the same elevation. While it would be rare for this to occur naturally (except for where a lake exists), this often happens with DEM data because of the roundoff errors. The natural smoothness can be restored using the following commands.

1. Select the *Smoothing Options* command from the *DEMs* menu.

These options determine how the DEM will be smoothed. By keeping the maximum change in elevation to 0.5 we won't smooth the elevations beyond what would be justified based on the original rounding to the nearest meter. Sometimes several iterations of smoothing are required to propagate changes over a large flat area. However, for this DEM you can use the default smoothing values.

1. Select the *Cancel* button.
2. Select the *Smooth DEM* command from the *DEMs* menu.

Most of the red regions should be gone. If you decide you would like to change the options for smoothing you can restore the original elevations by selecting the *Restore Elevations* command from the *DEMs* menu.

5.5 Conclusion

This concludes the DEM Basics tutorial. In this tutorial you have learned how to:

1. Import DEMs into *WMS*.
2. Use *WMS*'s DEM visualization tools.
3. Smooth DEM data.

In subsequent tutorials you will use DEMs as background elevation data for creating TINs.

DEMs

Supplementary Exercise for DEM Basics-Session 3

For this exercise, two adjacent 7.5 minute DEMs are provided for you to experiment with.

All files for this exercise are located in the *Supplimentary 3* directory.

1. Import and Clip

You want to clip out only the rectangular portion of the DEM you will need. This is a very common problem when working with USGS DEMs.

- a) Import the following USGS DEM files:

wallsrdg.dem

bridal.dem

- b) Use the following coordinates to clip out the rectangle you want:

North: 4468800

East: 464150

South: 4457300

West: 449000

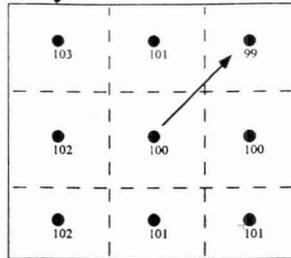
2. Experiment

- a) Hill shade the DEM, Can you identify the major features?
- b) Are there any flat regions? Can you eliminate all of them?
- c) View it in 3-D. What is the best setting for point display step.



Delineation with DEMs (Grids)

- Cell by cell flow directions

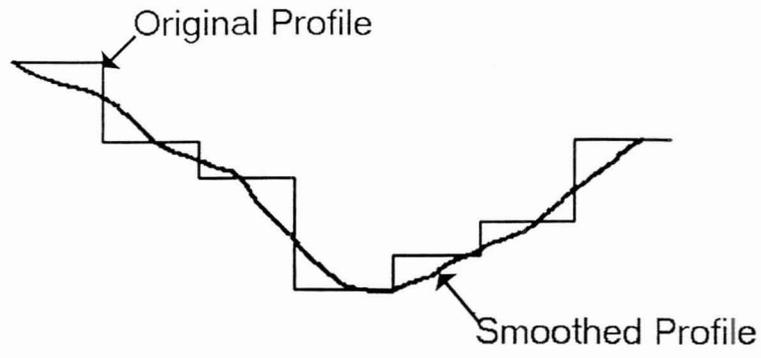


- Flow accumulations
- Streams and basin delineation

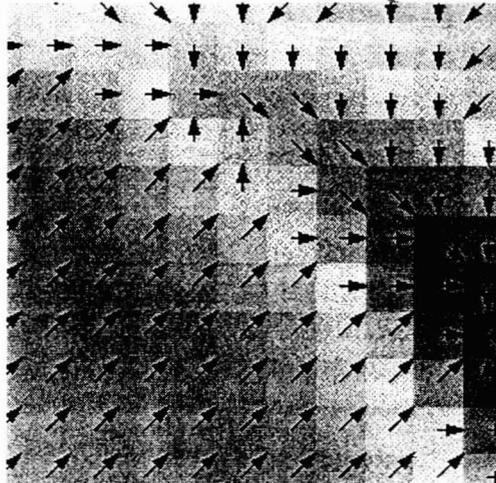
9 Steps in WMS for DEM Delineation

1. Smooth Elevations
2. Compute/Import flow directions
3. Compute/Import flow accumulation
4. Define watershed outlet
5. Convert DEM streams to Feature Objects
6. Add interior sub-basin outlets
7. Define basin(s)
8. Convert boundaries to polygons
9. Compute basin parameters

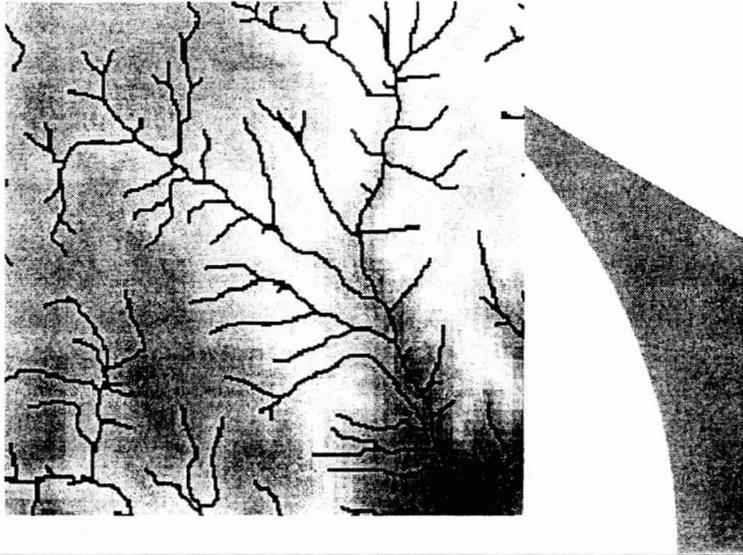
1. Smooth Elevations



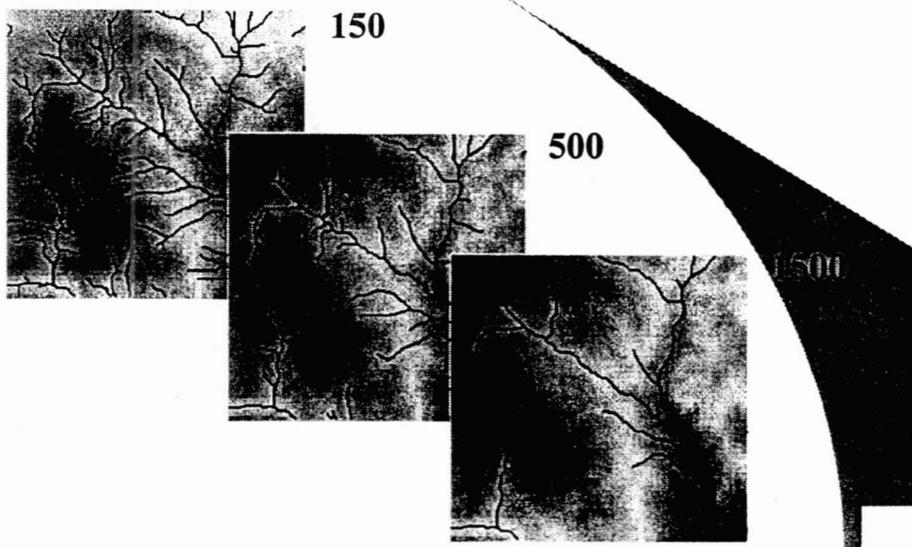
2. Compute Flow Directions



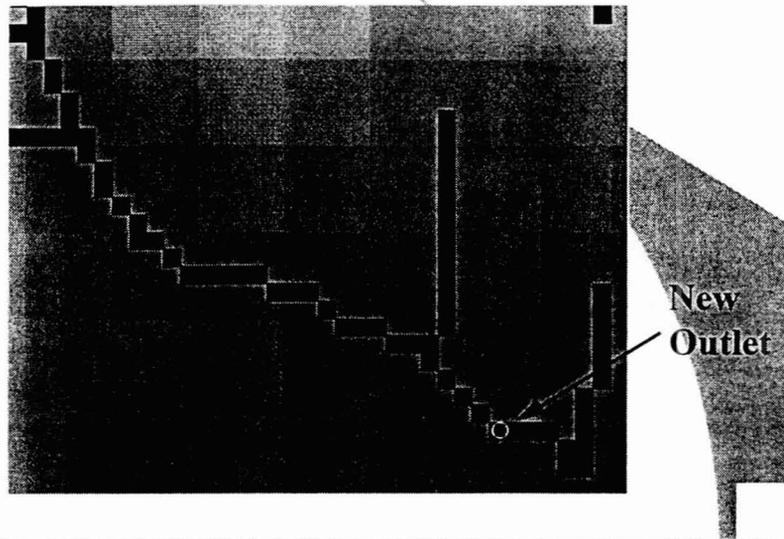
3. Compute Flow Accumulations



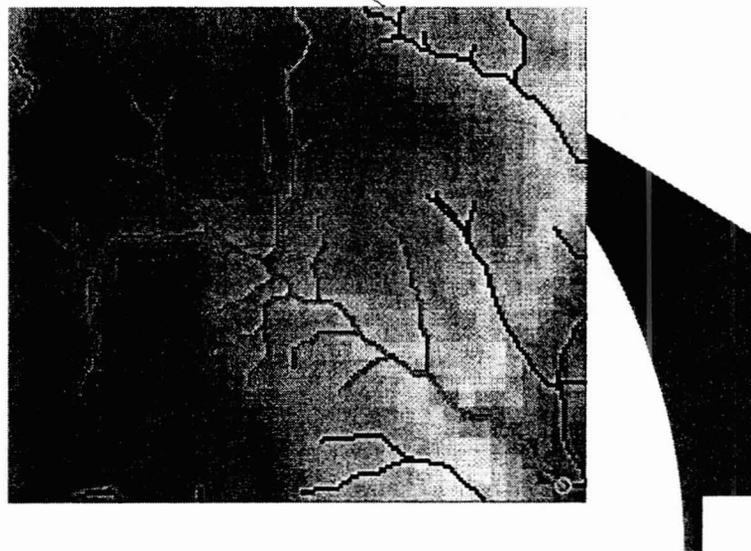
Accumulation Threshold



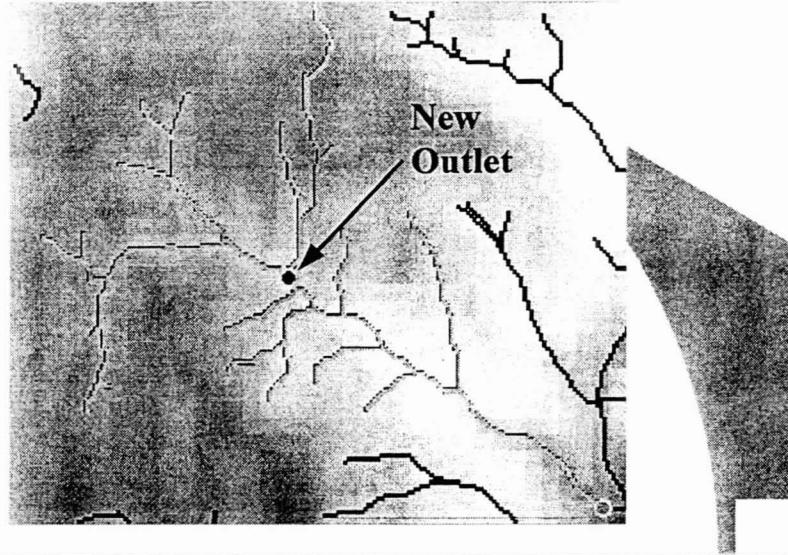
4. Define Watershed Outlet



5. Convert DEM Streams to Feature Objects



6. Define Interior Outlets



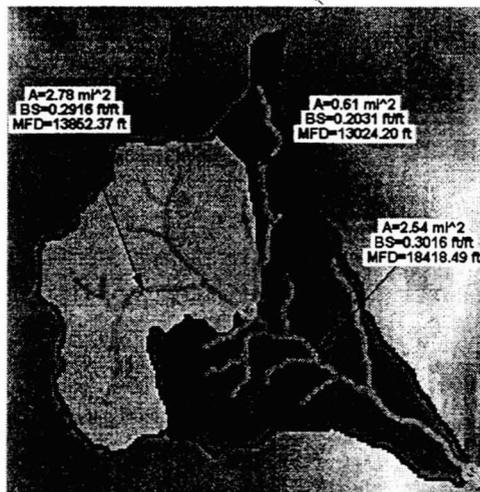
7. Define Drainage Basins



8. Convert Boundaries to Polygons



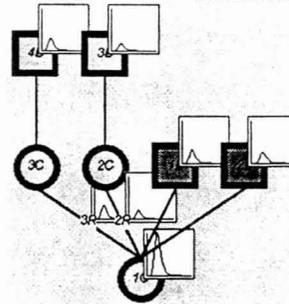
9. Compute Basin Parameters



If changes are made to outlets, basins must be redefined!!

Run Hydrologic Model

- NFF
- HEC-1
- TR-20
- Rational Method
- Others...



Strengths of DEMs

- Automated delineation with parameter calculations
- Abundance of data in this format
- Simple
- Large watersheds (small variations in flow patterns not significant)
- Generally quicker, less manual interaction required than TINs

Weaknesses of DEMs

- Small areas
 - variations in flow directions, resolution can be problematic
- Rigid data structure and can be a memory hog
- Not very dynamic
 - doesn't require much user interaction, but then again doesn't allow for it either
 - must store values with polygons

Watershed Delineation from DEMs

Estimated Time: 25 minutes

In the last tutorials you were introduced to DEMs as a source of digital elevation data. In WMS, DEMs can be used to develop watershed boundaries and important geometric parameters, or as a background elevation source for defining elevations at TIN vertices (see the tutorial in chapter 8).

In this tutorial you will learn how watersheds can be delineated from DEM data and then converted to feature arcs and polygons for use in defining a hydrologic model.

Specifically this tutorial covers the following topics:

- Importing a DEM.
- Computing flow directions with the TOPAZ program.
- Computing flow accumulations and conversion of DEM stream points to a stream feature arc.
- Addition of outlet points at any location along the stream network.
- Basin delineation and computing basin geometric parameters.
- Conversion of DEM basins to feature polygons.

6.1 Preparing the DEM

There are five DEM elevation files supported by WMS; USGS, USGS -SDTS, DTED, ARC/INFO ASCII grids, and GRASS ASCII grids. In this tutorial we will import a USGS formatted DEM file, but recognize that the same operations could be done using any of the supported formats.

6.1.1 Importing a USGS DEM

1. Switch to the *DEM module* by selecting the *DEM module* icon  from the top of the *Toolbox*.
2. Select *Import* command from the *File* menu.
3. Choose the *USGS DEM->DEM* options.
4. Select *OK*.

The window at the top of the *Import USGS DEMs* dialog shows a list of the DEM files that will be imported. Since there are none in the list to begin with, you will need to add some.

1. Click on the *ADD* button.
2. Use the file browser to find and open the file "*aspen.dem*". This file should be found in the tutorial directory. This will add the DEM contained in the file to the list.
3. Select the *OK* button in the *Import USGS DEMs* window.

6.1.2 Clipping a Region for Analysis

In the previous tutorial you learned how to specify a bounding box for importing only a portion of a USGS DEM file. You can also limit the area of analysis using the activate and inactivate DEM points commands.

1. Select the *Open* command from the *File* menu.
2. Find and open the file "*aspenrgn.map*". Since you are in the DEM module you will not see *.map files by default. You can change the filter or simply type in the file name.

This file contains a rectangle that illustrates the region you need active for analysis. In order to inactivate the remaining regions do the following:

1. Select the *Select DEM Point* tool .

2. While holding the left mouse button down drag a rectangle around the rectangle you just read in.
3. Select the *Set Active Region* command from the *DEMs* menu.
4. Go ahead and delete the inactive DEM points using the *Delete Inactive* command from the *DEMs* menu.

6.2 Delineating Stream Networks and Watershed Boundaries

6.2.1 Computing Flow Directions

Now that you have the region you want for analysis the next step is to compute the flow directions for the DEM using the TOPAZ program. This is done in two steps: 1) Computing the flow directions which requires launching the TOPAZ program from within *WMS*, and 2) reading in the resulting flow directions file.

1. Select the *Compute TOPAZ Flow Data...* command from the *Drainage* menu.
2. Use the default options and select *OK*.

At this point *WMS* will write two files, *DEDNM.INP* (input elevation data), and *DNMCNT.INP* (a control file for executing the *DEDNM* portion of *TOPAZ*). You should see some status messages in the window that appears.

1. When *TOPAZ* finishes select *OK* to close the application window.
2. Select the *Flow Directions...* command from the *Drainage* menu.
3. Select the *TOPAZ* option.
4. Select *OK*.
5. Find and open the file "*flovec.dat*".
6. Select the *Display Options...* command from the *Display* menu.
7. Turn on the display of *Flow directions* and select *OK*.

A flow direction vector for each DEM point will be drawn. Because of the resolution of DEM points it will be difficult to see the vectors so you will want to do the following:

1. Select the *Zoom* tool .

2. Zoom in on a small portion of the DEM until the vectors become visible. You should be able to get an idea of what TOPAZ has done in creating flow directions from the elevations.
3. Select the *Display Options...* command from the *Display* menu.
4. Restore the display settings by turning the *flow directions* option off.
5. Select *OK*.
6. Select the *Frame Image* macro .

6.2.2 Drawing Flow Paths

Once a flow direction grid has been imported into WMS, flow paths for any location may be drawn interactively.

1. Select the *Flow Path* tool .
2. Click on any location of the DEM.

A flow path will be traced across the DEM from point to point according to the flow direction grid. You may wish to continue clicking on several locations of the DEM to examine the flow patterns. You can also draw all flow patterns at once by:

1. Select the *Draw Flow Patterns* command from the *Drainage* menu.

This command draws a flow path from each DEM point. It is controlled by the *display step* so if you wish to see a higher or lower density of flow paths you need to adjust the display step in the *DEM Display Options* dialog.

6.2.3 Computing Flow Accumulations

In order to delineate a stream network a flow accumulation grid must be computed. The Flow Accumulation command uses the flow directions to determine how many DEM points "drain" to a given DEM point for every DEM point in the analysis area.

1. Select the *Flow Accumulations...* command from the *Drainage* menu.
2. Choose the *Read flow accumulation file* option with *TOPAZ* being the file type.
3. Select *OK*. Find and open the file named "*uparea.dat*."

The flow accumulations will be displayed when you have either read them in or computed them in WMS. The DEM points colored in blue represent

locations likely to be in a natural channel because their accumulation is high. The default threshold for display is typically 200, but you can change this.

1. Select the *Display Options...* command from the *Display* menu.
2. Set the *Min accumulation for display* value to be 1000.
3. Select *OK*.

Notice how the streams are smaller and have fewer branches. You can continue to experiment with this display option if you wish. When finished experimenting, set the threshold back to 200.

6.2.4 Converting Raster Streams to Stream Feature Arcs

Up until now the display of streams has been based solely on a threshold value, and before basin delineation can take place you must identify where you want watershed and sub-basin outlets to be along a stream vector.

1. Select the *Zoom* tool .
2. Zoom in around the branch identified in Figure 6-1.

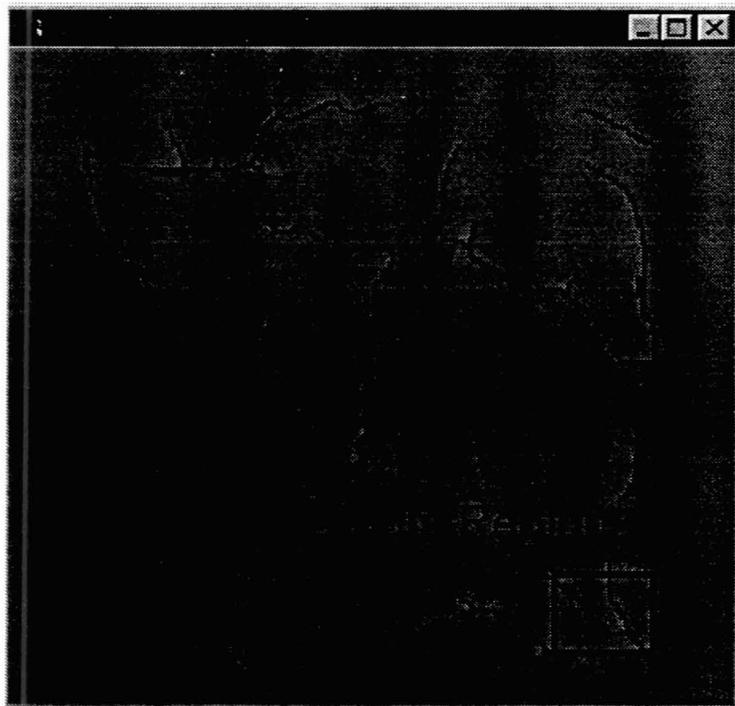


Figure 6-1 Aspen Grove Stream Network Branching Point.

1. Select the *Create Feature Point* tool . (Note that the map module tools are repeated in the DEM module because of the interaction with feature objects when delineating basins with DEMs).
2. Create a new feature point just upstream from the branching location in the stream you have just zoomed in on (see Figure 6-2).



Figure 6-2 Aspen Grove Watershed Outlet Location.

1. Select the *Frame Image* macro .
2. Select the *DEM->Streams Arcs...* command from the *Drainage* menu.
3. Turn on the *Display stream feature arc creation* and *Use feature points to create stream* options.
4. Set the *Threshold value* to 400.
5. Select *OK*.

Starting at the outlet point just created and working upstream, WMS converts the DEM points with accumulation values greater than or equal to 400 into a series of feature arcs. The feature arcs are assigned to be stream type.

6.2.5 Delineating Watershed and Sub-basin Boundaries

With the stream network defined as an arc you are ready to define the watershed boundaries.

1. Select the *Define Basins* command from the *Drainage* menu.
2. Select the *Refresh* macro .

The DEM points which “drain” to the outlet of the stream network previously defined are identified by using the flow direction information. You can now inactivate and eliminate all DEM points outside of the watershed.

1. Select the *Inactivate NULL Basin* command from the *Drainage* menu.
2. Select the *Delete Inactive* from the *DEMs* menu.

You can further subdivide the watershed into subbasins by converting some of the feature points to outlets.

1. Select the *Select Feature Point/Node* tool .
2. Select the two junction points identified in Figure 6-3. Remember you can multi-select the two points by holding down the shift key.
3. Select the *Node<->Outlet* command from the *DEM Drainage* menu to assign the nodes attribute type to be an outlet.
4. Redefine the basins with the new outlets by selecting the *Define Basins* command from the *DEM Drainage* menu.

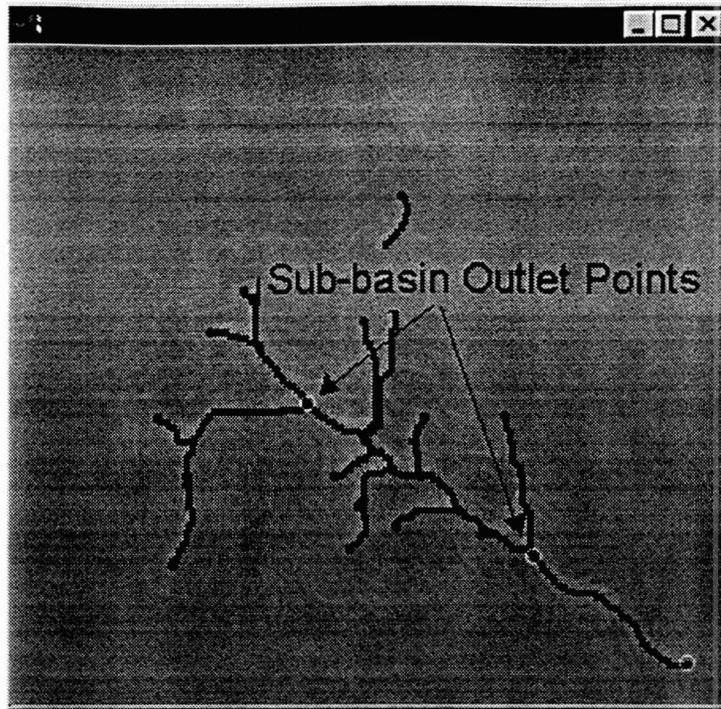


Figure 6-3 Sub-basin Outlets of Aspen Grove Watershed.

6.2.6 Converting Raster Basin Boundaries to Feature Polygons

Now that the basins have been delineated you can convert the boundaries to polygons.

1. Select the *Basins->Polygons* command in the *Drainage* menu.

6.2.7 Computing Basin Geometric Parameters

Finally you will want to compute the basin areas, slopes, runoff distances, etc. and associate them with the polygons.

1. Select the *Compute Basin Data* command from the *Drainage* menu.
2. Set the *Model Units* to meters and leave the *Parameter Units* settings at their default values.
3. Select *OK*.
4. Select the *Display Options* command from the *Display* menu.
5. Insure that the *units*, *area*, and *slopes* options are toggled on.
6. Select *OK* for both dialogs.

6.2.8 Hill Shade With Stream and Watershed Boundaries

The hill shade command demonstrated in the previous tutorial functions slightly differently once streams and watershed boundaries have been delineated. Any DEM point that is part of the stream or a basin boundary is colored accordingly when performing a hill shade.

1. Select the *Shade* command from the *Display* menu.

6.3 Conclusions

This concludes the tutorial on basin delineation from DEM data. You may wish to continue experimenting with the different options.

In summary you have learned how to:

- Import a DEM and activate a region for analysis.
- Compute flow directions and flow accumulations for a DEM.
- Convert raster streams to vector streams (feature objects).
- Define watershed and sub-basin boundaries.
- Convert raster basin boundaries to polygons.
- Compute basin data.

With the watershed delineated and basin data computed you could now continue to define a hydrologic model.

DEM Basin Delineation

Supplementary Exercise for Basin Delineation using DEMs-Session 6

In this exercise, you will combine several of the concepts you have learned about DEM Basin delineation and defining hydrologic models.

A residential development project is being planned west of Durango, Colorado. Your task is to compute the flow for a hundred-year storm at three different points in your watershed and at the watershed outlet point. The flow must be computed both before and after the residential development has taken place.

For this exercise, you will need the following files in the *supplementary 6* directory:

- *durango_west.dem*-A USGS 7.5-minute DEM Quad of the area located west of Durango, Colorado.
- *landnew.shp*-An ArcView® shape file containing land use boundaries after the proposed development takes place. A land use ID associated with each polygon is included in this file.
- *landold.shp*-An ArcView® shape file containing land use boundaries before the proposed development takes place. A land use ID associated with each polygon is included in this file.
- *soils.shp*-An ArcView® shape file containing soil boundaries for the watershed.
- *tr55I.dat, tr55IA.dat, tr55II.dat, and tr55III.dat*-Tabular unit discharge data used for computing hydrographs from TR-55 input parameters.
- *scland.tbl*-A file that relates the soil type and land use ID's in the shape files to a curve number value.

If you are experienced in running TOPAZ, you will also want to use the following files:

- *flovec.dat*-A TOPAZ-computed flow direction file that can be read into WMS.
- *uparea.dat*-A TOPAZ-computed flow accumulation file that can be read into WMS.

The following data values will be helpful for this project:

Watershed outlet point coordinates: X=239424.00, Y=4131270.75

Sub-basin outlet point coordinates:

- **Upper 2 sub-basins:** X=239310.00, Y=4134630.00
- **Upper middle 2 sub-basins:** X=240120.00, Y=4135980.00
- **Lower middle 2 sub-basins:** X=239250.00, Y=4137000.00
- **Lower sub-basin:** Same as the watershed outlet point coordinates (just use the same outlet point).

- **100-year 24-hour precipitation:** 5.0 inches
- **Rainfall distribution:** SCS Type II
- **Pond and swamp area in each basin:** 0.0% in upper 4 basins, 1.0% in lower 3 basins.
- **If the drainage area is below 2.0 mi², assume a time of concentration (T_c) of 0.7 hours.**
- **If the drainage area is above 2.0 mi², assume a time of concentration (T_c) of 1.0 hours.**
- **Travel time from upper 2 sub-basins:** 2.2 hours
- **Travel time from upper middle 2 sub-basins:** 1.5 hours
- **Travel time from lower middle 2 sub-basins:** 1.0 hours

Here's some helpful steps on how to compute the hydrographs for the desired points on the watershed:

1. Import the USGS DEM "durango_west.dem".
2. Compute and import the flow direction grid and flow accumulation grid.
3. Add an outlet point at the following coordinates: X=239424.00, Y=4131270.75

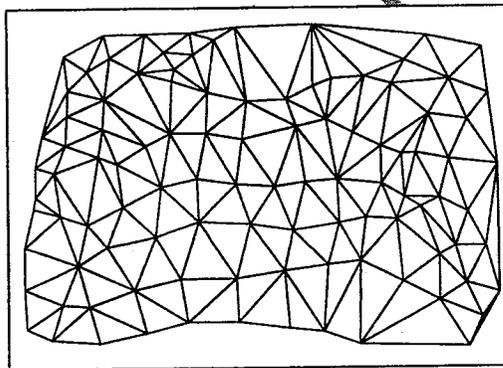
4. Create stream arcs from the outlet point just created.
5. Convert the nodes at the following locations to outlet points:
6. X=239310.00. Y=4134630.00
7. X=240120.00. Y=4135980.00
8. X=239250.00, Y=4137000.00
9. Define the drainage basins.
10. Convert the basin boundaries to boundary arcs.
11. Compute the basin data. Remember that your model units are in meters since your DEM was in UTM coordinates.
12. Go to the Tree module and select the Run Simulation command from the TR-55 menu. Initialize the TR-55 data and enter the appropriate values to each basin--
13. Basin Precipitation: 5.0 inches
14. Pond and swamp area in each basin: 0.0% in upper 4 basins, 1.0% in lower 3 basins.
15. Drainage area below 2.0 square miles...use a T_c of 0.7 hours.
16. Drainage area above 2.0 square miles...use a T_c of 1.0 hours.
17. Select the outlets and enter a travel time of 2.2 hours in the uppermost outlet, 1.5 hours in the next downstream outlet, and 1.0 hours in the next downstream outlet. The most downstream outlet has a travel time of 0.0 hours.
18. Close the TR-55 editor and do the following:
19. Import soils.shp
20. Import landold.shp
21. Compute the composite Curve Number for each basin. Before computing, you will need to import "scsland.tbl" from the *Compute Composite Runoff Coeff/CN's* window. You will also need to enter a computation step. I recommend using 100 Meters for the computation step.
22. After the curve numbers are computed, open the TR-55 window and compute the hydrographs. Follow the instructions after selecting the Compute hydrographs button in the TR-55 editor.
23. Repeat the last 4 steps after deleting the old land use coverage and importing the new land use coverage (landnew.shp).
24. After re-computing your hydrographs using the new curve numbers, observe the outflow before and after development. Record any observations.



Triangulated Irregular Networks (TINs)

- A TIN is a triangulated set of scattered xyz data points
- Each triangle represents a planar surface
- Together the triangles of a TIN form a piece-wise planar surface which honors the original data points

TIN



Why TINs?

- More accurate representation of surface features
 - Points are not restricted to lie on a grid lattice
 - High density of points where there are large variations in surface features
- Less redundant
 - Sparse data in flat regions, dense data in regions of high relief

How are TINs Created?

- Manual triangulation
 - Tedious, if not impossible for large data sets
 - Automatic triangulation
 - Delauney, others...
 - TIN Editing
 - Adding/Deleting Vertices
 - Breaklines
 - Edge swapping
- } Conditioning

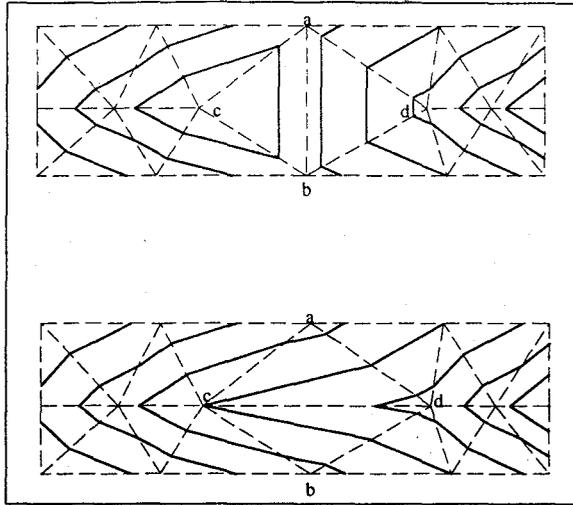
Limitation to Automated Triangulation Methods

- Triangle edges do not conform with natural breaks in terrain
 - streams, ridges, local min and max

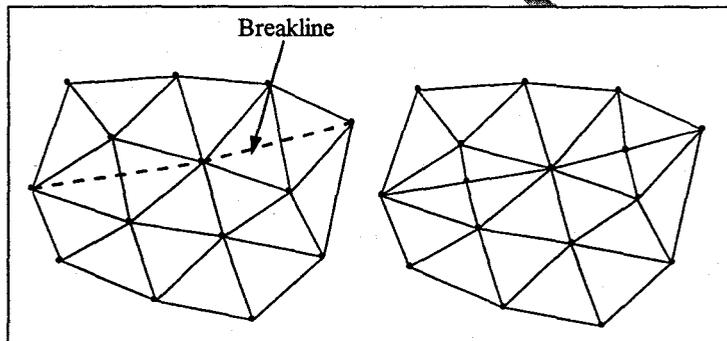
Conditioning the TIN

- Manual
 - Edge Swapping
 - Breaklines
 - Adding/Deleting points and triangles
- Automatic
 - Elevation smoothing
 - Pit/Flat Triangle/Flat Edge removal
 - automatic edge swapping

Edge Swapping

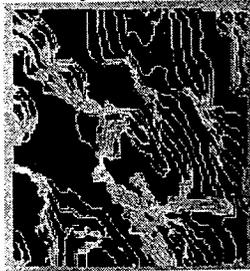


Breaklines



Smoothing Elevations

- Similar to DEM smoothing
 - iteration control
 - maximum adjustment control
- Only adjusts unlocked vertices



Eliminating TIN Anomalies

- Pits
 - Usually occur because a lack of resolution
 - Swap an edge or adjust an elevation slightly
- Flat Triangles
 - Insert a new vertex in the interior of the triangle and interpolate the elevation
- Flat Edges
 - same as TINs only along the flat edge

Display

- Contours
- Shaded
- Mapped images
- The z value could be another function value such as rainfall

TIN Basics

WMS allows you to quickly and easily import and manipulate terrain data. Besides DEM data, *WMS* can also use TINs (Triangulated Irregular Networks) for surface representation. Each TIN is constructed from a scattered set of x y z vertices. From the TINs generated by *WMS*, drainage basins can be delineated.

The vertices used to create a TIN can be imported from an ASCII file, from another TIN file (such as InRoads), or by interpolation from a DEM.

To use TINs in *WMS* effectively you will want to learn about the following:

- Importing data files.
- Using *WMS*'s viewing tools to change perspectives, allowing you to visualize a TIN in 3-D.
- Contouring the TIN.
- Using TIN display options to identify important features of the terrain model.
- Editing the TIN.

7.1 Reading Vertex Data Files Into *WMS*

After familiarizing yourself with the *WMS* window, you will be ready to load the sample vertex file that you will be using for the rest of this tutorial. This

ASCII text file is simply a list of XYZ coordinates. To load the sample vertex file, do the following:

1. Select the *TINs* module icon .
2. Under the *File* menu select *Open*.
3. Find and Open the file named "*jonescyn.tin*". By default this file is found in the tutorial directory.

A group of vertices will appear in the graphics window. If you can't see the vertices try selecting *Display Options* from the *Display* menu and turn vertices on.

7.2 Triangulating Vertices To Make a TIN

After the vertices have been read into *WMS*, they must be triangulated into a TIN.

1. Select *Triangulate* from the *TINs* menu.

During the triangulation process several long, thin triangles may be created around the perimeter of the TIN. Often these triangles are not wanted and so *WMS* provides a way of automatically selecting thin boundary triangles, so that you may delete them. To delete the triangles, do the following:

1. Select the *Select Boundary Triangles* from the *TINs* menu.
2. Choose the *Select thin boundary triangles only* option and select *OK*.

All thin boundary triangles will be shaded, indicating they are selected.

3. Select the *Delete* command from the *Edit* menu (You can also do this by selecting the *DELETE* or *BACKSPACE* key).
4. Select *OK* when the confirmation dialog box appears.

7.3 Changing the Viewing Parameters

To help you better visualize a three-dimensional terrain model, *WMS* provides multiple viewing perspectives of your TIN. The default view is plan view, where you are looking at your TIN from directly overhead. This is the easiest view in which to edit and do most of the drainage analysis with your TIN.

The *Oblique View* option in the *View* menu allows you to display the TIN in a perspective view.

1. Select *Oblique view* from the *View* menu. (You could also use the macro )

The display will be updated, and you will be viewing your TIN from a perspective view. This view emphasizes the peaks and valleys on your terrain model.

Oblique view shifts you from plan view to a viewing angle determined by the bearing and dip angles entered in the *View Angle* dialog box under the *View* menu. The default or plan view corresponds to a bearing of 0 degrees and a dip of 90 degrees.

You can use the *Rotate Image* tool  to quickly change viewing angles without having to use the *View Angle* dialog box. Use the *View Angle* dialog box when you want to set the view angle to an exact value or to check what the current viewing angle is.

To use the *Rotate Image* tool to change viewing perspectives, do the following steps:

1. Select the *Rotate Image* tool .
2. Hold down the CONTROL key on your keyboard, move your cursor to the middle of the graphics window, and click and hold down the mouse button while you drag the cursor in different directions. As you move the mouse, the TIN is constantly redrawn to reflect your changes. The bearing is changed by dragging the mouse horizontally, and the dip is changed by moving the mouse vertically. Any combination of the two directions can be used. The amount an object is rotated depends on how far the mouse is dragged. Dragging from the left edge of the edit window to the right edge rotates the object 180 degrees.

When working with complex TINs, your computer may not update the screen quickly enough. To change the viewing perspective in one step without any intermediate screen updates, do the same steps outlined above without holding down the CONTROL key.

3. When you are finished, select *Plan view* macro .

7.4 Displaying Contours on a TIN

WMS has several options for displaying contours on a TIN. WMS can display lined contours, color-filled contours, and contour elevation labels.

To display contour lines, do the following steps:

1. Select *Display Options* from the *Display* menu.

2. Click on the *Contours* option so that it is enabled.
3. Click on the *OK* button.

The screen will redraw, and the TIN will be overlaid with contour lines.

As you can see, vertices and triangles were also displayed along with the contour lines. Often when displaying contours you want to disable vertex and triangle display. To disable vertex and triangle display, do the following:

1. Select the *Display Options* command from the *Display* menu.
2. Click on both the *Vertices* and *Triangles* options so they are disabled.
3. Click on the *OK* button.

The screen will now redraw with only the contour lines and the outer boundary of the TIN.

In addition to displaying lined contours, *WMS* can display color filled contours. When active, this option fills the area between two adjacent contour lines with a color corresponding to the upper contour value. Two types of shading are available: hues of varying colors and multiple shades of a single color.

To display color filled contours shaded with hues of varying colors, do the following steps:

1. Select *Contour Options* from the *Display* menu.
2. Select the *Color fill between contours* option.
3. Click on the *OK* button.

The screen will redraw, and the TIN is displayed with hues corresponding to elevations indicated by the color legend displayed in the upper left-hand corner of the graphics window.

To display contours shaded with multiple shades of a single color, do the following:

1. Select *Contour Options* from the *Display* menu.
2. Select the *Color Ramp* tab.
3. Select the *Ramp of intensity* option.
4. Click on the *OK* button.

After the screen redraws, your TIN will be shaded with multiple shades of a single color indicating various elevations.

WMS allows Contour labels to be created and placed manually on a TIN. To display contour labels, do the following:

1. Select *Plan View* from the *View* menu.

Contour labels can be displayed only in Plan view.

2. Select the *Create Contour Label* tool .
3. Click on a point interior to your TIN.

A value indicating the elevation of the nearest contour to the point clicked will appear. Multiple labels can be displayed on a TIN. A label can be removed by holding down the SHIFT key while clicking on the label. Labels remain and are displayed each time the model is redrawn until the TIN is edited in any way at which time they are deleted.

For the next section of this tutorial you will want contours turned off. To turn off the contours and restore vertex and triangle display, do the following:

1. Select *Display Options* from the *Display* menu.
2. Turn *Contours* off.
3. Turn *Triangles* back on.
4. Change the color of triangles to a color of your choice by clicking on the color-filled box adjacent to the toggle. Click on the color box in the attributes dialog. In the color chooser select any color you want, except black. If triangles are black no shading will occur.
5. Click on the *OK* button.

7.5 Creating a Shaded Image of Your TIN

WMS provides several options for generating shaded images of TINs. TIN shading converts the wire frame TIN that you normally see into a shaded image. To shade a TIN, do the following:

1. Select *Oblique View* from the *View* menu.
2. Select *Shade* from the *Display* menu.

The screen will redraw and your TIN will appear as a solid shaded object rather than as a wire-frame image.

WMS also allows you to vary the simulated light source's position. This is useful when the default setting does not provide enough contrast in the shaded image. To vary the light source position, do the following:

1. Select *Light Angle* from the *Display* menu.

A dialog containing a shaded sphere will appear. The way the sphere is shaded indicates how the light source is directed at the TIN when shading.

2. Click on the sphere at a new location to change the light source direction.
3. Click on the *OK* button.
4. Select *Shade* from the *Display* menu.
5. If the shaded image is too dark or too light, change the light angle again.
6. When you are finished shading, select *Plan View* from the *View* menu to restore the default view.

7.6 Mapping an Image to a TIN

You can map an image to your TIN so that you can see the effects of shading with an image "draped" over the TIN. This operation is relatively simple and can be a highly effective way to show-off your data.

You will need to be sure that the image is appropriately registered. In this example you will read in an already registered image, but if you need to review this see section 2.3 of this tutorial manual.

1. Select the Map module .
2. Select the *Open* command from the *File* menu.
3. Find and open the file named "jonescyn.img."
4. Select the *Display Options...* command from the *Display* menu.
5. Select the *Image Display Options* button.
6. Turn on the *Texture map to surface when shaded* option.
7. Set the option to *Map to active TIN*.
8. Select the TINs module .

9. Select the *Shade* command from the *Display* menu. This may take 30-60 seconds depending on the speed of your computer processor.

You may want to experiment with the following adjustments to get the best view of a mapped image:

- Change the view angle using the *Rotate* tool .
- Change the Z-magnification by selecting *Z Magnification...* command from the *View* menu. This will exaggerate the 3-D effect so that terrain features appear more prominently. A value of 1-3 is generally appropriate, if the value is too large the TIN will become overly distorted.
- Change the lighting angle by selecting the *Light Angle...* command from the *Display* menu and then choose the *Shade* command from the *Display* menu.

When you are finished reset the display options.

1. Select *Display Options...* command from the *Display* menu.
2. Select the *Map* tab.
3. Select the *Image Display Options* button.
4. Turn off the *Texture map to surface when shaded* option.
5. Select the *TIN* tab.
6. Turn on *Contours*.
7. Select *OK*.
8. Select the *Plan View* macro .

7.7 Adding Breaklines to Preserve Geometric Features

Because of the way *WMS* triangulates vertices, geometric features such as ridges or valleys may not appear on your TIN as they do in the original terrain. You can use the *Add Breakline(s)* option to enter these features manually.

To add a breakline, do the following:

1. Select the *Select Vertex Strings* tool .
2. Click on any vertex in your TIN.

The vertex you clicked on will now be highlighted in red.

3. Click on another vertex that is not attached to your original vertex (i.e. connected by a triangle edge).

The two vertices you selected are now connected by a solid red line. This red line, when processed, will force triangle edges to be honored along it.

4. Select *Add Breakline(s)* from the *TINs* menu.

WMS adds vertices and retriangulates your TIN so that triangle edges are honored along the line you entered with the *Select Vertex String* tool.

Several vertices can be added to a string before selecting the Add Breaklines command. If a mistake is made entering a vertex string single vertices can be removed by pressing the DELETE/BACKSPACE key. To abort entering a vertex string using the ESCAPE key.

7.8 **Swapping Triangle Edges**

Local changes in the TIN topology can be made by swapping the edges of two adjacent triangles. This process is similar to the breakline method described above since it allows you to force triangle edges to coincide with important TIN features.

To swap triangle edges, do the following:

1. Click on the Swap Edges tool .
2. Click on the edges of several triangles and notice how the edge configuration is swapped.

Some triangle edges can not be swapped, since by so doing overlapping triangles would be formed. If you try to swap one of these edges, you will hear a beep and the edge will not be swapped.

7.9 **Selecting and Deleting Vertices on a TIN**

Some functions require one or more vertices to be selected before you perform the function. This portion of the tutorial familiarizes you with the different options for selecting vertices and how they are used in conjunction with deleting and dragging.

When selecting vertices, *WMS* allows you to work with either single or multiple vertices simultaneously.

To select a single vertex:

1. Click on the Select Vertices tool .
2. Move the mouse pointer over a vertex on your TIN and click on that vertex. The vertex will now be highlighted, indicating it is selected.
3. Click on another vertex. The vertex you selected originally is now unselected, and the new vertex is now the selected one. Notice that the xyz values of the vertex are placed in the vertex edit box.

Often you will want to manipulate multiple vertices, as when you are deleting a section of a TIN. WMS provides three methods for selecting multiple vertices:

- Select multiple vertices individually by holding down the SHIFT key and selecting several points.

If the SHIFT key is depressed while selecting allows you to select multiple points. If the SHIFT key is not depressed, only one point can be selected at a time.

- Select multiple vertices by holding down the mouse button and dragging a box enclosing the portion of the TIN you want selected. When the mouse button is released, all vertices lying within the box are selected.
- Select multiple vertices by using a selection polygon. Under the *Edit* menu, select *Select With Polygon*.

Before a vertex can be deleted it must be selected. To delete a vertex, do the following:

1. Select a single vertex on your TIN.
2. Press the DELETE/BACKSPACE key, and when a dialog box appears confirming your deletion, click on the OK button.

Remember that by selecting multiple vertices before pressing the DELETE key, you can delete multiple vertices in one step.

7.10 Dragging Vertices on a TIN

WMS allows you to easily move a vertex with the select vertices tool. The dimensions on which a vertex is dragged depends upon the editing view. In Plan view, only the x and y values are changed. In any view other than plan, the z value is the only value affected.

First, turn on several display options so you can see the effects of vertex modifications more easily:

1. Select *Display Options* from the *Display* menu.
2. Enable the *Contours*, *Channel edges*, *Pits*, and *Ridge edges* options, and leave the triangle and vertex display enabled so you will be able to see the points you are editing.
3. From within the *Display Options* dialog click on the *Contours...* button. Select the *Normal linear contours* option.
4. Click on the *OK* buttons of each dialog.

To drag a vertex, do the following:

1. Make sure you are in Plan view, and with the *Select Vertices* tool  active, click on a vertex near the center of your TIN. While holding down the mouse button, drag the point to a new location. Notice how the TIN stretches to follow your changes and how only the x and y values of that point are changed in the vertex edit dialog in the upper left side of your screen. Notice also how there are limits to the amount a vertex may be dragged so that the triangulation will remain valid.
2. With the vertex still selected (four squares are drawn around the vertex to indicate that it is selected), click in the x, y, or z vertex edit box in the upper left of the screen and change the value. Notice that you can not change the x or y value in such a way that the triangulation will become invalid in the same way that there was a limit to the distance a node could be dragged above.
3. Enter a new value into the z edit box in the edit window adjust the elevation of the selected vertex. Notice how the contours, channel edges, pits, etc. display, are all updated instantaneously as the values are modified.
4. Continue moving/editing the vertices until you are comfortable with how these options work.

7.11 Inserting Vertices on a TIN

After you have triangulated your initial set of vertices, it might be necessary to manually add vertices in certain locations to help improve the definition of the TIN.

Before adding vertices, it is wise to lock the existing vertices. Locking vertices serves two purposes. First, it prevents them from being moved (intentionally or accidentally) as described above. It also distinguishes them

from points which are subsequently added. This distinction is often important since the original vertices represent data with a greater amount of certainty (and which you are less likely to want to change).

To lock the vertices, do the following:

1. Select *Lock/Unlock Vertices* from the TIN menu. The color of the vertices will change from the unlocked color (black by default) to the locked color (red by default).

To add vertices, do the following:

1. Select *Plan view* from the *View* menu.

Plan view is the only view that you can use to enter new vertices.

2. Select the *Create Vertices* tool  from the tool palette.
3. Click anywhere within the TIN.
4. Accept the default z-value by selecting the *OK* button.

WMS will retriangulate the affected portion of the TIN, incorporating the new vertex. Notice that the new vertex is black indicating that is unlocked and can be adjusted. A default z value for the new vertex will be determined from the surrounding triangles.

5. Add a few more vertices, this time experimenting with different Z-values. Watch how the TIN changes as the new vertices are inserted into the triangulation.
6. Select *Refresh* from the *Display* menu.

7.12 Changing Display Attributes

Throughout this tutorial you have turned various display attributes (contours, vertices, triangles, etc.) off and on using the TIN display options. Besides toggling the display, colors, line widths, and line styles may be changed for contours, triangles, or any of the different objects which can be displayed.

To change display attributes, do the following:

1. Select the *Display Options* command from the *Display* menu.
2. Click on the *Contours...* button next to the contours display toggle box.
3. Click on the *Color Ramp* tab.

4. In the *Color Options* tab click on the default contour color box.
5. A color palette appears from which you may assign a new contour color.
6. Change the width style to 2.
7. Click on the *OK* button for the *Attributes* dialog.
8. Click on the *OK* buttons for the *Contour Options* and *Display options* dialogs.

Continue experimenting with turning off and on display options and changing any of their attributes. PC NOTE: MS Windows only allows dashed lines when the thickness is 1.

The ability to change line thicknesses and styles comes in particularly handy when creating hard copies where all lines are typically drawn in black.

7.13 Conclusion

TINs are one of the different types of terrain models that can be used for performing drainage analysis in WMS. The next tutorial illustrates how TINs can be created from feature object data. This is important since an arbitrary triangulation of scattered data points may not produce a TIN which is suitable for drainage analysis (i.e. triangle edges enforced along streams and ridges). If you have a set of scattered data points initially it may be a good idea to triangulate them and then treat them as a background elevation map (similar to the DEMs of the previous tutorial) for creating a new TIN from feature object data.

In this tutorial you have learned the basics of viewing and manipulating TINs. These concepts apply whether you are using the TIN for visualization only, as a background elevation map for creating a more suitable TIN for drainage analysis, or as a TIN from which a watershed will be characterized.

TIN Editing

Estimated Time: 25 minutes.

In order to define a stream network and delineate basins, you must first have flow defined everywhere on the TIN. Triangulating a set of raw data points rarely produces such a TIN. Often, flat triangles and edges, artificial pits, and discontinuities in channel segments exist. For this reason, a set of tools is provided in *WMS* which allows you to alter the initial triangulation, using your own knowledge about the site being modeled, so that drainage is properly defined and channel edges accurately represented. A certain amount of TIN editing was required for all of the TINs used in the previous tutorial. In this and the next tutorial, you will learn the different methods available for pre-processing TINs before using them for drainage analysis.

Data points used for the initial triangulation generally come from one of two different sources: digitized data from an existing contour map or gridded data such as is provided by the USGS. Different types of problems arise with each of these types of data, and therefore the editing approach is generally different. This tutorial covers editing techniques appropriate for digitized data, while techniques for gridded data is the topic of the next tutorial.

Stream network creation in the previous tutorials was done automatically. However, having a continuous set of well defined channel segments is a prerequisite to creating streams in this fashion. While having a continuous set of channel segments is ideal, it is not always practical, and therefore an alternate method of stream network creation is presented in this tutorial. This method allows you to interactively select points which make up a stream or input a digitized set of stream points from a file.

17.1 Objectives

In this tutorial you will learn basic editing techniques and automated techniques which apply specifically to data which has been generated by digitizing a contour map. The steps covered include the following:

- Reading the TIN data file.
- Triangulating the data points.
- Removing boundary triangles.
- Swapping edges to remove flat triangles and define channel segments.
- Adding breaklines so that triangle edges honor important features.
- Inserting additional points.
- Interactively moving points.
- Automatically removing flat triangles.
- Creating streams by interactively selecting a network of vertices.
- Using the display options to guide the editing process.

17.2 Reading the Data File

The TIN you will be reading for this tutorial only contains the vertices, as triangles have not yet been created (you can use this file as an example format for data you may already have and wish to read into WMS). To read the file, do the following:

1. If you are continuing from a previous WMS session, select *New* from the *File* menu.
2. Select the *TINs* module icon .
3. Select *Open* from the *File* menu.
4. Use the file browser to find and open the file "*digitize.tin*". This file should be found in the tutorial directory.
5. Select the *Display Options* command from the *Display* menu.
6. Make sure that *triangles* and *vertices* are turned on.

7. Select *OK*.

After opening the file, you should see several data points drawn in the main graphics window.

17.3 Triangulating the Data Points

Triangulation is the first process that must be done any time a new set of data points is read into *WMS*. You can watch the triangulation process proceed as will be shown, however displaying the triangulation process is more time consuming, and therefore you typically won't want to do this. To triangulate the points, do the following:

1. Select *Triangulation Options* from the *TINs* menu.
2. Turn on the *Display triangulation process* option.
3. Click on the *OK* button.
4. Select *Triangulate* from the *TINs* menu.
5. Select the *Lock/Unlock Vertices* command from the *TINs* menu.

Locking the original set of data points is a good idea since location and elevation of these points are more certain, and you do not want to move them accidentally. Any additional points created will by default be unlocked.

6. Select *Display Options* from the *Display* menu.
7. Turn on *Contours*, *Channel edges*, and *Flat triangles*.
8. Turn off *Boundaries*.
9. Click on the *OK* button.

The display should show the triangulated TIN. Notice the flat triangles displayed in red, and channel edges in yellow.

17.4 Removing Thin Triangles on the Boundary

Several long thin triangles around the perimeter of the TIN exist. Often these triangles are outside of the area of interest and, as seen along the bottom, can create several flat triangles, since all perimeter points were digitized from the same contour. These triangles can be selected for deletion in the following manner:

1. Select the *Select Boundary Triangles* command from the *TINs* menu. Several of the thin boundary triangles will automatically be selected.
2. Make sure the the *Select thin boundary triangles only* option is chosen and select *OK*.
3. Select the *Delete* command from the *Edit* menu.
4. Confirm that you want the triangles deleted by selecting *OK*.

The boundary triangles selected will be deleted and the display updated appropriately. This process could have been done manually by using the select triangles tool  and the techniques for selection described in the reference manual.

17.5 Swapping Edges

It appears as if there is some type of channel on the right side of the TIN. However, because of the flat triangles and the current TIN topology, the channels are not all present nor continuous. Edge swapping can often be used to remove flat triangles and force triangle edges where channel segments should be. The figure below shows four edges which need to be swapped in the following manner. To swap edges, do the following:

1. Select the *Swap Edge* tool .
2. Click on the edges labeled 1-4 (in that order). Notice how the contour, flat triangle, and channel edge display is immediately updated.

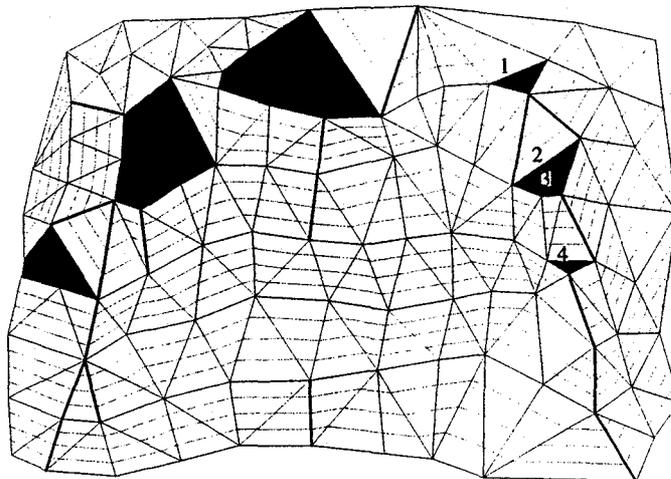


Figure 17-1 Before Swapping

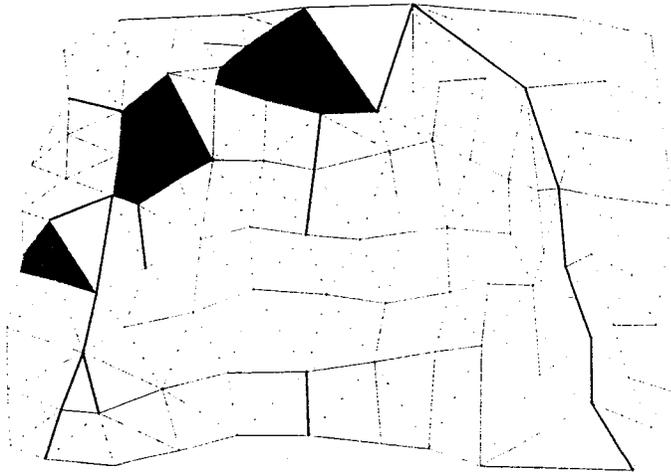


Figure 17-2 After Swapping

17.6 Adding Breaklines

Another method which can be used to force triangle edges along important terrain features such as ridges and streams is breaklines. The remaining flat triangles will be removed using a breakline. The figure above acts as a guide for placing the breakline. To add a breakline, do the following:

1. Click on the *Select Vertex String* tool .
2. Click on the vertices marked 1-4 (in that order) in Figure 17-3.
3. Select the *Add Breaklines* command from the *TINs* menu.

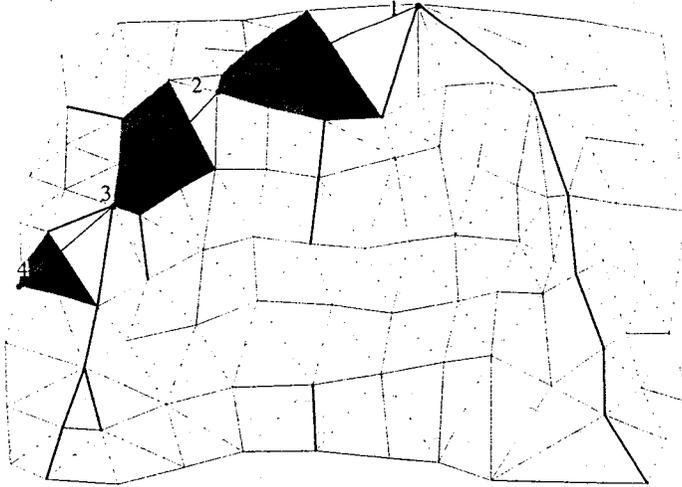


Figure 17-3. Before Adding Breakline

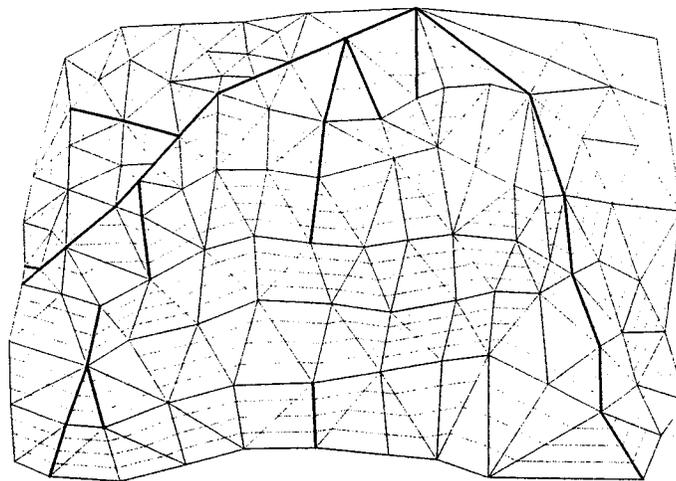


Figure 17-4 After Adding Breakline

17.7 Removing False Dams

False dams occur when a triangle edge is created perpendicular to a stream channel (from one bank to the other), as opposed to along the channel. The figure shown below shows two different cases where false dams are present.

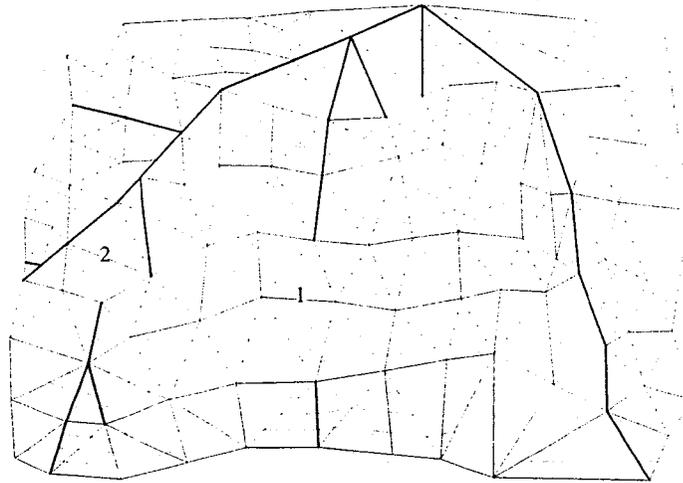


Figure 17-5 Before Removing False Dams.

It may not be obvious from the contours that a channel edge should exist at 1; however, the discontinuity in the channel segment provides a clue that the edge should be swapped. It should be more obvious that edge 2 forms a false dam. False dams can always be removed by swapping edges. To remove false dams, do the following:

1. Select the *Swap Edge* tool .
2. Click on edges 1 and 2 as shown in Figure 17-5.

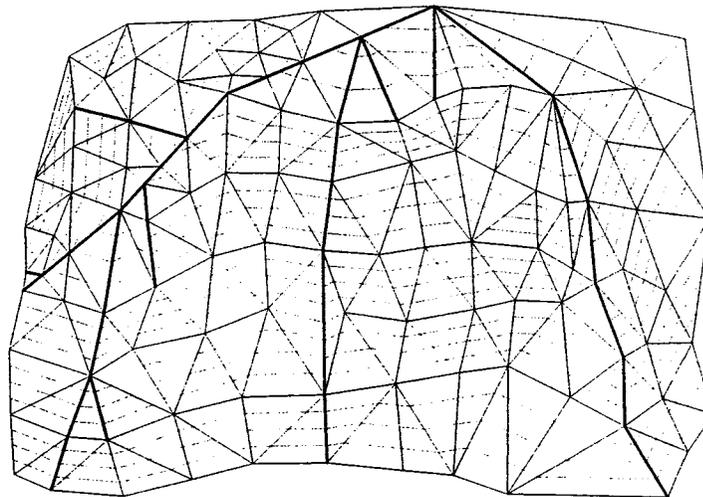


Figure 17-6 After Removing False Dams.

17.8 Inserting Additional Points

You may have noticed that there are several small channel edges connected to the main stream network. Often, no such channels exist and for clarity you wish to remove them. Typically these types of problems can be corrected by inserting additional points into the triangulation. Figure 17-7 shows how the two small channels segments near the upper middle portion of the TIN can be removed by placing a new point at A in the following manner:

1. Select the *Create Vertex* tool .
2. Click on the TIN at or about point A.
3. Accept the default z value by selecting *OK*.
4. A new point will be inserted into the triangulation, and both adjacent stream edges should disappear.

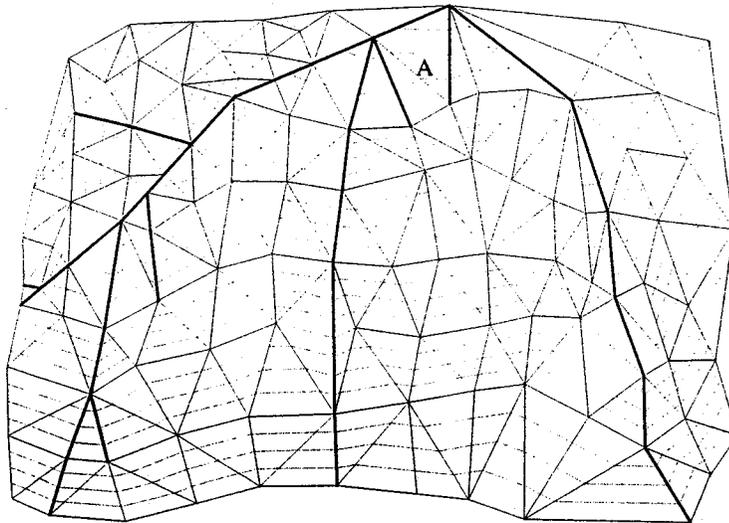


Figure 17-7 Before Adding the Point

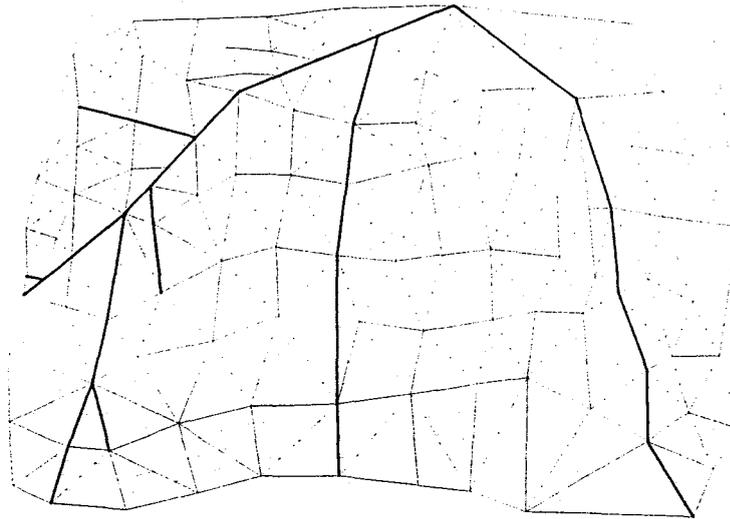


Figure 17-8 After Adding the Point.

1. Click on the *Select Vertices* tool .
2. Click on the new vertex and, while holding the mouse button down, drag it around. You should see the contours, stream edges and any other TIN display option which is turned on update immediately. This function of *WMS* is particularly useful when editing.
3. With the vertex selected, the *z* value is displayed in the text edit box in the *Edit Window*. You can edit the *z* value by changing it in this edit field.
4. Restore the *z* value of the new vertex to a logical position (make the contours look similar to the contours shown in Figure 17-8).

17.9 Automated Editing Techniques

The TIN you have been working with so far in this tutorial is only a small part of a much larger TIN created using a digitizer. The editing techniques you have learned thus far could be used repeatedly to correct flat triangles, flat edges, false dams, etc., for the entire TIN. However, removal of flat objects can often be done automatically, greatly increasing the speed and efficiency at which TINs can be prepared for doing drainage analysis. To perform these editing techniques, do the following:

1. Select the *New* command from the *File* menu.
2. Select the *Open* command from the *File* menu.

3. Find and open the file "auto.tin".
4. Select the *Triangulate Options* command from the *TINs* menu.
5. Turn off the *Display triangulation process* command.
6. Click on the *OK* button.

Displaying the triangulation for larger data sets can take significantly more time.

7. Select the *Triangulate* command from the *TINs* menu.
8. Select the *Select Boundary Triangles* command from the *TINs* menu.
9. Select the *Select thin boundary triangles only* option.
10. Select *OK*.
11. Select the *Delete* command from the *Edit* menu.
12. Click on the *OK* button to confirm the deletion of the selected triangles.
13. Select the *Lock/Unlock Vertices* command from the *TINs* menu.

In order to automatically remove flat objects, the vertices should be locked.

14. Select the *Interpolate Flat Triangles* command from the *TINs* menu.

This operation could take several seconds to a few minutes depending on your hardware. After a few seconds you should start to see the flat triangles and edges being modified. When complete you should see that channel edges run through the entire region of flat triangles.

15. Select the *Display Options* command from the *Display* menu.
16. Turn on *Pits*.
17. Click on the *OK* button.

As the display is updated, notice that there are several pits in the stream. These pits occur as a result of removing flat triangles and edges, and not necessarily because of actual pits in the stream.

17.10 Creating Streams Manually

Recall from earlier tutorials that streams could be automatically delineated from outlet points if a continuous set of channel edges existed. Even though a continuous set of channel edges exists in the current TIN, it is still interrupted by several pits. In order to automatically delineate the streams, TIN editing would have to be done to remove the pits and ensure that each successive stream vertex from upstream to downstream had a lower elevation. This process often takes a considerable amount of time, and therefore an alternative to automatically delineating stream networks is provided in *WMS*.

Stream networks can be created by manually selecting a set of vertices, **from downstream to upstream**, regardless of whether the next upstream vertex is higher or lower in elevation. This allows you to run a stream right through the set of pits in the channel above without having to do any additional editing. This type of stream creation is possible because once flow reaches a stream, it is routed downstream strictly by the connectivity of the stream. This type of stream creation has several important uses:

1. Pits in natural streams don't have to be removed using TIN editing techniques.
2. Streets, canals, storm drains, etc., which are not necessarily represented as stream edges in a TIN can be modeled.
3. A set of stream vertices can be imported and stamped into the TIN.

Along with the advantages there are a couple of pitfalls which you must be aware of:

1. Calculated stream slopes may not be accurate. However, overall averages still give reasonable results.
2. Improper placement of a stream, street, storm drain, etc., can cause significant error in computed basin areas and drainage characteristics.

In general, creating streams in this fashion is a powerful tool, but it must be used with common sense and a good understanding of the terrain being modeled. Streams can be manually entered by following the steps outlined below:

1. Click on the *Select Vertex String* tool .
2. Starting at the outlet point, select the vertices corresponding to the stream (the lower right corner of the TIN). Figure 17-9 can be used as a guide to start entering the stream. You don't have to select vertices along triangle edges since a breakline will be inserted where necessary, but be sure to pass through the pits.

3. After selecting several vertices, select the *Create Streams* command from the *Streams* menu.
4. Click on the *OK* button if asked if you would like to process a breakline before the stream is created.

You can stop entering points with the *Select Vertex String* tool  at any time and use the *Create Streams* command. Begin the next segment from where you left off, or come back to a branching point and begin entering that branch. A couple of important rules should be remembered when entering manually created streams.

1. Streams should always be entered from downstream to upstream.
2. Streams can not cross existing streams.
3. The first vertex entered for a new stream becomes an outlet, unless you are branching off from an already existing stream.
4. If stream edges created manually are swapped, or the triangulation of any of these edges are changed, the streams will no longer be valid.
5. The *Pan*  and *Zoom*  tools can be very useful when entering streams.

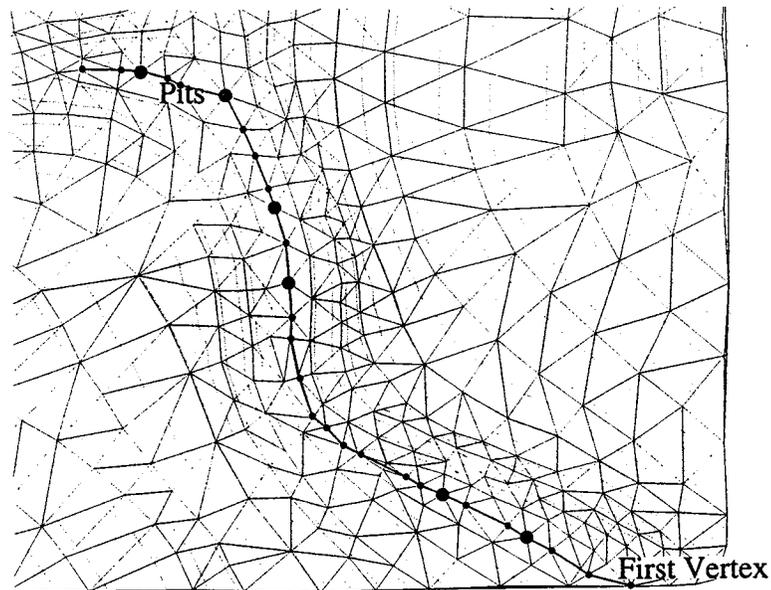


Figure 17-9 Stream Creation

17.11 Conclusions

In this tutorial you have learned many important TIN editing techniques, as well as an alternate method of creating stream networks. The important topics covered include the following:

- Triangulating a raw set of data points.
- Removing flat triangles and edges by swapping edges, inserting breaklines, and adding additional points.
- Recognizing the power of the instantaneous feedback of the display of flat triangles, channel edges, contours, etc., when editing a TIN.
- Automatically removing flat triangles and edges.
- Creating streams by manually selecting the network of vertices.

TINs

Supplementary Exercise for TIN Basics-Session 4

For the supplementary exercise with TINs do the second tutorial on TIN editing.

All files for this exercise are located in the *Supplimentary 4* directory.

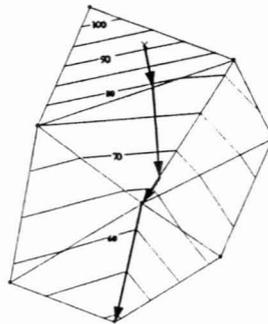


Watershed Delineation - TINs

- The piece-wise linear nature of TINs make them ideal for computing flow directions
- Basins are created by combining the triangles whose flow paths pass through a common point

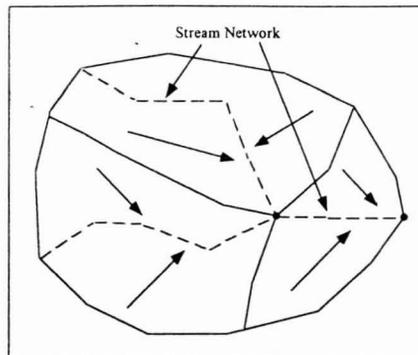
Flow on a TIN

- Piece-wise linear surface
- Flow based on path of steepest descent across triangle faces



Defining Basins

- Accumulate contributing triangles at outlets



Ten Steps Using TINs

1. Obtain Background Elevation
2. Smooth Background Elevation
3. Create Conceptual Model
4. Redistribute Vertices on Conceptual Model
5. Create TIN
6. Edit TIN
7. Complete Stream Networks and Outlets
8. Delineate Basins
9. Refine TIN
10. Compute Basin Parameters

Step 1: Obtain Background Elevation

- Scattered data triangulated into a TIN
- DEM
 - USGS 7.5 quads, or 3 arc-second
 - USGS SDTS
 - ARC/INFO
 - GRASS



Step 2: Smooth Background Elevation

- TINs or DEM



Step 3: Conceptual Model

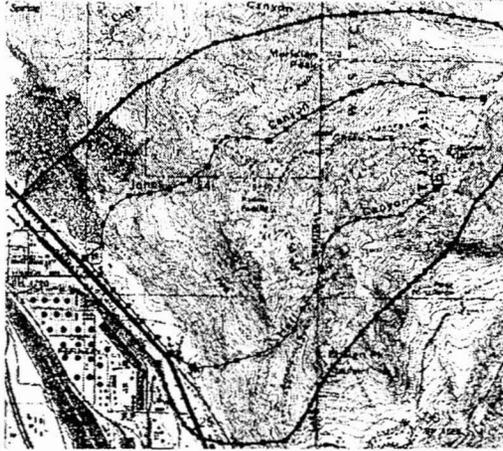
- On-screen digitizing from a geo-referenced TIFF image
- Conversion from a DXF file
- Conversion from a DLG file
- Contours from background elevation map
- Use of Hillshaded DEM
- Import GIS data

Conceptual Model



Step 4: Redistribute Vertices

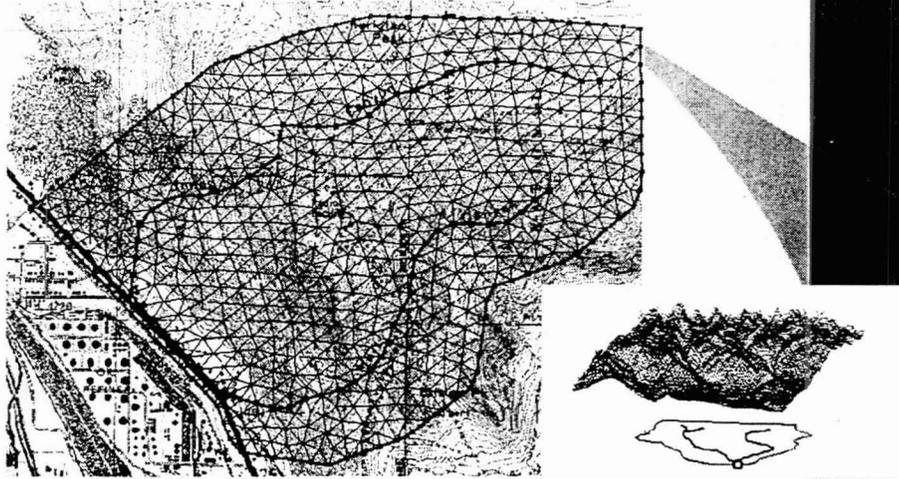
- Determines TIN vertex spacing



Step 5: Create TIN

- Bounding polygon determines limits
- All interior feature arcs are forced as breaklines
- Stream arcs are used to create a stream network within the TIN
- Elevations for TIN vertices are mapped from background elevation source

TIN From Conceptual Model

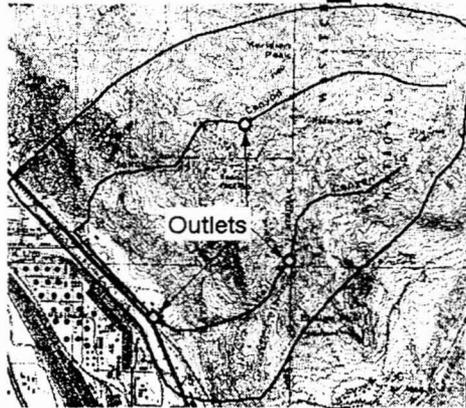


Step 6: Edit TIN

- Flat Triangles
- Flat Edges
- Pits

Step 7: Complete Stream Network and Outlet Definitions

- Outlets define locations for sub-basins



Step 8: Delineate Basins



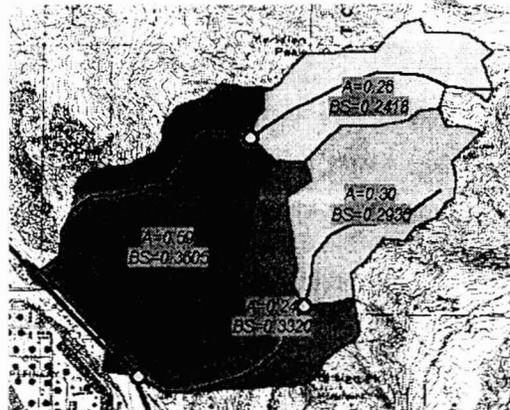
Step 9: Refine TIN

- Split flow vertices
 - results in triangles belonging to two basins
- Refine boundaries
 - splits triangles which straddle basin boundaries
- Delete triangles that do not belong to any basins
 - NULL basin

Step 10: Compute Basin and Stream Parameters

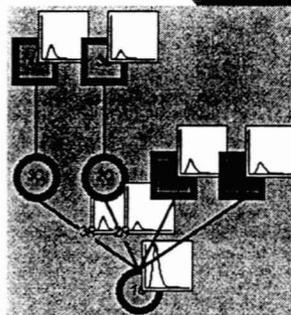
- Basins
 - Area
 - Slope
 - Mean elevation
 - Length
- Streams
 - Length
 - Slope

Basin Parameters



Run Hydrologic Model

- NFF
- HEC-1
- TR-20
- Rational Method
- Others...



Strengths of Using TINs for Basin Delineation

- Automated basin delineation with parameter calculations
- “Adaptive” resolution
 - you can use most any elevation data source
- Urban areas
 - where small variations in flow can be significant
- It was in WMS first
 - reservoir definition, storage capacity curves, flood-plain delineation

Weaknesses of Using TINs for Basin Delineation

- Lack of available data
 - With conceptual model approach this is not such a big factor anymore
- Extra steps
 - local editing

Watershed Delineation from TINs

Estimated Time: 20 minutes

The last three tutorials have introduced you to using Feature Objects and DEMs and TINs for background elevation maps. In this tutorial you will combine these tools together to create a TIN which can be used for watershed delineation parameter computation.

Specifically this tutorial covers the following topics:

- Reading in a registered TIFF image.
- Reading in a map file for use in defining key watershed features.
- Importing a DEM and using the feature objects to create a TIN.
- Delineating a watershed and sub basin boundaries from the TIN.
- Computing basin geometric parameters.
- Using drawing objects for annotations.

8.1 Reading a Registered TIFF Image

The feature object data you will be using in this tutorial was created from a TIFF map in the same way shown in section 2.2 of these tutorials. This same TIFF image, which was previously registered can be read into WMS by:

1. If you are continuing from another tutorial select the *New* command from the *File* menu.
2. Select the *Map* module icon .
3. Select *Open* from the *File* menu.
4. Type **.img* in the file name edit field (this will cause the file browser to only list file names ending with *img*).
5. Find and *Open* the file named "*jonescyn.img*".

After a few seconds of reading the file and registering the image, the TIFF image of a portion of USGS quad map will appear in the Graphics window.

8.2 Reading in a Map File

As stated in the previous section, the registered image can be used as a backdrop for on-screen digitization of streams and a rough boundary. These feature objects will be used to create a TIN that will conform to the boundary polygon and maintain triangle edges along all streams or other interior feature arcs.

If you haven't done the tutorial on creating feature objects from image, or if you need a refresher, refer to section 2.3.3 in this tutorial manual before proceeding to the next section. If you have completed the Feature Objects tutorial and already feel comfortable with how feature objects are created, then do the following:

1. Select *Open* from the *File* menu.
2. Find and *Open* the file named "*jonescyn.map*".

This map files contains a bounding polygon and two stream branches

8.3 Importing a Background Elevation Map

Elevations for vertices created inside the boundary polygon can be interpolated from a background elevation source such as a DEM or another TIN. Without a background elevation map the TIN will not have z values and therefore will be unusable for watershed delineation. The tutorial in Chapter 5 teaches you how to work with DEMs as background elevation maps. If you have not yet done this tutorial or need a refresher you may wish to do it now. To import a background elevation map.

1. Select *Import* from the *File* menu.

2. Choose the *Arc/Info grid->DEM* option.
3. Select *OK*.
4. Select the *Add* button in the Import DEM dialog.
5. Find and *Open* the file named "*jonescyn.grd*".
6. Select *OK*.
7. Select the *DEM* module .
8. Select the *Smoothing Options* command from the *DEMs* menu.
9. Set the number of iterations to 3.
10. Select *OK*.
11. Select the *Smooth DEM* command from the *DEMs* menu.

Smoothing the DEM will help to eliminate the affects of rounding to the nearest meter.

Contours of the DEM will be overlaid on the image. If the color filled contour options is active you may wish to change to linear contours by:

1. Select the *Contour Options* command from the *Display* menu.
2. Choose the *Normal linear contours* option.
3. Select *OK*.

8.4 Creating a TIN and Defining a Watershed

With the feature objects defined, and a background elevation map you are ready to create the TIN.

8.4.1 Redistributing Vertices

The density of vertices that are created inside the boundary polygon will be proportional to the density of vertices along the feature arcs. In order to have control over the TIN vertex density, WMS allows you to automatically redistribute vertices along arcs to a higher or lower density (decrease or increase spacing between arc vertices). To redistribute the vertices:

1. Select the *Map* module .

2. Select the *Select Arc* tool .
3. Choose the *Select All* command from the *Edit* menu.
4. Choose the *Redistribute* command from the *Feature Objects* menu.
5. Specify the *Subdivide each end uniformly* with a *Specified spacing* of 100 (meters).
6. Select the *OK* button.

8.4.2 Creating a TIN

To create a TIN from the defined feature objects.

1. Select the *Select Polygon* tool .
2. Click anywhere inside the boundary polygon.
3. Select the *Create TIN* command from the *Feature Objects* menu.
4. Select *No* when prompted if you would like to save your data before proceeding. The drainage coverage and TIN cannot coexist in WMS so the drainage coverage is deleted as a result creating the TIN. This prompt allows you to save the drainage coverage prior to having it deleted from WMS so that you can come back and use it later should you need to.
5. Turn on the *Display triangulation process* option (this will allow you to see, and better understand, how the TIN creation process works).
6. Select *OK* in the *Create TIN Options* dialog.
7. When prompted to delete the DEM select *YES*.
8. Select the *Display Options* command from the *Display* menu.
9. Select the *Image Display Options* button.
10. Turn off the *Draw on xy plane behind all objects* option.
11. Select *OK* to the *Image Display Options* and *Display Options* dialogs.
12. Select the *Delete* command from the *Drawing Objects* menu.
13. Confirm that you want to delete all drawing objects.

A TIN should be generated inside the bounding polygon and a stream enforced along the stream arcs. The TIN does not represent the actual watershed

boundary, but can now be used to delineate the watershed and define sub basins.

8.5 Watershed Delineation

The feature object boundary used to create the TIN was not the watershed boundary. In fact it was created to be larger than the watershed boundary so that a more precise watershed boundary can be delineated from the TIN.

1. Switch to the *TINs* module .
2. Select the *Display Options* from the *Display* menu.
3. Turn off *Vertices* and turn on *Pits* and *Flat triangles*.
4. Select the *OK* button.

8.5.1 Eliminating Pits and Defining Boundaries

You should not have any flat triangles, but there may be a few pits which must be removed before defining watershed and sub basin boundaries.

1. Select the *Smooth Pits* command from the *TINs* menu.

A single pit will likely remain along the stream, but since it is part of the stream it will not affect basin delineation.

1. Choose the *Define Basins* command from the *Drainage* menu.

8.5.2 Correcting Split Flow Problems

For some models locations of split flow occur, causing an ambiguity in sub basin definition. If this occurs for your model you will get an error message stating that "triangles drawn in red belong in two drainage basins." If this happens you can correct the problem by:

1. Select the *Correct Split Flow* command from the *Drainage* menu.

You no longer need the display of pits, flat triangles and split flow vertices.

1. Select the *Define Basins* command from the *Drainage* menu.
2. Select the *Display Options* from the *Display* menu.
3. Turn off all of the display options.
4. Select the *Drainage* tab.

5. Turn on *Fill drainage basins*.
6. Select *OK*.

8.5.3 Refining Boundaries and Eliminating Exterior Triangles

Triangles are assigned to drainage basins based on following a flow path from the centroid of the triangle to the stream/outlet encountered first. Since some triangles will straddle the basin boundaries there is a small amount of error in the current basin definition. You can correct this by refining the boundaries.

1. Choose the *Refine Boundaries* command from the *Drainage* menu.
2. Remove the exterior triangles by choosing the *Delete Null Basin Triangles* command from the *Drainage* menu.

8.5.4 Computing Basin Geometric Parameters

The brown and orange areas represent the sub basins of the two stream branches. By default WMS created an outlet point at the downstream-most part of the feature arcs. Basins were created for each upstream branch of the outlet. You can now compute the basin geometric parameters for the watershed, and alter the delineation by eliminating exterior triangles, merging basins, adding additional outlets, etc.

1. Choose the *Compute Basin Data* command from the *Drainage* menu. Be sure the model units are set to *meters* and specify the parameter units to be whatever you wish.
2. Select *OK* to compute the basin data.

8.5.5 Adding a New Stream

Additional streams and outlets can be created for the TIN, even after disposing of the feature objects and establishing an initial delineation.

1. Select *Display Options* from the *Display* menu.
2. Turn on *triangles* and *contours*.
3. Select the *Drainage* tab.
4. Turn off *Fill drainage basins*.
5. Select *OK*.
6. Select the Zoom tool .

7. Drag a rectangle around the region identified in Figure 8-1.

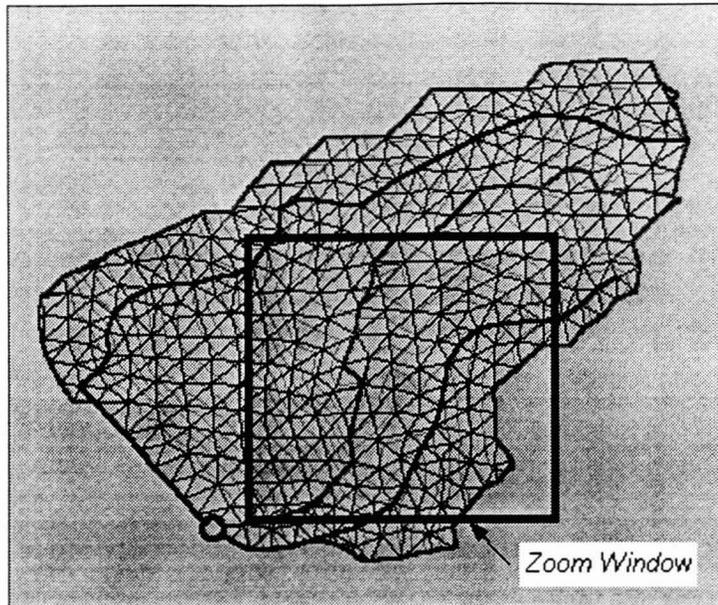


Figure 8-1 Zoom Window.

You will now add a new stream that was not present in the initial feature object definition.

1. Select the Select Vertex String tool .
2. Starting with the vertex already in the stream click one by one on the vertices identified in Figure 8-2. A red line will indicate the string of vertices selected.

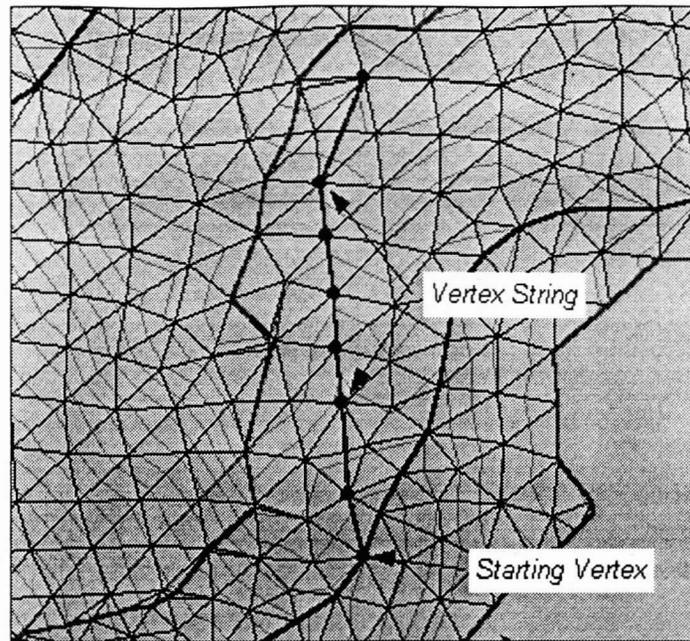


Figure 8-2 Entering a Vertex String to Create a New Stream.

1. After selecting the last vertex, select the *Create Stream* command in the *Streams* menu.
2. Select *Yes* to process the breakline before entering the stream.

8.5.6 Creating Additional Outlets

1. Select the *Select Vertex* tool .
2. Select the vertex where the newly created stream meets the existing stream.
3. Select the *Add Outlets* command in the *Streams* menu.
4. Select the *Frame Image* command from the *View* menu.
5. Select the vertex midway up the upper branch as indicated in Figure 8-3.
6. Select the *Add Outlets* command in the *Streams* menu.

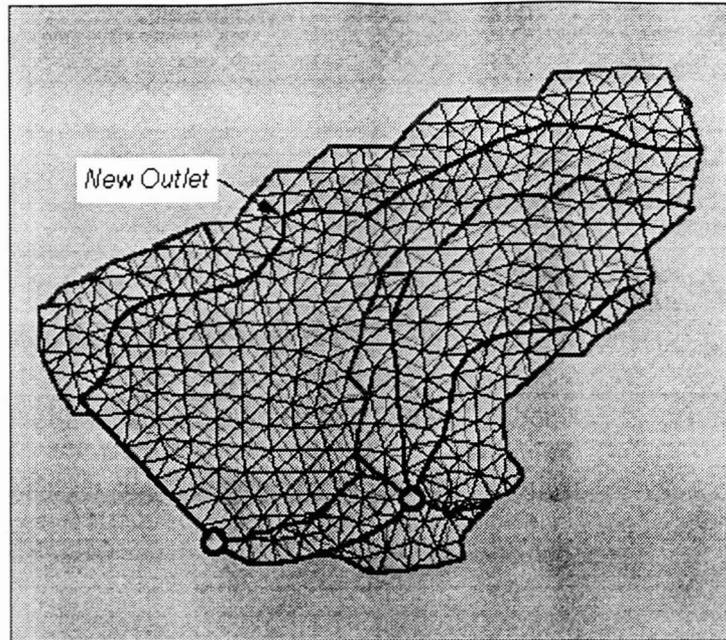


Figure 8-3 Vertex for New Stream

You must now redefine the boundaries so that the new outlet is recognized.

1. Select the *Display Options* command in the *Display* menu.
2. Turn off *triangles* and *contours*.
3. Select the *Drainage* tab.
4. Turn on *Fill drainage basins*.
5. Select *OK* for both dialogs.
6. Select the *Define Basins* command from the *Drainage* menu.
7. Select *Refine Boundaries* from the *Drainage* menu.
8. Select the *Compute Basin Data* command from the *Drainage* menu.
9. Select *OK* on the *Units* dialog.

8.5.7 Merging and Splitting Basins

WMS always creates a basin for each upstream branch from an outlet point. However, it is sometimes preferable to merge these two basins together.

1. Select the *Select Basins* tool .
2. Click inside the *yellow* basin.

3. Hold the SHIFT key down and click inside the *green* basin.
4. Select the *Merge Basins* command from the *Drainage* menu.

The text displaying the area of the basins disappears. This means that the basin geometry has changed, and the basin data must be recomputed.

1. Select the *Compute Basin Data* command from the *Drainage* menu.
2. Select *OK* on the *Units* dialog.
3. Select the yellow basin again.
4. Select the *Split Basins* command from the *Drainage* menu to again split the watershed into separate basins for each branch.
5. Select the *Compute Basin Data* command from the *Drainage* menu.
6. Select *OK* on the *Units* dialog.

8.5.8 Deleting and Existing Outlet

You can eliminate outlets once they have been created by doing the following.

1. Select the *Select Vertex* tool .
2. Select the outlet of the stream branch of the basin you just split.
3. Select the *Delete Outlets* command from the *Streams* menu.
4. Select the *Compute Basin Data* command from the *Drainage* menu.
5. Select *OK* on the *Units* dialog.

8.5.9 Basin Label Text

By default some text annotations for the basin geometric parameters are placed at the centroid of the basin. You can view any of the computed parameters by:

1. Select the *Display Options* from the *Display* menu.
2. Select the *Drainage* tab.
3. Examine all of the different parameters. Turn on *Basin slope* and *Average overland flow*.
4. Select the *OK* button.

Text showing these values, in addition to the area and id, now appears at the basin centroid. When many attributes are displayed, or for larger models, it is sometimes convenient to relocate the display of text attributes.

1. Select the *Place Basin Labels* tool .
2. Click inside of one of the basins and while holding the mouse button down, drag the cursor to a region outside of the TIN but relatively close to the selected basin and release the mouse button.
3. Select the *Refresh* command from the *Display* menu.

The label moves to the location where you let up on the mouse button and an arrow from this point to the location where you first clicked inside the basin is drawn. If you want to place the label in a different position (or have the arrow point to a different location inside the basin) repeat the process.

8.6 Adding Annotations to a Project

Annotations, including text, lines (with or without arrows) and rectangles and ellipses (filled or outlined). This section of the tutorial could be part of any or all of the tutorials but is included here to highlight the use of the annotation tools.

1. Switch to the Map module .
2. Select the create text tool .
3. Click on any point near the boundary of the defined watershed. A dialog will appear which allows you to specify the text string, the color and the font. Type "Jones Canyon Basin" in the text edit field and change the color to blue.
4. Select the OK button.

If the text did not appear where you want, or if you would like to move it after it has been created you can do the following:

1. Select the select drawing object tool .
2. Click on the text. It will be outlined with a box.
3. Click anywhere in the box and while holding down the cursor drag the text to a new location.
4. Select the create line tool .

5. Click on a point that is near the text and then double-click on a point in the center of the brown (upper) basin.
6. Select the select drawing object tool .
7. Double-click somewhere along the newly created line. This is equivalent to clicking once on the line and then choosing the attributes command as was done with the rectangle. Double-clicking to change attributes can be done with any of the drawing objects.
8. Change the line arrowhead attributes so that an arrow is placed at the end.
9. Change any other attributes you wish and select the OK button.

Your display should look similar to Figure 8-4. Try adding other annotations and experiment with the different options available to each drawing object.

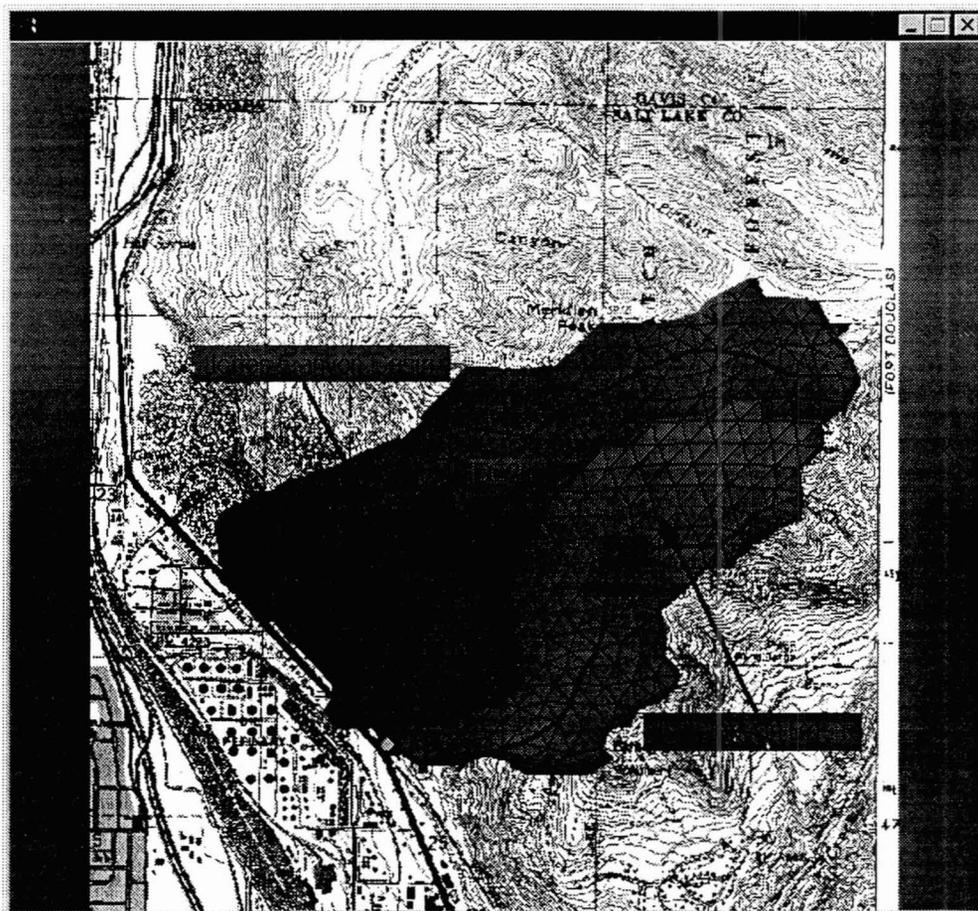


Figure 8-4 Annotations for Jones Canyon Watershed.

8.7 Conclusions

In this tutorial you have learned how to use images in conjunction with DEM and feature object data to define a TIN that can be used for basin delineation. You have also learned how to use annotations to “dress” up the final display of any project.

Watershed Delineation

Supplementary Exercise for Basin Delineation using TINs-Session 7

This exercise will give you additional practice at creating a watershed from background elevation data and a map file.

All files for this exercise are located in the *Supplimentary 7* directory.

1. Spanish Fork Canyon

- a) Import the following USGS DEM files:
 wallsrdg.dem
 bridal.dem
- b) Read in the following map file:
 sforig.map (if you created this in the Map supplementary exercise then you may wish to use your own map file).
- c) Create a TIN from the map and DEM file, you may wish to redistribute the vertices first.
- d) Eliminate pits or any other TIN anomaly.
- e) Add at least two additional outlets to the stream.
- f) Delineate the watershed.
- g) Correct any split flow problems.
- h) Refine Boundaries
- i) Compute basin data.
- j) Save the tin as mysf.tin

2. Jordanelle

- a) Import the following USGS DEM files:
 slc_e.dem
 slc_w.dem
- b) Read in the following map file:
 jordanel.map
- c) Create a TIN from the map and DEM file, you may wish to redistribute the vertices first.
- d) Eliminate pits or any other TIN anomalies.
- e) Add additional outlets to the stream.
- f) Delineate the watershed.
- g) Correct any split flow problems.
- h) Refine Boundaries
- i) Compute basin data.
- j) Save the TIN as myjord.tin



Mapping GIS Layers

- Drainage Coverage is primary layer in WMS
- Additional Coverages used to map attributes
 - Land Use
 - Soil
 - Rainfall
 - Time of Travel (lines rather than polygons)
- Some coverages allow use of either vector or raster data

Map Overlay



Weighted Average

Predominant

Computation Step

- If gridded data are used the grid cell size determines the computation step
- If vector (coverage) data are used you must specify the computation time step
 - Polygons of soil, land used, etc. are rasterized (gridded) into grid cells of the specified step

Rasterizing Polygons

- Small Cell Size
 - More accurate
 - Longer to compute
- Large Cell Size
 - Less accurate
 - Faster to compute
- Find right balance



Applications

- Composite Curve Numbers
- Runoff coefficients
- Time of Travel for computing t_c and lag

The NRCS (SCS) Curve Number (CN)

- Much has been written
 - many supporters
 - just as many opponents
- The fact is, it is commonly used and comes up time and time again with different models
 - HEC-1, TR-20, TR-55

How are CN's Derived

- Based on two physical characteristics:
 - hydrologic soil type
 - land use
- A matrix of CN numbers based on these two factors is used to determine the most appropriate value for a basin
- Large CN values correspond to high runoff

Hydrologic Soil Classifications

- A - high infiltration rates
 - .30 - .45 in/hr (sands, deep loess, aggregated silts)
- B - moderate infiltration rates
 - .15 - .30 in/hr (shallow loess and sandy loams)
- C - slow infiltration rates
 - .05 - .15 in/hr (clay loams, low organics, high clay)
- D - very slow infiltration rates
 - 0.0 - .05 in/hr (high water table, clay layer near surface)

Land Use

- The type of land use affects how water is “held” on the land surface until it can transpire, evaporate, or infiltrate
- Impervious areas are often treated separately

Typical Table of Values

Land Use	A	B	C	D
Fully developed urban - Poor	68	79	86	89
Fully developed urban - Good	39	61	74	80
Pasture, grassland, or range - Poor	68	79	86	89
Pasture, grassland, or range - Fair	49	69	79	84
Pasture, grassland, or range - Good	39	61	74	80
Meadow	30	58	71	78
Wood-grass combination - orchard - Poor	57	73	82	86
Wood-grass combination - orchard - Fair	43	65	76	82
Wood-grass combination - orchard - Good	32	58	72	79

Composite Curve Numbers

- Watersheds can never be characterized by a single value for CN
 - you should try to subdivide the watershed along boundaries that are hydrologically similar
- Area weighted averaging techniques can be used to develop a composite curve number

Automating Composite CN Calculations in WMS

- Data defined through coverages or grids
 - hydrologic soil type
 - land use
 - table relating the two
- Computations slightly different for TIN, DEM, or Feature Object based watersheds

Computations for Feature Object Based Watersheds

- Finds the land use coverage or grid
- Finds the soil type coverage or grid
- Computing the Composite CN-
 - For each basin polygon, a temporary grid is created.
 - For each point in this grid, WMS computes the CN based on the soil type ID (0 = A, 1 = B, 2 = C, 3 = D) and land use ID at this point.

Computations for Feature Object Based Watersheds (Continued)

- Composite CN Equation

$$CN = \frac{\sum_{i=1}^n CN_i \times A_i}{A}$$

- n = The number of grid cells inside the basin.
- CN_i = CN for a grid cell inside the basin.
- A_i = The area of the i^{th} grid cell inside the basin.
- A = The area of the basin.

Computations for DEM Based Watersheds

- DEM-based watersheds use the same method as feature object-based watersheds.
- A soil type coverage or grid is required.
- A land use coverage or grid is required.

Computations for TIN Based Watersheds

- Finds the land use coverage or grid
- Finds the soil type coverage or grid
- Land use and soil type values are assigned to each triangle based on land use grid/coverage and soil type grid/coverage data
- The Composite CN is computed for each basin

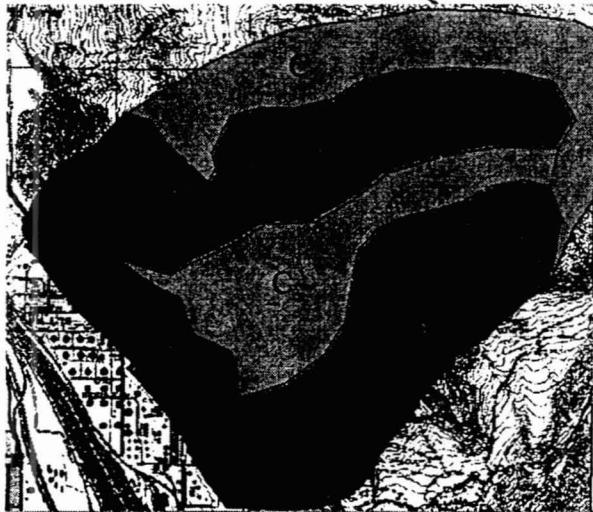
Computations for TIN Based Watersheds (Continued)

- Composite CN Equation

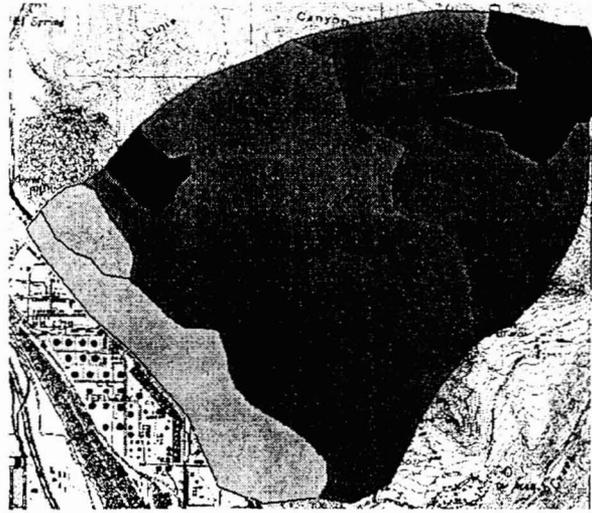
$$CN = \frac{\sum_{i=1}^n CN_i \times A_i}{A}$$

- n = The number of triangles inside the basin.
- CN_i = CN for a triangle inside the basin.
- A_i = Area of the i^{th} triangle inside the basin.
- A = The area of the basin.

Hydrologic Soil Coverage



Land Use



Composite CN Values



Steps for Computing the Composite CN in WMS

- Delineate a watershed model
 - Feature Objects, DEMs, or TINs
- Read in a soil type coverage or grid
- Read in a land use coverage or grid
- Read in a table relating land use and soil type to curve number
- Compute the composite curve number

Considerations for Computing the Composite CN in WMS

- The land use and soil type data should cover the entire watershed model. If it does not, incorrect CN's may be computed for some basins.
- When computing CN's, WMS uses data from a land use coverage/grid as an ID. The data from a soil type coverage/grid are used directly to determine the soil type (0=type A, 1=type B, 2=type C, and 3= type D)

Automating Green-Ampt Calculations in WMS

- Data defined through coverages or grids
 - Soil type ID's
 - Land use ID's
 - Table requirements:
 - Table relating soil type ID to XKSAT, RTIMP, and % Effective values.
 - Table relating land use to I_a , RTIMP, and Percent Vegetation values.
- Computations slightly different for TIN, DEM, or Feature Object based watersheds

Automating Green-Ampt Calculations in WMS

- Land use/Soil Type table ID's and land use coverage/grid ID's should match.

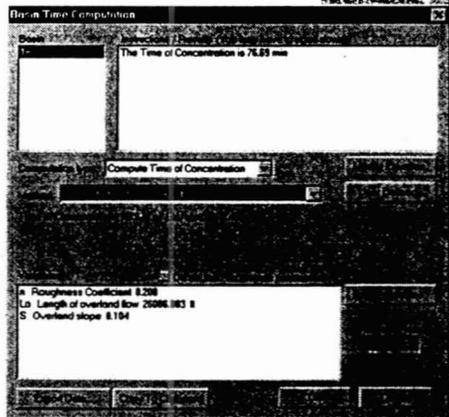
ID	I_a	RTIMP	Percent Veg.	Land Use Description
76	10,300	115,00	150,00	Low Density Residential
77	10,250	130,00	150,00	Medium Density Residential
90	10,300	115,00	150,00	Public and Quasi-Public
82	10,150	155,00	160,00	Retail/Commercial
86	10,250	145,00	150,00	Vacant

ID	XKSAT	RTIMP	Pct. Effective	Soil Description
3	10,58	10,00	10,00	Antho-Carrizo-Maripo complex
6	10,62	150,00	10,00	Anthony-Arizo complex
8	10,96	150,00	10,00	Arizo cobbly sandy loam
10	10,94	150,00	10,00	Brics-Carrizo complex, 1-5% Slopes
13	10,01	150,00	10,00	Carefree-Beardsley complex
22	10,04	150,00	10,00	Contine clay loam
24	10,02	150,00	10,00	continental clay loam, 0-3% Slopes

Steps for Computing Green-Ampt parameters in WMS

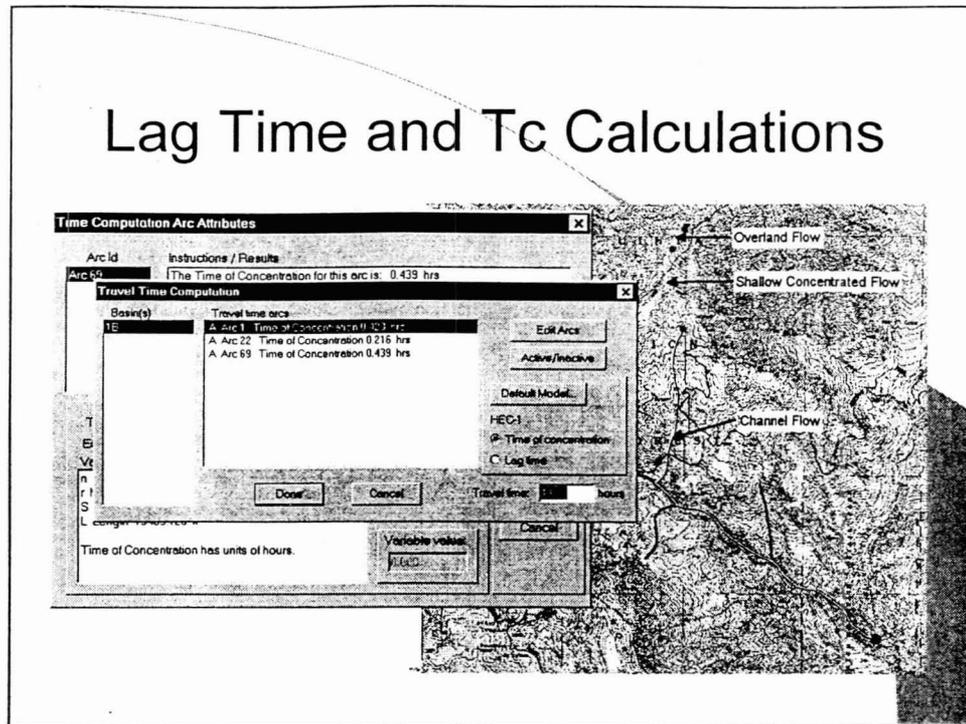
- Delineate a watershed model
- Read in a soil type coverage or grid
- Read in a land use coverage or grid
- Read in two tables:
 - Table relating soil type ID to XKSAT, RTIMP, and % Effective values.
 - Table relating land use to I_a , RTIMP, and Percent Vegetation values.
- Compute the Green-Ampt parameters

Lag Time and T_c Calculations



A=6.11 m² /
BS=0.2821 ft/ft
AOFD=1221.35 ft
MFD=28086.80 ft
MFS=0.1036 ft/ft

Lag Time and Tc Calculations



Steps for Computing Travel Times

- Delineate a watershed
- Create a Time Computation Coverage
- Define separate arcs within a basin representing different flow segment types
- Assign an equation to each arc
- Define required parameters and compute arc travel time
- Sum arc travel times within a basin

Computing Curve Numbers

Estimated Time: 45 minutes.

One of the most important parameters to compute when running a hydrologic simulation is the curve number or runoff coefficient for a basin. Besides being one of the most sensitive parameters, the curve number is also one of the most difficult parameters to compute. Fortunately, WMS has tools that make computing curve numbers for a basin a simple task.

This tutorial will outline the different methods that can be used to compute composite curve numbers for delineated drainage basins and sub-basins. First, curve numbers will be computed for a drainage coverage using a land use grid and a soil type grid. Next, curve numbers will be computed for a drainage coverage using a land use polygon coverage and a soil type grid. Finally, curve numbers will be generated for a TIN using a land use polygon coverage and a soil type polygon coverage.

In addition to computing composite *curve numbers*, WMS also computes composite *runoff coefficients* for delineated basins and sub-basins. The final section of this tutorial will show how composite runoff coefficients can be computed for a drainage coverage using gridded soil data.

15.1 Computing Composite Curve Numbers

In this tutorial, you will learn how to compute composite curve numbers using several different methods. The topics covered include the following:

- Reading in a land use grid.

- Reading in a soil type grid.
- Reading in and defining sub-basins in a drainage coverage.
- Importing a land use table.
- Computing curve numbers for a drainage coverage.
- Reading in a land use polygon shapefile.
- Computing curve numbers for a drainage coverage using a land use polygon coverage and a soil type grid.
- Reading in a soil type polygon shapefile.
- Reading in a TIN with defined sub-basins.
- Computing curve numbers for a TIN using a land use polygon coverage and a soil type polygon coverage.

15.2 Reading a Land Use Grid

Arc/Info and GRASS ASCII grids can be read into WMS. These grids can either represent land use or soil type. The first step in computing composite curve numbers for a drainage coverage is to read in a land use grid:

1. Select *New* from the *File* menu.
2. Confirm that you want to delete everything.
3. Select the *Import* command from the *File* menu.
4. Select the *Gridded Attributes->WMS Attributes* option and select the *OK* button.
5. Select the *Land use* option for the *Mask*.
6. Select *Arc/Info* for the file type, and select the *OK* button.
7. Select and open "*asplugrd.asc*". Wait a few seconds while the land use grid is being displayed on your screen.

15.3 Reading in a Soil Type Grid

The next step is to read in a soil type grid. Soil type grids in Arc/Info or GRASS ASCII format can be read into WMS. In this section, you will read in

an Arc/Info ASCII soil type grid prepared from polygon data for the Aspen Grove watershed.

1. Select the *Import* command from the *File* menu.
2. Select the *Gridded Attributes->WMS Attributes* option and select the *OK* button.
3. Select the *Soil type* option.
4. Select *Arc/Info* for the file type, and select the *OK* button.
5. Find and open "*aspsogrd.asc*". Wait a few seconds while the soil type grid is being displayed on your screen.

15.4 Reading in and Defining Sub-basins in a Drainage Coverage

The next step is to define your model or read in an existing model.

1. Select the *Open* command from the *File* menu.
2. Type "*.sup" in the file name window so only WMS super files are shown. Select and open "*aspencn.sup*".

15.5 Importing a Land Use Table

Next, you will import a land use table. This land use table relates land use and soil type to curve number values. You have already read in a land use grid, a soil type grid, and a drainage basin model. This table, relating land use and soil type to actual curve number values, is the final piece of the puzzle to compute curve numbers for each basin.

1. Select the *Hydrologic Modeling module* icon .
2. Select the *Compute Runoff Coeff/CN...* command from the *Calculators* menu.
3. Select the *Import* button.
4. Find and open "*scstut.tbl*". Notice that the table appears in the *Mapping* text window after it is imported.

15.6 Computing Curve Numbers for a Drainage Coverage

The final step is to compute the curve numbers for the drainage coverage. This step is simple since WMS does this automatically. As an overview, the following data are required to compute composite curve numbers for a drainage coverage:

- A drainage coverage with outlet points and sub-basins defined.
- A land use grid or a land use polygon coverage.
- A soil type grid or a soil type polygon coverage.
- A land use table relating land use and soil type to curve number.

Now do the following:

1. Insure that *SCS Curve numbers* is selected for the computation type. For both soil type and land use the default option of using a grid for computation should be selected.
2. Select the OK button.

Curve numbers will be computed and displayed for each of the basins.

15.7 Reading in a Land Use Polygon Shapefile

Now that you have computed curve numbers for the Aspen Grove sub-basins using a land use and soil type grid, you will compute curve numbers using a land use coverage and a soil type grid. To do this, you must first create a new land use coverage and read in a land use shapefile:

11. Select the *Map module* icon .
12. Select the *Coverages...* command from the *Feature Objects* menu.
13. Select the *New* button in the *Coverages* window.
14. Change the name of the coverage to *Land Use* and change the coverage's *Attribute set* to *Land Use*.
15. Select the *OK* button in the *Coverages* window.
16. Select the *Import...* command from the *File* menu.
17. Select the *Shape file->Feature objects* option and select the *OK* button.

18. In the *Import Shapefile Data* window, select the polygons file browser button.
19. Find and open "*aspenlu.shp*".
20. In the polygon attribute text window, notice that the *LU_CODE* database field is automatically mapped to the *Land use* attribute in WMS. This occurred only because the attribute name in the land use file was *LU_CODE*. If the attribute name is anything other than *LU_CODE* it can still be assigned to the WMS Land Use variable manually using the *Attribute mapping...* dialog.
21. Select *OK* and wait a few seconds while the land use shapefile for the Aspen Grove watershed is read in and displayed.

15.8 Computing Curve Numbers for a Drainage Coverage Using a Land Use Polygon Coverage and a Soil Type Grid

Now that you have imported a land use polygon coverage, you are ready to compute composite curve numbers for your drainage coverage using the land use polygon coverage you just imported and the soil type grid.

1. Again select the *Coverages...* command from the *Feature Objects* menu.
2. Select the *Drainage coverage* in the window displaying all the coverages and make it the active coverage by selecting the *Active* button. Select *OK* in the coverages window.
3. Select the *Hydrologic Modeling module* icon .
4. Select the *Compute Runoff Coeff/CN...* command from the *Calculators* menu.
5. Select the option to compute SCS Curve numbers using a soil type grid and a land use coverage (You will have to select the *Use a land use coverage for determining land use* option).
6. Enter a *Drainage Coverage Computation step* of 50.0 (Meters in this case).
7. Select the *OK* button in the *Compute Composite Runoff Coefficients/CN's* window. The computation may take anywhere from a minute to several minutes, depending on the speed of your computer. The computed curve numbers using this method should match the previously computed curve numbers.

When the curve number is computed for the drainage coverage, each polygon in the drainage coverage is visited. For each polygon, a temporary grid is created around the polygon with a cell width equal to the user-specified computation step. Each cell in this temporary grid is visited. For each temporary grid cell that is inside a basin, the curve number is computed (if it can be). Thus, the smaller the computation step, the longer WMS will take to compute the curve numbers for the basins. This computation step should be about equal to the smallest soil type or land use grid cell size (depending on whether you are using a soil type or land use grid to compute the composite curve number).

15.9 Reading in a Soil Type Polygon Shapefile

Besides being able to compute curve numbers for drainage coverages, WMS also computes curve numbers for TINs in much the same way. In this section, you will compute curve numbers for the same basins in a TIN-based model of the Aspen Grove watershed. Also, instead of using grids to compute the composite curve numbers for this model, you will import a soil type polygon shapefile and use a soil type polygon coverage and a land use polygon coverage.

1. Select the *Map module* icon .
2. Select the *Coverages...* command from the *Feature Objects* menu.
3. Select the *New* button in the *Coverages* window.
4. Change the name of the coverage to *Soil Type* and change the coverage's *Attribute set* to *Soil Type*.
5. Select the "Drainage coverage" and select the *Delete* button. Select *OK* when prompted for confirmation.
6. Select the *OK* button in the *Coverages* window.
7. Select the *Import...* command from the *File* menu.
8. Select the *Shape file->Feature objects* option and select the *OK* button.
9. In the *Import Shapefile Data* window, select the polygon file browser button.
10. Select and open "*aspenso.shp*".
11. In the polygon attribute text window, notice that the *HYDGRP* database field is automatically mapped to the *SCS soil type* attribute in WMS. Since we want to map *HYDGRP* to WMS *SCS soil type* and

there is nothing else to map, select the *OK* button. If your database field for hydrologic soil group is named something other than *HYDGRP* then you can manually map it to the WMS parameter by selecting the *Attribute mapping* button and defining which database field represents hydrologic soil group.

12. Select *OK* and wait a few seconds while the soil type shapefile for the Aspen Grove watershed is read in and displayed.

15.10 Reading in a TIN with Defined Sub-basins

Now that you have a land use polygon coverage defined and a soil type polygon coverage defined, you can import a TIN containing defined sub-basins. Then, you can compute the composite curve numbers for your TIN using the land use and soil type polygon coverages.

1. Select the *TIN module* icon .
2. Select the *Open...* command from the *File* menu.
3. Select and open "*aspentin.sup*".

15.11 Computing Curve Numbers for a TIN Using a Land Use Polygon Coverage and a Soil Type Polygon Coverage

In this section, you will compute composite curve numbers for your TIN file using a land use polygon coverage and a soil type polygon coverage.

1. Select the *Display Options* command from the *Display* menu.
2. Select the *Drainage* tab.
3. Turn on the display of *Basin CN's*.
4. Select *OK*.
5. Select the *Tree module* icon .
6. Select the *Compute Runoff Coeff/CN...* command from the *Calculators* menu.
7. Select the option to compute SCS Curve numbers using a soil type coverage and a land use coverage (You will have to select the *Use a soil type coverage for determining soil type* and the *Use a land use coverage for determining land use options*).

8. You do not need to enter a computation step when computing the curve number for a TIN. Just leave this value alone.
9. Select the *Import* button.
10. Find and open "*scstut.tbl*".
11. Select the *OK* button in the *Compute Composite Runoff Coefficients/CN's* window. The computation may take anywhere from a few seconds to several minutes, depending on the speed of your computer. The computed curve numbers using this method are slightly different than the curve numbers computed for the drainage coverage. This is because the shape of the basins are also slightly different.

15.12 Computing Composite Runoff Coefficients for a Drainage Coverage Using a Soil Type Grid

Besides being able to compute composite curve numbers, you can also compute composite runoff coefficients in WMS. These runoff coefficients can be computed from a soil type grid and can be used in a Rational method simulation.

1. Select the *New* command from the *File* menu. Select *OK* when prompted if you want to delete everything.
2. Select the *Map module* icon .
3. Select the *Open...* command from the *File* menu.
4. Find and open "*aspencn.sup*".
5. Select the *Import...* command from the *File* menu.
6. Select the *Gridded Attributes->WMS Attributes* option and select the *OK* button.
7. Select the *Soil type* option for the mask.
8. Select *Arc/Info* for the file type, and select the *OK* button.
9. Select and open "*aspsogrd.asc*".
10. Select the *Tree module* icon .
11. Select the *Compute Runoff Coeff/CN...* command from the *Calculators* menu.
12. Select the *Rational method runoff coefficients computation* option.

13. Select the *Import* button in the *Compute Composite Runoff Coefficients/CN's* window.
14. Find and open "*soiltype.tbl*". Notice that the table appears in the *Compute Composite Runoff Coefficients/CN's* window after it is imported.
15. For soil type, select the option to *Use a soil type grid for determining soil type*.
16. Select the *OK* button.

You have now computed the runoff coefficients for your watershed. The computed runoff coefficient values can be used in the Rational Method to determine watershed runoff. You can view these values by doing the following:

1. Select any of the square basin icons in the *Hydrologic Modeling Tree Window*.
2. Select the *Run Simulation* command from the *Rational* menu.

You should see the composite runoff coefficient for this basin. You can view the values of the other basins by selecting their respective basin icons.

15.13 Conclusions

This concludes the Curve number computation tutorial. You are encouraged to experiment with all of the commands explained in this section until you feel comfortable with the tools available to you for computing curve numbers.

In this section, you have learned how to:

- Compute curve numbers (or runoff coefficients) for drainage coverages and TINs using:
 - soil coverages or soil grids
 - land use coverages or land use grids
 - tables relating land use to CN or soil type to runoff coefficient.

Time of Concentration Calculations

Estimated Time: 30 minutes.

Travel times (time of concentration, lag time, and travel time along a routing reach) are critical to performing analysis with any of the hydrologic models. In this tutorial you will learn about the two different ways WMS can be used to compute time of concentration for a TR-55 simulation (lag times are computed in the same way):

- Runoff distances and slopes for each basin are automatically computed whenever you create watershed models from TINs or DEMs and compute basin data. These values can then be used in one of several available equations in WMS to compute lag time or time of concentration.
- If you want to have a little more control (and documentation) over the lag time or time of concentration, you will use a time computation coverage to define critical flow paths. Time computation coverages contain flow path arc(s) for each sub-basin. An equation to estimate travel time is assigned to each arc and the time of concentration (or lag time) is the sum of the travel times of all arcs within a basin. Lengths are taken from the length of the arc and slopes derived if a TIN or DEM are present.

In this tutorial you will compute the time of concentration for the two sub-basins and the travel time between outlet points in the watershed shown below. You will use the TR-55 library of equations, but you could just as easily use one of the other pre-defined equations, or enter your own equation.

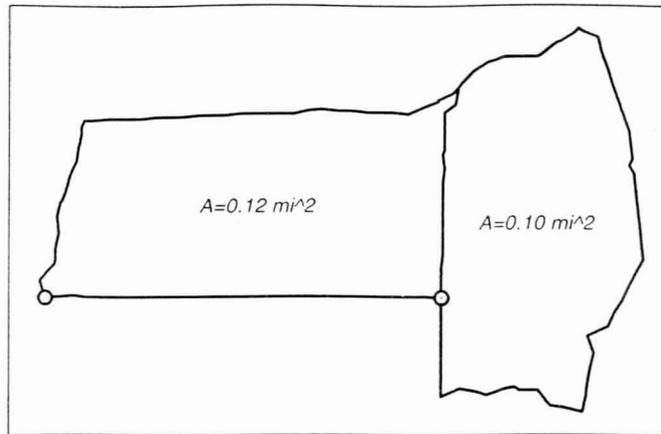


Figure 10-1 American Fork Watershed

10.1 Launching WMS

If you haven't already, start WMS by double-clicking on the *WMS* icon on the desktop or choosing *WMS 6.0* from the Start menu.

10.2 Reading a TIN File

You will first read in a TIN that has already been processed and used to delineate two sub-basins.

1. Select the *TINs* module .
2. Select the *Open* command from the *File* menu.
3. Find and *Open* the file named *aftr55.tin*. By default it should be in the tutorial directory.
4. Select the *Compute Basin Data* command from the *Drainage* menu.
5. Select *Feet* for model units, *square miles* for *Basin Areas*, and *Feet* for *Distances*.
6. If the display options for triangles and vertices are on turn them off by selecting the *Display Options* from the *Display* menu and toggling off the check box in front of the vertices and triangles display options.
7. Select *OK*.

Your display should now show the basin boundaries and stream network.

10.3 Defining Flow Path Arcs

Flow paths can automatically be traced across a TIN (or DEM with flow directions computed) using the flowpath tool. You can take advantage of this functionality in *WMS* to set up your flow path segments.

1. Select the *Map* module .
2. Select the *Coverages* command from the *Feature Objects* menu.
3. Set the default coverage type to *Time Computation* in the *Attribute Set* drop down combo box.
4. Confirm that you want to change coverage types by selecting *OK*.
5. Select the *Create Feature Points* tool .
6. Create feature points at the two locations marked by an X in the figure below. Be sure that there is one point inside of each basin boundary.

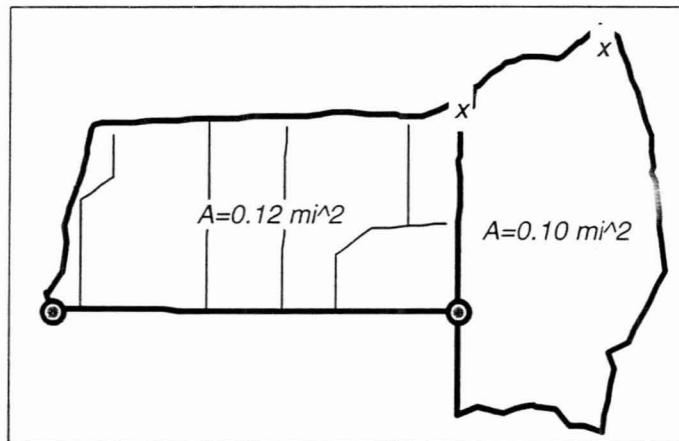


Figure 10-2 Starting Locations for Time of Concentration Arcs.

These points represent the furthest point from the outlet for their respective basins. Now feature arcs will be created from these points to the outlet by following the flow path.

1. Select the *Select Feature Point/Node* tool .
2. Select both feature points just created. Use the shift key to multi-select (the last point you just created will likely already be selected).
3. Select the *Node->Flow Arc* command from the *Feature Objects* menu.
4. Choose the *Create multiple arcs* command.
5. Select *OK*.

The *Create multiple arcs* option will cause WMS to break the flow path arcs once they enter the stream defined on the TIN. The TR-55 method (others are similar as well) use three different flow segments to compute time of concentration: sheet flow (up to 300 feet), shallow concentrated flow, and open channel flow. Since WMS will automatically break the arcs between overland and channel flow two of three segments will already be defined. You will need to separate the sheet flow from the shallow concentrated flow before setting up the equations.

1. Select the *Select Feature Vertex* tool .
2. The figure below identifies locations approximately 200-300 feet downstream from the beginning of the flow path arc. Select one of these vertices.
3. Select the *Vertex<->Node* command in the *Feature Objects* menu.
4. Repeat for the other flow arc (alternatively you could multi-select the two vertices and convert them to nodes at the same time).

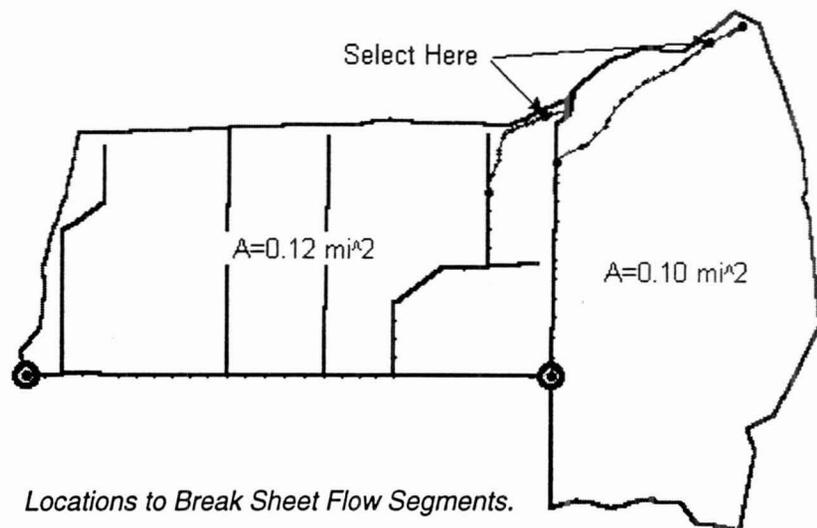


Figure 10-3 Locations to Break Sheet Flow Segments.

You should now have three arc segments for each basin. These arcs will be used to compute a time of concentration for the TR-55 analysis. However we will also want to compute a travel time for the flow from the upper basin to the lower basin. This will require a flow path segment between the upstream outlet and the lower outlet. The flow path arcs from the lower basin already include flow for part of this distance, but there remains a segment for which no flow path arc exists.

1. Select the *Streams->Flow Arcs* command from the *Feature Objects* menu.

This will create an arc that joins the outlet of the upper basin with the flow path segment already created for the lower basin. Where the flow path segments meet the arc from the lower basin is split into two. This will leave

four arcs to define time of concentration for the lower basin (two will actually be in the stream). The arc just created, while in the lower basin, will not be used when computing time of concentration.

You have used the Node->Flow Arcs and Streams->Flow Arcs commands to automatically generate the flow path segments from flow on a TIN. Similarly arcs could have been created from flow on a DEM with flow directions, or they could be created manually using the create arc tool . When defining arcs manually you must keep in mind that, just like stream arcs in the drainage coverage, direction is important. Time computation arcs should be defined from downstream to upstream.

10.4 Assigning Equations to Time Computation Arcs

With the flow path segments defined you are now ready to assign equations that will be used to determine a travel time for the arc. Use the following figure as a guide while defining the equations.

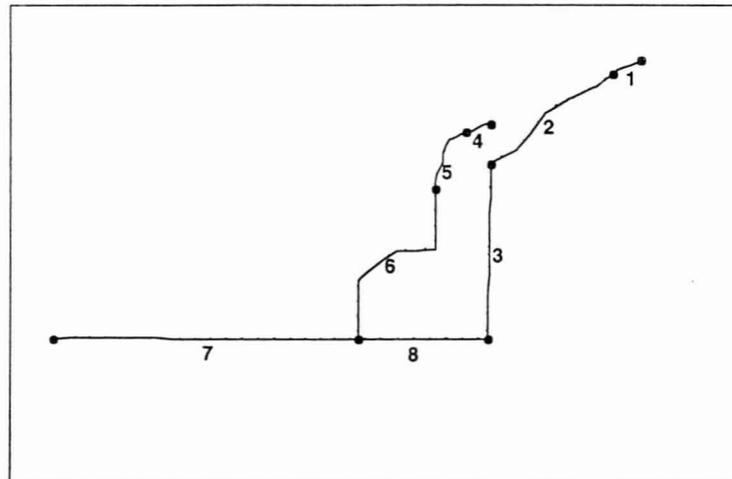


Figure 10-4 Time Computation Arcs.

1. Select the *Select Arc* tool .
2. Select the arc labeled 1.
3. Select the *Attributes* command from the *Feature Objects* menu.

By default the arc will be a TR-55 sheet flow equation arc, so all you need to do is define the overland manning's roughness coefficient and the 2yr-24hr rainfall. Length and slope will already be defaulted from the selected arc.

1. Click on the "*n Mannings*" line in the *Variables* text window.
2. Enter a value of 0.24 in the *Variable Value* edit window.

3. Click on the *rainfall* line in the *Variables* text window.
4. Enter a value of 1.1.

You should notice in the Instructions/Results window that you are told what variables need to be defined before a travel time can be computed. Once you have entered all the necessary values, this same window reports the travel time for this arc. In this way, you can compute travel time for any arc segment no matter what the application is.

1. Select *OK*.
2. Repeat the previous steps for the arc labeled 4. Use a Manning's roughness of 0.15 and a rainfall value of 1.1.

You have now defined equations for the overland sheet flow segments in each basin and are ready to define the next two arc segments as shallow concentrated flow.

1. Select the arc labeled 2.
2. Select the *Attributes* command in the *Feature Objects* menu (note you could have double-clicked on the arc labeled 2 to bring up this same dialog).
3. Change the arc type to *TR-55 shallow conc eqn.*
4. Click on the *paved* line in the *Variables* text window.
5. Enter "no" in the *Variable Value* edit window.
6. Select *OK*.
7. Repeat for the arc labeled 5, using the same arc type. In this case set the *paved* value to *yes*.

The remaining arcs will be defined as open channel flow arcs.

1. Select the arc labeled 3.
2. Select the *Attributes* command in the *Feature Objects* menu.
3. Change the arc type to *TR-55 Open channel eqn.*
4. Click on the *Manning's n* line.
5. Enter a value 0.016 in the *Variable Value* edit window.
6. Select the *hydraulic radius* line in the *Variables* window.

7. Select the *Hydraulic Radius* button to open up the channel calculator so that the hydraulic radius can be computed from rough estimates.
8. Change the *Channel type* to *Rectangular*.
9. Enter a longitudinal slope of 0.015.
10. Enter a width of 3.0.
11. Make sure the *Enter depth* option is chosen and enter a depth of 0.33 (an approximated depth since we do not know what the flow is at this point).
12. Select the *Calculate* button.
13. Select *OK*.

You have now defined the necessary parameters for computing travel time using the TR-55 open channel flow (Manning's) equation. Repeat for the arcs labeled 6, 7, and 8 using the following values:

1. For arc 6 use a Manning's n of 0.016, and a hydraulic radius of 0.35.
2. For arc 7 use a Manning's n of 0.016, and a hydraulic radius of 0.41.
3. For arc 8 use a Manning's n of 0.016, and a hydraulic radius of 0.38.

If you wish you can continue to experiment with the channel calculator to compute the hydraulic radius rather than entering the given values.

You now have defined equations and variable values for each flow path segment. You can change these equations, variables, add new flow path segments, etc. in order to determine the best flow paths and most appropriate equations for each basin. In other words, this process is subjective and it may take a few iterations to get the best value.

10.5 Using the Time Computation Arcs to Compute Time of Concentration for a TR-55 Simulation

Before assigning time of concentrations to each basin you need to decide which model you want to use. In this tutorial you will be running *TR-55*, but *WMS* includes similar links to use of the time computation arcs in the *TR-20* basin data dialog, the *HEC-1* Unit Hydrograph method dialog, and the *Rational Method* dialog. In addition, you can view the travel time for any arc as demonstrated in the previous section and then use/enter the value for a hand calculation or some other model.

1. Select the *Hydrologic Modeling* module .

2. Select the *Run Simulation* command from the *TR-55* menu.
3. Select the *Initialize TR-55* data button (memory to store TR-55 variables is not allocated in *WMS* until you initialize a TR-55 simulation).
4. Select the *Select Basin* tool .
5. Select the upper basin.
6. Change the *Curve Number* to 70.
7. Enter a *Rainfall* value of 1.5.
8. Change the *Rainfall distribution* to *Type II*.
9. Select the *Compute Tc – Map Data* button. You will see the three arcs that time computation arcs that are in the basin.
10. You can create a detailed report as a text file if you want by selecting the *Export Data* or *Copy to Clipboard* buttons.
11. Select *Done* and the sum of the travel times for these arcs will be used as the time of concentration for this basin.

Note that you could bring up the time computation attributes dialog and change the equation or any of the equation variables by selecting the *Edit Arcs* button. Now you will set up the TR-55 basin data for the lower basin.

As a comparison you will now compute the time of concentration using the pre-computed runoff distance and slope parameters and one of the equations defined in *WMS*. To see the values that *WMS* has computed you will need to modify the display options.

1. Select the *Done* button.
2. Select the *Display Options...* command from the *Display* menu.
3. Select the *Drainage* tab.
4. Toggle on *Max flow distance* and *Max flow slope*.
5. Select *OK*.

Notice that the longest flow path distance within a basin is displayed for each basin as well as the slope along that flow path. These are the variables that will be used in a separate equation to determine the time of concentration.

1. Once again select the upper basin.

2. Select the *Run Simulation* command from the *TR-55* menu.
3. Note that the time of concentration is currently about .57 hours and then select the *Compute TC - Basin Data* button.

In the Basin Time Computation dialog you can pick from one of several lag time or time of concentration equations that are pre-programmed in WMS (see the reference manual for a complete description of these equations). You can also create your own equation from the computed variables, or parameters that you define yourself. In this tutorial we will use the Kerby equation for time of concentration, but you may wish to experiment on your own with some of the other equations or by making your own.

1. Select the *Time of Concentration* option from the *Computation type* drop-down box.
2. Choose the *Kerby method for overland flow* equation in the drop-down list of equations. Notice that all variables needed for this equation are defined and so the time of concentration is reported in the text window at the top of the dialog.
3. Select *OK*.

You should notice that the time of concentration is very nearly the same as before (.57 hours).

At this point you might be wondering why you would ever want to create a time computation coverage and go through the effort of defining travel time arcs and assign equations to each if the same value can be determined using the parameters already computed from a TIN or DEM?

1. Select the *Compute TC - Basin Data* button again.
2. Select the *Kirpich method for overland flow on bare earth* equation.
3. Select *OK*.

You should now see that the computed time of concentration is less than half of what it was previously. You should now set the time of concentration back to the one originally computed by the flow arc method. To do this:

1. Select the *Compute Tc - Map Data* button.
2. Select *Done*.

The lesson here is the equations using Basin Data work great if you are confident the equation represents the same kind of land surface you are dealing with. If not, you are likely better off to define equations along separate flow path segments using the Map Data. Of course, in either case you can customize the equations to suit your own needs.

1. Select the lower basin from the graphics window.
2. Change the *Curve Number* to 80.
3. Enter a *Rainfall* value of 1.5.
4. Change the *Rainfall distribution* to *Type II*.
5. Select the *Compute Tc - Map* button.

This time you will notice that there are more than three arcs. This is because one of the flow path arcs (the one labeled 8 above) will be used to compute travel time between outlets but still is within the lower basin. Notice that WMS defaults it's status to inactive. WMS will try to make the best choice of arcs that are used for time of concentration for a basin, but at times you may need to set the active/inactive status on your own.

6. Select *Done*.

10.6 Using the Time Computation Arcs to Compute the Travel Time Between Outlet Points

Besides using time computation arcs for computing time of concentration or lag time for a basin, you can also use them to compute the routing travel time of a channel.

1. Select the *Select Outlet* tool .
2. Select the upstream outlet (the outlet to the upper basin).
3. Select the *Compute travel time...* button.
4. Make sure that both of the downstream arcs are active.
5. Note the time of concentration for these arcs.
6. Select *Done*; the time of concentration from the arcs should now appear in the *Travel time* box.

10.7 More TR-55

While you were entering the data for each of the two basins you should have noticed that instructions are given in the TR-55 data window to let you know what must be entered before a peak Q can be determined. Once you did enter all of the data the peak Q was computed and displayed in this same window. You can also get help for anything listed in this window.

1. Select the *Select Basin* tool .
2. Select the upper basin.
3. Select the line in the TR-55 data window where the peak Q is displayed.

Notice that the TR-55 equation for computing peak flow is displayed in the help window. You can get help for missing data as well.

1. Change the Time of concentration to 0.0.
2. Select the line in the TR-55 data window that instructs you to define a time of concentration.

Now you are told the units for time of concentration as well as the available options for determining it.

1. Reenter the time of concentration by typing in the appropriate value, or if you can't remember use the *Compute Tc - Map Data* button again.
2. Select the *Compute Hydrograph* button.
3. Select the *Show Hydrograph Window* command from the *Display* menu.
4. Click on the hydrograph icon that is displayed by the upper basin.

10.8 Conclusions

This completes the tutorial on using the time computation coverage to compute time of concentration and travel times. In the process you have also learned about the TR-55 interface. You are encouraged to continue experimenting with the time computation arcs and the TR-55 options.

Computing Green-Ampt Parameters

Supplementary Exercise for Computing Curve Numbers-Session 8

In the Curve number tutorial, you learned how to compute curve numbers and runoff coefficients using WMS. In this supplementary exercise, you will learn how to compute composite Green-ampt parameters for sub-basins using WMS.

Your county is considering switching to the Green-Ampt method for computing watershed losses. This method is more scientific than the SCS Curve Number method and should provide more accurate results. The problem with the Green-Ampt method is there's lots of data involved. Fortunately, you have WMS. Your county has assigned you, the county hydrologist, to write a new county hydrology manual in which the Green-Ampt method is used to compute runoff. Your task is to figure out how to compute Green-Ampt parameters in WMS and view these parameters.

Once you have computed the Green-Ampt parameters, you can define a HEC-1 model and compute a hydrograph using these Green-Ampt parameters. This hydrograph can then be compared with the real hydrograph at a gage station in your experimental watershed. After tailoring Green-Ampt parameters to your county's soil types and land use types, you can gather your data and write your new county hydrology manual. Your experimental watershed is the Aspen Grove watershed, located near Provo, Utah.

For this exercise, you will need the following files in the *supplementary 8* directory:

- *aspenlu.shp*-An ArcView® shape file containing land use boundaries in your experimental watershed. A land use ID associated with each polygon is included in this file.
- *aspenso.shp*-An ArcView® shape file containing soil boundaries for your experimental watershed. A soil type ID associated with each polygon is included in this file.
- *asplugrd.asc*-An Arc/Info® ASCII grid file containing land use ID's for your experimental watershed.
- *aspsogrd.asc*-An Arc/Info® ASCII grid file containing soil type ID's for your experimental watershed.
- *greenlu.tbl*-A file that relates the land use ID's in the shape files or grid files to the initial loss (IA), the percent of the basin that is impervious (RTIMP), and the Percent vegetation.
- *greenso.tbl*-A file that relates the soil type ID's in the shape files or grid files to the Hydraulic conductivity (XKSAT), the RTIMP value for the soil, and the percent of the basin where the RTIMP value is effective.
- *aspentin.tin*-A WMS TIN file containing a model of your experimental watershed.

Here's some helpful steps on how to compute the Green-Ampt parameters for the watershed sub-basins:

1. Read the file **aspentin.tin** and compute the basin data (the model units are in meters).
2. Import the files **aspenlu.shp** and **aspenso.shp** into separate land use and soil type coverages.
3. Import the files **aspsogrd.asc** and **asplugrd.asc**. These Arc/Info® ASCII grid files can be read in by selecting the *DEM Mask->DEM Attributes* option from the *Import* command in the *File* menu. Select a *land use* mask when importing **asplugrd.asc** and a *soil type* mask when importing **aspsogrd.asc**.
4. After you're done reading in the land use and soil type grids, you may want to turn them off so they are not displayed (displaying takes a long time). To this, go to the DEMs module and select the *Display Options* command. Turn off the options to display land use and soil type grids.
5. Go to the Tree module and select the *Composite Runoff Coeff/CN* command from the *Calculators* menu.
6. Select the option to *Compute Green-Ampt parameters*.
7. For *Land use mapping*, import **greenlu.tbl**. For *Soil type mapping*, import **greenso.tbl**.

- First, compute the Green-Ampt parameters using the land use and soil type grids. Enter the Green-Ampt parameters in the table below (Go to the HEC-1 Loss method window to determine the Green-Ampt for basins 1B, 3B, and 4B):

Basin Name	IA	DTHETA	PSIF	XKSAT	RTIMP
1B					
3B					
4B	(0.2731)	(0.3500)	(3.2757)	(0.9203)	(26.7289)

- Next, compute the Green-Ampt parameters using the land use and soil type coverages. Enter the Green-Ampt parameters in the table below:

Basin Name	IA	DTHETA	PSIF	XKSAT	RTIMP
1B					
3B					
4B	(0.2731)	(0.3500)	(3.2741)	(0.9215)	(26.7289)

- Finally, compute the Green-Ampt parameters using the land use grid and soil type coverage. Enter the Green-Ampt parameters in the table below:

Basin Name	IA	DTHETA	PSIF	XKSAT	RTIMP
1B					
3B					
4B	(0.2731)	(0.3500)	(3.2741)	(0.9215)	(26.7289)

Notice that computing Green-Ampt parameters using grids is much faster and yields similar values. The disadvantage of the grid method is that it uses more of the computer's memory.



So What About a Hydrograph?

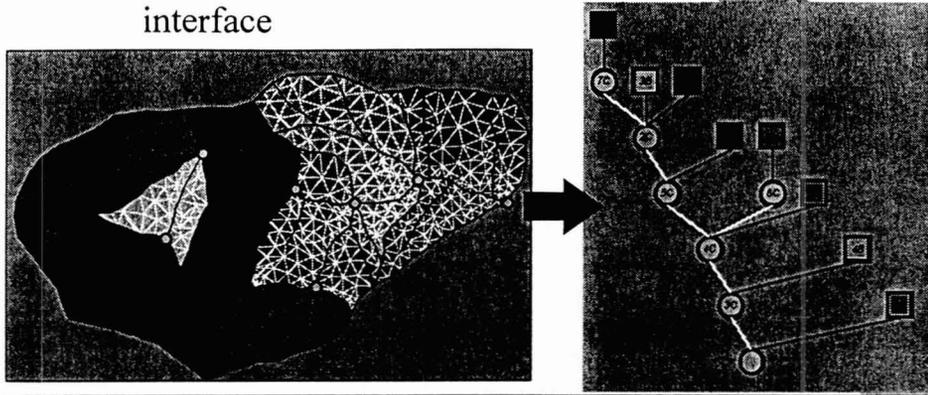
- Statistical Methods
 - regression analysis
- Deterministic
 - rainfall characteristics required
 - algorithms for determining
 - losses (excess)
 - conversion of excess to flood runoff

Interfaces in WMS

- Regression
 - NFF
- Deterministic
 - HEC-1
 - TR-20
 - TR-55
 - Rational

What is a Topologic Tree?

- Topologic representation of the watershed
 - All models in WMS use the Tree developed from geographic data to access the specific interface



More Topologic Trees

- Basins and outlets can be selected from the geographic data or the tree
- Trees can be created without geographic data
 - more detail in a later lesson
 - additions to trees must be made in the Terrain Window when a geographic data is present

NFF

- NFF - National Flood Frequency Program
- Regional regression equations
 - equations created on a state-by-state basis
- Compiled into the NFF program
 - USGS
 - FHWA
 - FEMA

What are the Regression Equations?

- Transfers flood characteristics from gaged to ungaged watersheds
- Through a set of characteristic variables
 - watershed
 - climatic
- Estimates of flood-peak discharges and estimates of typical flood hydrograph for specified recurrence intervals

How are Equations Created?

- State subdivided into “homogeneous” hydrologic regions
- T-year flood-peaks for gaged locations determined using a Pearson Type III distribution
- Least squares regression analysis used to relate peak floods to watershed and climatic characteristics

Regression Equations

- Equations are generally reported in the following form:

$$RQ_T = aA^bS^cP^d$$

RQ_T - T-year rural flood-peak discharge

A - drainage area

S - channel slope

P - mean annual precipitation

a,b,c,d - regression coefficients

Adding Your Own Equation

- Equations only update every 3-4 years
- Your state/area may have a different or updated equation
- You can place a file named "custnff.equ" in WMS directory
 - See 14.2.6 and 20.23 of WMS reference for more information

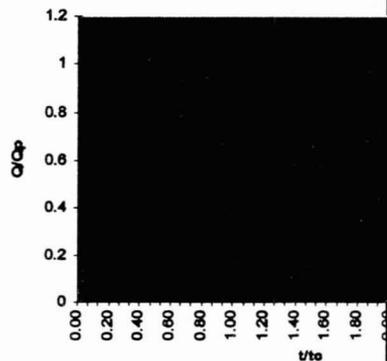
Basin Characteristics

Watershed Characteristic	#States	Climatic Characteristic	#states
Drainage Area	51	Mean annual precip.	19
Channel Slope	27	Rainfall for a duration	14
Storage area lakes/ponds	16	Forest cover	8
Elevation of watershed	13	Min. mean Jan. temp	4
Channel length	6	Mean annual snowfall	2
Basin shape	4		
Soil characteristics	3		
Stratified drift	1		
High elevation index	1		
Relative relief	1		
Drainage frequency	1		

Hydrographs from NFF

- Typical hydrograph can be computed from Q
 - An average, not representative of any particular rainfall
- Composed of three essential parts:
 - Peak discharge
 - Basin lag time
 - Dimensionless hydrograph ordinates

The NFF Dimensionless Hydrograph



Applicability & Limitations

- Rural equations should only be used for rural watersheds
- Should not be used in conjunctions with dams/detention ponds affect peak discharge
- Should use parameter values within the ranges identified by the NFF program
- Drainage area must always be computed
- Curves for watersheds in two different regions can only be computed if both have equations for the desired T-year interval. No provision for watersheds in two states
- To insure peak discharges do not exceed the max envelope, care must be taken to choose the maximum flood-region
- No estimates are allowed for the 500-year peak discharge - it may be less
- Some regions do not have a 100-year equation - examine frequency plots if some form of extrapolation will be used
- Hydrographs do not accurately account for snowmelt runoff, or watersheds which typically have a double peak

How Does NFF Work in WMS?

- 1 Delineate a watershed
- 2 Initialize an NFF analysis
- 3 Define maximum flood-region
- 4 Select the state and hydrologic region
- 5 Define any climatic variables
- 6 Compute peak discharges
- 7 Generate a hydrograph
 - Must define a basin lag time

National Flood Frequency Program (NFF) Interface

Estimated Time: 20 minutes.

The National Flood Frequency program developed by the USGS provides a quick and easy way of estimating peak flow values and hydrographs at ungaged sites. This data can be used in the design of bridges and culverts, flood-control structures, and flood-plain management. It utilizes regression equations which have been developed for each state. Besides an interface to the NFF program, WMS can be used to calculate many of the variables used by the regression equations.

In this tutorial you will use an existing TIN, "rockynff.tin", and the regression equations for Texas. The TIN allows you to compute basin area and slope, each used in the Texas regression equations. A TIN is not necessary to use NFF and you may wish to construct only a tree if you already have values for your state's equation variables.

11.1 Launching WMS

Before launching *WMS*, you should have installed *WMS* on your computer according to the directions in the *WMS Installation Guide*. If you have not yet installed *WMS*, please do so before continuing.

To launch the PC version of *WMS*:

Double click on the *WMS* icon .

To launch the Unix version of *WMS*:

1. Bring up an xterm window.
2. Change to the directory containing the *WMS* executable.
3. type **wms** at the command line.

11.1.1 Normal Mode Operation

If you have not licensed the digital terrain and map modules you will need to switch to *Demo* mode by selecting the *Demo Version On* command in order to complete this tutorial.

If you are not licensed to run NFF, you can still run this tutorial to evaluate the interface, and determine of what value it can be for your work. The demo tutorial is subject to the following restrictions:

1. Texas is the only state available to test.
2. Only the values specified in the tutorial may be used to do actual computations.

11.2 Reading a TIN File

You will first read in a TIN which has already been processed and used to delineate a single basin.

8. Select the *TINs* module .
9. Select the *Open* command from the *File* menu.
10. Find and *Open* the file named *rockynff.tin*. By default it should be in the tutorial directory.

11.3 Prepare the TIN to Use With NFF

The watershed in *rockynff.tin* has previously been prepared for basin delineation. However, you must compute basin data in order to use the geometric parameters in conjunction with NFF.

1. Select *Compute Basin Data* from the *Drainage* menu.
2. Set model units to *feet* (since the original TIN data is in feet).
3. Set basin areas to *square miles*.

4. Set distances to *feet*.
5. Select *OK*.

In order to see the parameters which will be used with the NFF program you can turn them on for display.

1. Select the *Display Options* from the *Display* menu.
2. Select the *Drainage* tab.
3. Turn on the following toggles: *Show units*, *Basin areas*, *Basin slopes*.
4. Select *OK*.

Basin attributes are displayed at the centroid of the basin.

11.4 Running an NFF Simulation

The geometric data computed from the TIN has automatically been stored with the NFF data. You can now run a simulation, using the data derived from the TIN.

1. Select the *Hydrologic Modeling* module .
2. Select the *Select Basin* tool .
3. Select the basin icon or anywhere within the basin boundary.
4. Select the *Run Simulation* command from the *NFF* menu.

Demo users will need to select *NFF - Read Simulation*, and choose "rockynff.nff", then select *NFF - Run Simulation*, and Texas will be automatically loaded.

The NFF dialog appears. The instruction window at the bottom of the dialog indicates the steps that remain to complete the set up of an NFF simulation.

11.4.1 Choosing a State

WMS is linked to a database of state-by-state regression variables which are used in the NFF program to compute peak flows. This tutorial will focus on Texas, but if you are not running in demo mode you may wish to return at the end and test another state.

1. Select the *State* button.

2. Scroll through the states until you see Texas and then select it by clicking on it.
3. Select *OK*.

11.4.2 Defining Regions

The regions with defined regression equations are shown. Scroll through them, and notice the available urban equations. For this tutorial, we will assume that 70% of the study basin is located in Region 5 and 30% in Region 6. (Refer to region maps for your state in the NFF manual to determine if your basin overlaps more than one hydrologic region).

1. Select region 5 by clicking on it in the regions list window.
2. Select the button with a right arrow.
3. Now repeat the last two steps for region 6.

The variables required by each region's equations are displayed along with a defaulted U.S. Maximum Flood Region (the edit field in the upper right portion of the dialog), as shown on page 16 of the NFF manual.

11.4.3 Assigning Areas to Selected Regions

The total basin area is supplied by the TIN, but you must assign the amount overlapping each hydrologic region.

1. Select the *Assign % Watershed Which Overlaps Regions* button.
2. Enter 70 for the *percent area* for region 5.
3. Enter 30 for the *percent area* for region 6.
4. Select the *Compute Areas From Percents* button.
5. Select the *OK* button.

The area variables are updated with the computed values.

Highlight each of the selected regions and move the mouse over each variable's edit field. Examine the help window (you may need to relocate the *National Flood Frequency Regression Equations for Texas* dialog) to see the range of values used for each variable in defining the regression equations.

11.4.4 Defining Regression Variables

The slope shown on the TIN differs from that defaulted for region 5 because slopes computed for the TIN are in ft/ft, and slope data for the Texas equations is in ft/mile and has been converted by NFF.

Three of the four required variables are now determined. You may overwrite any variable values supplied by a TIN if you would like to test your own values. Original TIN values may be restored using the *Restore Values From TIN* button.

1. Select region 6 by clicking on it in the *Regions overlapped by watershed* text window.
2. Set the *Precipitation* (labeled with a P) variable for region 6 to be 15.

11.4.5 Computing Peak Discharges

You are now ready to compute flow data. Check the instructions window for any items overlooked.

1. Select the *Compute Peak Discharges* button.

Rural peak flows are displayed for the 2 yr - 500 yr storms, along with a maximum flood envelope computed using the maximum flood region, as defined in the edit field in the upper right. Predefined standard error is also shown, and is considered valid provided that variables were within specified range. If the regions have any equivalent years of data known for the site, computed peaks can be weighted with the known values. You will be prompted for weighting if this is possible.

11.4.6 Exporting Flow Data

Once flow data is computed it may be exported to a text file in the format shown in the window, along with pertinent information used in computing the peak flow values.

1. Select the *Export Flow Data* button.
2. Locate the directory, and define a name for the file.
3. Select the *Save* button.

The exported file can be viewed using any word processor.

11.4.7 Computing a Hydrograph

A dimensionless unit hydrograph can be used to define a hydrograph for any interval, based on the peak flow. This hydrograph will be based on the peak discharge and basin lag time.

1. Select the *Compute Peak Discharges* button.
2. Select a recurrence interval from the window where computed peak flows are reported (select the line of text containing the recurrence interval).
3. With the recurrence interval highlighted, select the *Compute Hydrograph* button.
4. Select the *Auto Lag Time* button.
5. Select the *Espey Rural Method* option from the drop down list for computing lag time.
6. Select *OK*.

The computed lag time is shown in the lag time edit field.

1. Select *OK*.
2. Select the *Done* button.

A hydrograph icon will appear in the *Graphics Window*. You can examine the hydrograph in more detail by:

1. Select the *Show Hydrograph Window* from the *Display* menu.
2. Select the hydrograph icon in the *Hydrologic Modeling Window*.

The hydrograph is displayed and may be manipulated using commands the *Hydrograph* menu.

When the hydrograph became selected the basin became unselected. Re-select the basin by clicking inside the basin using the *Select Basin* tool . If you want to see the computed peak flows again you must select the *Compute Peak Discharges* button.

11.4.8 Saving and Reading Simulations

You may wish to save this NFF simulation containing a state, selected regions, and defined variables. If you are running a demo version of WMS you should skip this step.

1. Select the *Save Simulation* command from the *NFF* menu.

The simulation may then be restored at a later date using the *Read Simulation* command in the *NFF* menu.

11.4.9 Using the National Urban Equations Defined in NFF

A national urban regression equation has been defined for use by any state, in addition to several equations particular to specific urban areas. Urban equations available for use are identified in the available equations window by the "\$" symbol for local, and "#" symbol for national. Local urban equations may not be used in conjunction with rural equations, consequentially rural equations disappear when a state urban region has been selected. However, the national equation is dependent on computed flows from the corresponding rural region, and therefore must be used in conjunction with the appropriate rural region.

To use the *National Urban Equation* based on the flows you have just computed:

1. If the basin is not selected and the *National Flood Frequency Regression Equations* is not up the select the basin using the *Select Basin* tool , and choose the *Run Simulation* command from the *NFF* menu.
2. Select the *#National Urban Equation* from the *regional regression equations* text window.
3. Click on the right arrow to move it to the *regions overlapped by watershed* window.
4. Enter the following values:
 - BDF = 8
 - IA = 45
 - SL = 42.03
 - RI2 = 1.8
 - ST = 7
5. Select the *Compute Peak Discharges* button.

Computed urban peak flows are shown along with previously computed rural peak flows. You may wish to read pages 8-9 in the *NFF* manual for further information regarding the National Urban Equation.

11.4.10 Using Individual State Urban Equations Defined in NFF

The National Urban Equation is common to all states, but some states have local urban equations as well (Texas is one of those states). To use urban equations defined for Texas you must unselect all of the regions defined in the *regions overlapped by watershed* text window. Available state urban equations will reappear, and you can select one for analysis. NOTE: The watershed defined by the TIN is not an urban watershed, but we will demonstrate how state-wide urban equations are used with this same data.

1. Highlight *Region 5* in the *regions overlapped by watershed* window.
2. Unselect it by pressing the button with the left arrow.
3. Repeat the last two steps for *Region 6* and *#National Urban Equation*.

Scroll down the *regional regression equations* text window and notice that urban equations have been defined for Houston, Austin, and Dallas.

1. Select *\$Dallas* as the urban region for analysis by selecting it in the *regional regression equations* text window and then selecting the right arrow. (Notice that all other regions disappear, since they cannot be used in conjunction with a statewide urban equation).
2. Enter 24.60 for A.
3. Enter 30.0 for UI.
4. Select the *Compute Peak Discharges* button.

Projected peak flows for the Dallas urban basin are shown.

11.5 Conclusions

This completes the NFF tutorial for computing rural discharges at ungaged sites. You may select "Done", or if you are not in demo mode you may choose another state for analysis.

NFF

Supplementary Exercise for the NFF Interface-Session 9

There are numerous different ways to continue experimenting with the NFF interface. Try using the following tins as if they are part of your state. You can also try using them in other states. Be sure to eliminate interior outlet points so that you only have a single watershed.

All files for this exercise are located in the *Supplimentary 9* directory (Except for files created by you).

1. **Spanish Fork Canyon**

- a) You can use the sf.tin file or if you delineated it on your own you will have a mysf.tin file.
- b) Define an NFF model and compute the hydrographs for the sub-basins using your own state's data.

2. **Jordanelle**

- a) You can use the jordanel.tin file or if you delineated it on your own you will have a myjord.tin file.
- b) Define an NFF model and compute the hydrographs for the sub-basins using your own state's data.



HEC-1

- Lumped parameter
- One of the most commonly used rainfall/runoff programs
- Options to use several different methods

What can HEC-1 be used for?

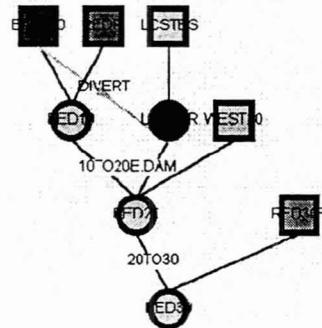
- Rainfall/runoff for “single” event storms
- Infiltration losses
- Creating unit hydrographs for a watershed
- Complete runoff hydrograph for a watershed
- Routing hydrographs downstream
- Combining hydrographs of different sub-basins

Limitations of HEC-1

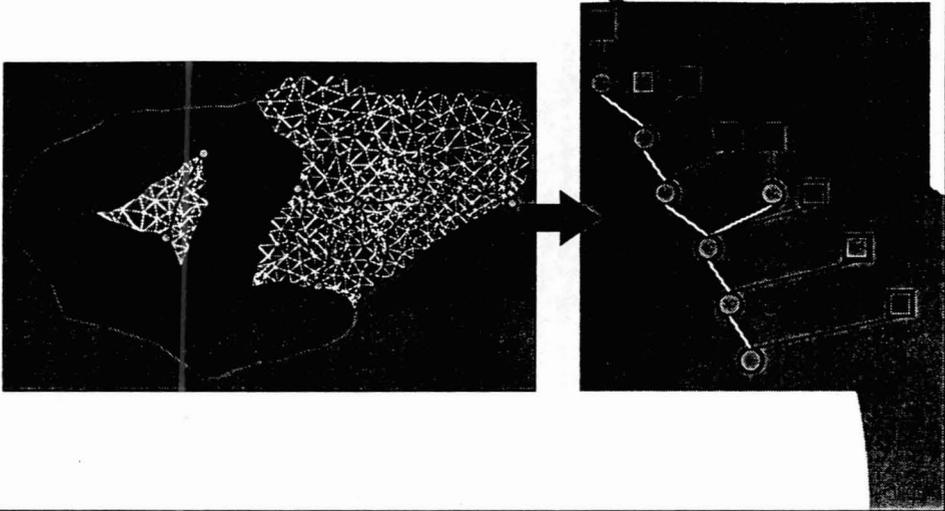
- Lumped parameter model
- Single event
- No stage computations
- Hydrologic NOT Hydraulic stream flow routing
- Level pool reservoir routing

HEC-1 Model Structure

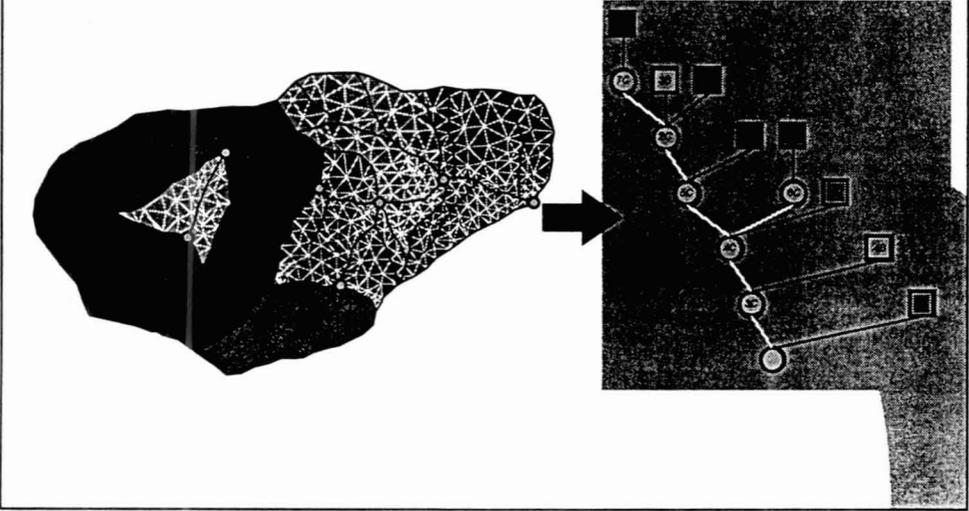
- Basins
- Outlets
- Reservoirs
- Diversions



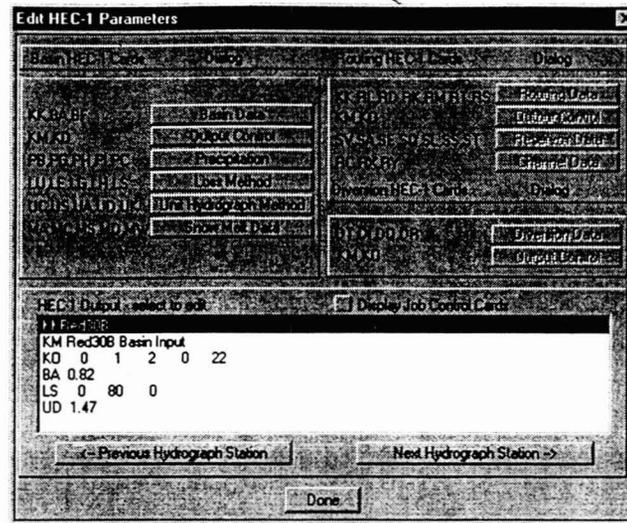
Topologic Model



HEC-1 Model Created by WMS



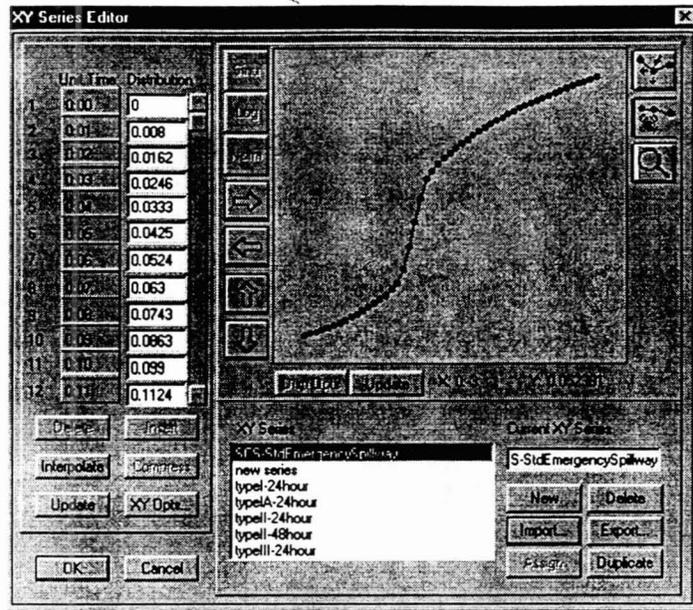
Assigning HEC-1 Parameters



Precipitation

- Hyetograph
 - Total rainfall
 - Time distribution of rainfall
- Spatial distribution
- Temporal distribution

XY Series Editor



Using Gages



Basin Parameters

- Precipitation (could apply to all)
- Base flow
- Losses
- Unit hydrograph method
- Snowmelt data

Available Loss Methods

- Initial-Uniform (LU)
- Exponential (LE)
- Green-Ampt (LG)
- Holtan (LH)
- SCS (LS)

SCS Method (CN)

- Initial abstractions
 - Soil moisture capacity in inches
 - $.2S$ where $S=1000-10CN/CN$ by default
- Curve Number (CN)
 - Soil group
 - Land use
 - Never account for development

Initial and Uniform Method

- Initial amount satisfied
 - .8-1.2 inches for sandy soils
 - .4-1.0 inches for clay soils
- Thereafter the uniform rate is used.
 - .1-.3 in./hr for sandy soils
 - .05-.15 for clay soils
- should be developed for your specific basin

Exponential Method

- Must be determined using optimization routines
- Therefore, used infrequently

Holtan Method

- Accounts for plant cover
- Accounts for some moisture capacity recharge

Green-Ampt Method

- Actual physics of soil percolation
- More sophisticated

Which Loss Method?

- SCS, Initial-Uniform, Green-Ampt preferred
 - Data availability
- Use Exponential only with calibrated data
- Use Holtan if plant growth is significant
- All methods work if given good data

Surface Runoff

- Unit Hydrograph Methods
- Distributed Methods

Unit Hydrographs of HEC-1

- Derived
- Synthetic
 - SCS
 - Clark
 - Snyder

SCS Method

- Lag time is the only required parameter
- Applicable only to moderately sloped watersheds
 - Should not use for flat or steep watersheds

Clark Method

- Time of concentration
 - Many different ways to determine
- Storage coefficient
- A user supplied or default time area curve

Snyder Method

- Time of concentration
- Peaking coefficient

WMS Computed Parameters

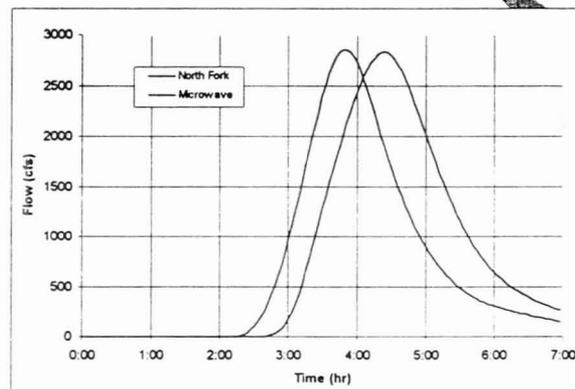
Basin Geometric Attributes				
Variable Names		Value		
Common	WMS	R/m	mi/km	Description
A	A	1.3e+008	4.713	Basin area (mi ² or km ²).
Lo	AOFD	1383.013	0.262	Average overland flow length.
S, So	BS	4.34e-002		Basin (overland) slope.
L	MFD	18183.695	3.446	Basin length along main channel from outlet to upstream boundary.
	MFDS	3.12e-002		Basin slope along main channel from outlet to upstream boundary.
Lca	CSD	8923.709	1.690	Length along main channel from outlet to point opposite centroid.
Sca	CSS	2.19e-002		Slope along main channel from outlet to point opposite centroid.
Lc	MFL	15752.705	2.983	Maximum flow (watercourse) length.
Sc	MSS	0.00e+000		Maximum flow (watercourse) average slope.
CN	CN	83		Current curve number (CN) (defined in Losses Dialog).
I	RTIMP	0.00e+000		Current percent impervious area (defined in Losses Dialog).

Done

Routing Stream Flow

- Travel time
- Attenuation
- Volume changes - channel losses

Effects of Routing on Hydrograph



HEC-1 Routing Methods

- Kinematic Wave
- Muskingum-Cunge
- Muskingum
- Straddler-Stagger and Tatum (Averaging)
- Modified Puls Storage Method
- Normal Depth Storage Method

Segmenting the Channel Reach

- Segment Length = Distance Flood Wave travels in one time step
- Subdivide the channel reach
- NSTPS = Travel time/Computational Time interval
- Travel time = Length of reach/Velocity of flood wave
- Velocity of flood wave = Channel velocity
* ~ 1.5

Diversions

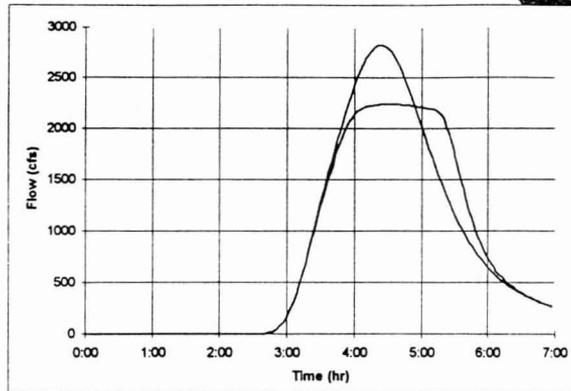
- May occur at any point in the watershed
- Must define relationship between flow in the channel and diverted flow.
- Diverted flow can be returned at any point downstream
- Diverted flow may also be routed to account for lag and attenuation

Reservoirs and Detention Ponds

- Effect on runoff hydrograph
 - attenuation
 - lag time
- Design of reservoirs and reservoir structures
- Safety
 - Dam breaching and breaking

Level Pool Routing

- Modified Puls



Requisite Input Parameters for HEC-1

- Storage
- Discharge

Reservoir Storage Options

- Table of Volumes
- Volumes and elevations
- Areas and elevations

Reservoir Discharge Options

- Discharge
- Discharge and elevations
- Spillways
- Low level outlets
- Pumps
- Overtopping (Dam Safety)

Tools for HEC-1 Analysis

- Reservoir creation
 - storage capacity curves (only with a TIN)
- Time area curve generation
 - approximates travel time with flow distance
- Thiessen polygons for gauges
- Lag time and time of concentration equations
- Curve Number computations
- Snow melt zone calculations
- NEXRAD (rain gauge) rainfall interpolation
- Phoenix Valley S-Graphs

How Does HEC-1 Work in WMS?

- 1 Delineate a watershed
- 2 Initialize an HEC-1 analysis
- 3 Define precipitation
- 4 Define basin parameters
- 5 Define routing parameters
- 6 Add reservoirs/diversions
- 7 Generate a hydrograph

HEC-1 Interface

Estimated Time: 30 minutes.

Once a watershed has been appropriately subdivided into smaller basins, you can use *WMS* to enter parameters for defining a complete HEC-1 input file. Geometric attributes such as areas, lengths, and slopes are computed automatically from the TIN geometry. Parameters such as loss rates, base flow, unit hydrograph method, and routing data are entered through a series of interactive dialog boxes. Once the parameters needed to define an HEC-1 model have been entered, an input file with the proper format for HEC-1 can be written automatically. There is no need to use an editor or even look at the file.

Since only parts of the HEC-1 input file are defined in this tutorial, you are encouraged to explore the different available options of each dialog, being sure to select the given method and values before exiting the dialog.

9.1 Objectives

In this tutorial you will load a TIN which has already been delineated into several different sub basins. You will read in a partially defined HEC-1 input file and finish defining all necessary parameters. Finally you will run HEC-1 and view the resulting hydrographs from within *WMS*. The steps covered include the following:

- Reading TIN, Map and HEC-1 data files.
- Defining gages, and precipitation for the watershed.

- Defining the SCS loss rate for all basins.
- Defining a hydrograph method and an equation to compute lag time for the basins.
- Defining the routing parameters from one outlet to the next.
- Computing geometric attributes.
- Computing curve numbers (CN) from land use and soil type coverages.
- Creating an HEC-1 input file.
- Running HEC-1.
- Reading the TAPE22 hydrograph results file and displaying the results.
- Computing time area curves.
- Defining elevation zone areas for a snow melt analysis.

9.2 Reading the Data Files

A super file will be used to read in the necessary data files. A super file simply contains the names of other files that may be components of your project, and works best if each component is in the same directory as the super file. You may edit the super file with any standard editor such as Notepad or DOS Edit if you find it necessary. On UNIX operating systems you will undoubtedly need to change the directories to match where *WMS* has been installed.

The TIN you will be reading is the same one used in the last tutorial, except that a stream network and five drainage basins have already been defined. Land use and soil type coverages have been defined and saved in a map a file. An HEC-1 input file will also be read as part of the input so that some of the basic input can be predefined. To read in the file, do the following:

1. If you are continuing from a previous tutorial select *New* from the *File* menu and confirm that all data should be deleted.
2. Select *Open* command from the *File* menu.
3. Find and *Open* the file named *rockyhec.sup*. By default it should be in the tutorial directory.

Once the files are read in you should see the Rocky TIN and two polygon coverages.

4. Select *Display Options* from the *Display* menu.
5. Turn the display of all entities off.
6. Select the *Drainage* tab.
7. Be sure the display of Outlets, Outlet names, Stream networks, Drainage basin boundaries and both gage types are all turned on.
8. Select the *OK* button. The display is updated and you should see the watershed divided into five sub basins.

You may want to move or turn off basin labels as described in previous tutorials.

9.3 Computing Geometric Attributes

Some of the geometric attributes are not saved as part of the TIN file, therefore be sure to compute them each time you wish to define an HEC-1 input file.

1. Select the *Compute Basin Data* command from the *Drainage* menu.
2. Specify feet for the model units, sq. miles for area, and feet for length and then select the *OK* button.

After the computation, the basin areas will be displayed at the centroid of the drainage basin. If you wish, you can turn these labels off from the drainage display options dialog.

Once computed, basin areas, stream lengths and stream slopes are automatically stored for use in defining the HEC-1 input file. The flow distances and basin slopes computed are used to compute lag times with the Tulsa equation.

9.4 Entering Gages

The gage method for rainfall will be used in each basin. To do this you must define the gages with their associated precipitation accumulation and distribution values. The table below will aid you in creating the rain gage network.

Table 7.1. Gage Information.

NAME	X-Coordinate	Y-Coordinate	Precipitation
gage1	10500	25000	1.38
gage2	18000	12000	1.41
gage3	35670	7530	1.36
gage4	35900	23320	1.32

These gages represent total storm stations, and therefore no temporal distributions are associated with them. After entering these four gages, you will be instructed to enter one gage that defines the temporal distribution for all four gages. All of the gages can be entered from within the Gages dialog as explained below:

1. Change to the *Hydrologic Modeling* module .
2. Select the *Gages* command from the *HEC-1* menu.
3. Select the *New* button.
4. Enter the name for the first gage in the name box.
5. Enter the values of the first gage from the above table.
6. Select the *New* button again and define the values for the second gage.

Continue this process until all four gages have been entered.

You must now enter one gage which defines the temporal distribution for all of the gages:

1. Select the *New* button.
2. Enter the name "gage5" in the name edit box.
3. Enter the x-y coordinates $x=29,000.0$ and $y=15,400.0$.
4. Click the *Storm Total Station* check box off.
5. Click the *Temporal Distribution Station* check box on.
6. Enter today's date in the appropriate edit boxes.
7. Select the *Define Series* button.
8. Click on the *Import* button found in the lower right-hand corner of the dialog box.
9. Locate and select the file called *scstabl5.ser*. By default it is located in the tutorials directory. A list of six defined rainfall distributions will appear in the xy series window.
10. Click on the *SCS-StandardEmergencySpillway* series to make it the active xy series. The plot window will show the distribution for this series.

The values corresponding to this standard distribution are automatically inserted into the text edit fields. You are free to scroll

through and examine the values. You are also free to change the distribution according to a pattern of interest to you.

11. Click on the *OK* button of both dialogs.

The storm total stations will be displayed along with the Thiessen network made up of the four storm total stations. The temporal distribution gage is displayed in a different color than the other gages, and is not included in the Thiessen network since it defines only a distribution and not a storm total. Later, as basin data are computed, weights corresponding to the Thiessen network will be assigned to each basin.

9.5 Defining Job Control Parameters

Each HEC-1 input file must have certain basic parameters defined. Job control data is entered in the following manner:

1. Select the *Job Control* command from the *HEC-1* menu. A dialog will appear with several different job control options.
2. Enter three text strings of your choice in the ID text edit fields. These text fields correspond to the identification cards at the top of the HEC-1 input file.
3. Set the computational time interval to 10.
4. Set the date fields to today's date.
5. Set the number of hydrograph ordinates to 75.
6. Click on the OK button to close the dialog.

9.6 Defining Basin Parameters

HEC-1 basin parameters that can be defined using *WMS* include precipitation, loss rates, unit hydrograph method, and base flow. The first set of parameters to enter are the loss rates:

1. Click on the *Select Drainage Basin* tool .
2. Choose the *Select All* command from the *Edit* menu.
3. Select the *Edit HEC-1 Parameters* command from the *HEC-1* menu. You may wish to reposition your windows so that the HEC tree and the *Edit HEC-1 Parameters* can be seen.

9.6.1 Loss Method

HEC-1 provides several different methods for performing rainfall loss computations. For this tutorial we will be using the SCS curve number method. The actual curve numbers will not be entered at this time, but will be computed from land use and soil type coverages later. However you must be sure to specify that you want to use the SCS method.

1. Select the *Loss Method* button.
2. Select the SCS curve number (LS) method.
3. Select *OK*.

9.6.2 Unit Hydrograph Method

Once the loss rate method has been set, you are ready to define the unit hydrograph method which will be used for computing the basin hydrographs. To define the unit hydrograph parameters, do the following:

1. Select the *Unit Hydrograph Method* button.

The dialog you will see provides you with the ability to choose from the different unit hydrograph methods available in HEC-1, and then define the appropriate value. The method you will be defining is the SCS dimensionless unit hydrograph method.

2. Click on the *SCS dimensionless* method option.
3. Click on the *Compute Parameters – Basin Data* button.
4. Set the computation method to the *Tulsa* (rural) method.

The lag time used in the SCS dimensionless method is a function of basin slope, and lengths of flow within the watershed. Since these parameters are computed by WMS when computing basin data, the lag times can also be computed. Refer to the users manual to see a full description of the Tulsa district equation.

5. Repeat step 4 by selecting each basin listed in the *Basin* text window and assigning the *Tulsa* (rural) method to it.
6. Click on the OK button of both dialogs.

9.6.3 Precipitation

The last basin parameter you will enter here is the precipitation. To enter the precipitation parameters, do the following:

1. Select the *Precipitation* button.
2. Choose the gage option.
3. Select the OK button.
4. Select *Done* in the *Edit HEC-1 Parameters* dialog.

Weights are assigned automatically using the Thiessen method when computing basin data. In order to make this happen you must compute the basin data once again.

5. Select the *TINs* module icon .
6. Select the *Compute Basin Data* command from the *Drainage* menu.
7. Specify feet for the model units, sq. miles for area, and feet for length and then select the OK button.

9.7 Defining Routing Data

Routing parameters are defined at outlet points, and they define the reach from a given outlet to the next downstream outlet. Stream lengths and slopes are automatically computed and associated with the appropriate outlet when computing drainage data from the *Drainage* menu. For the example model you are working with, you will be defining Muskingum routing for the three outlet points in the stream (no routing needs to be defined at the outlet representing the watershed outlet). To define this routing method, do the following:

1. Change back to the *Hydrologic Modeling* module .
2. Select the *Select Outlets* tool .
3. Click one of the three “upstream” outlet points (not the bottom most outlet). The three “upstream” outlets should have routing data defined.
4. Select the *Edit HEC-1 Parameters* command in the *HEC-1* menu.
5. Choose the *Routing Data* button.
6. Select the *Muskingum* routing option (be sure it is the Muskingum - RM option and not the Muskingum Cunge - RD option).
7. Enter a value of .2 for *X*.
8. Determine the routing time for this reach by doing the following:

9. Find the length of the channel (you should be able to see a value for *L* such as 16992.7 in the Muskingum-Cunge edit field - it will be dimmed but you should be able to read it).
10. Estimate the channel velocity (use 3.0 ft/sec).
11. Divide the length by the estimated velocity ($16992.7/3 = 5664.2$ sec.)
12. Divide by 3600 sec/hr to get travel time in hours. Enter this value for *AMSKK*.
13. Finally determine how many time steps it will take to route the flow through this channel segment and enter the value in the *NSTPS* field. This can be determined by dividing the travel time (in minutes) by the computational time step entered in the *Job Control* dialog (10 minutes).
14. Select the *OK* button.
15. Repeat steps 2-8 for the other two outlets where routing is to be defined.
16. Select the *Done* button in the *Edit HEC-1 Parameters* dialog.

9.8 Computing Composite Curve Numbers

Besides computing geometric data, composite curve numbers can be determined using the land use and soil coverages read in with the model.

1. Select the *Composite Runoff Coeff/CN...* command from the *Calculators* menu.
2. Select *Compute Using Coverage* for Soil Type.
3. Select *Compute Using Coverage* for Land Use.
4. Select *OK* to compute composite CN's.

So that you can be satisfied that lag times and CN numbers have been computed, double click on any one of the basin icons in the Tree window. This brings up the Edit HEC-1 Parameters dialog. You will see the composite curve number of the selected basin following the LS record (it is the second number) shown in the HEC-1 cards window. The lag time computed can be seen following the UD card. Select the other basins and note the differences in these two parameters.

1. Select the *TINs* module icon .
2. Select the *Display Options* option from the *TINs* menu.

3. Turn on the soil group, land use, soil group legend, and land use legend options.
4. Select *OK*.

You should see the different combinations of land use/soil group combinations that have been assigned to the TIN.

9.9 Running and Saving HEC-1 Files

You have now completed the definition of the HEC-1 model and are ready to run the analysis. HEC-1 is a stand-alone program, but can be launched from within *WMS* using the following steps:

1. Change to the *Hydrologic Modeling* module .
2. Select the *Check Model* option from the *HEC-1* menu.

This will perform a check on the data and try to determine problems that should be addressed before HEC-1 can be successfully run. Just because the model checker does not find any errors, does not insure that HEC-1 will compute the correct result, but it should help.

3. Select *DONE*.
4. Select the *Run HEC-1* option from the *HEC-1* menu.

HEC-1 needs the names of three files: 1) the input file (which will be saved when running HEC-1). 2) an ASCII output file outlining the processes used and the final results. 3) a TAPE22 solution file containing hydrograph information which can be read back into *WMS* for display.

5. Enter *rocky* in the prefix for all files edit field and click on the *Update All Prefixes* button.
6. Click *OK* to start HEC-1.

The *View File* command in the *File* menu can be used to examine the ASCII output file should problems occur. On PC's the view file command brings the file up within the Windows Notepad program, whereas on UNIX operating systems you can specify an editor (vi by default) to bring the file up in.

9.10 Displaying Hydrograph Results

The TAPE22 file generated by HEC-1, and stored in the third file specified when running HEC-1 from within *WMS* contains hydrographs for each basin

of the watershed, for the routed hydrographs from outlet points, and for the combination of basin and routed hydrographs at each outlet. You can read this file and display the hydrographs in the hydrograph window. To read the hydrograph file, do the following:

1. Select the *Show Hydrograph Window* from the *Display* menu.

If you are running on a UNIX operating system your hydrographs have already been read in and you can skip the next two steps. If you are running Microsoft Windows you will need to follow the next two steps to read in the hydrographs.

1. Select the *Open* command from the *Hydrographs* menu.
2. Find and *Open* the solution file used to store the resulting hydrographs. If you did not actually run HEC-1, or are having difficulties you can read in tape22.bak.

After you have successfully read the hydrograph file, you should see several small hydrographs displayed in the upper right corner of each outlet and basin in the *Hydrologic Modeling* window. You can examine the hydrographs in greater detail by selecting them for display in the hydrograph window. Two hydrographs for some of the outlets will be displayed representing the hydrograph before and after channel routing. To view the hydrographs in the hydrograph window, do the following:

1. Select one of the hydrographs by clicking on the miniature.
2. Hold the SHIFT key down while selecting more hydrographs. All currently selected hydrographs should be displayed in the hydrograph window.
3. Select the *Display Options* command from the *Display* menu.
4. Select the *Plot Options* button.
5. Turn on the *Display X grid* toggle.
6. Turn on the *Display Y grid* toggle.
7. Set any other parameters you wish to experiment with.
8. Click on the *OK* button of both dialogs.

9.11 Conclusion

This concludes the tutorial on defining HEC-1 files and displaying hydrographs. The concepts learned include the following:

- Entering job control parameters.
- Defining basin parameters such as loss rates, precipitation, and hydrograph methodology.
- Defining routing parameters.
- Creating gages, and their associated Thiessen networks.
- Writing HEC-1 input files.
- Reading hydrograph results and manipulating their display.

Creating Topologic Models For HEC-1

Estimated Time: 40 minutes.

This chapter, if you are familiar with HEC-1, will help you understand how *WMS* compares to and how it differs from traditional HEC-1 modeling. It is useful for new users of HEC-1 as well, since it will help you become familiar with what types of structures can be modeled with HEC-1, and how they are set up in *WMS*.

The example problem illustrated here is the Red River watershed. It is chosen since it illustrates many of the basic capabilities available in HEC-1. Furthermore, since it is the first example given in the HEC-1 User's Manual, it is a good source for illustrating how traditional HEC-1 models are created in *WMS*. This tutorial will not lead you through every step to completely set up the model. Instead you will be shown how to create and enter data for each type of object (i.e. basins, outlets, reservoirs, diversions) and then data for other similar objects will be provided so that you can repeat the steps. A completed model of the Red River watershed is provided in the tutorial directory. If you don't wish to enter all of the necessary data, or if you want to skim through this section and look at the results, you can read this file in using the *Open HEC-1 File* command in the *HEC-1* menu.

The Red River watershed demonstrates the following capabilities available in HEC-1:

- Rainfall data using gages.
- Calculating hydrographs using base flow, losses, and unit hydrograph methods.

- Flood hydrograph routing by the channel storage method.
- Reservoir routing using both spillway and low-level outlet options.
- Diversions.
- Input of time series data using time increments different than the computational time step.

18.1 The Topologic Tree Representation

The schematic shown in Figure 18-1 illustrates how different components in the Red River watershed are modeled using HEC-1. Figure 18-2 shows a completed tree diagram for the same watershed created using *WMS*. A few important differences need to be noted.

- **Names** for all hydrograph computation units (outlets, basins, etc.) must be unique. In the traditional HEC-1 diagram the name LOSTBR, EAST10, and RED30 are all used twice. In *WMS* the names must be unique so that resulting hydrographs read from a TAPE22 file can be associated with the proper hydrograph station. For this reason the basins LOSTBR and RED30 were changed to LOSTBS and RED30B.
- **Diversions** may exist from basins to outlets or from outlets to outlets. In the traditional HEC-1 schematic the EAST10 outlet point exists solely to provide a point from which a diversion can be defined. In *WMS* the diversion is defined directly from the EAST10 basin so there is no need to have the extra EAST10 outlet point. Diverted flow will be taken from the EAST10 basin before being combined with the REDRI basin at the RED10 outlet point.
- **Reservoirs** are defined with separate icons (triangles) in the traditional HEC-1 schematic, whereas in *WMS* they are defined at outlet points. There are actually a couple different ways the same problem can be set up using reservoirs in *WMS*. Figure 18-3 and Figure 18-4 illustrate these alternatives.

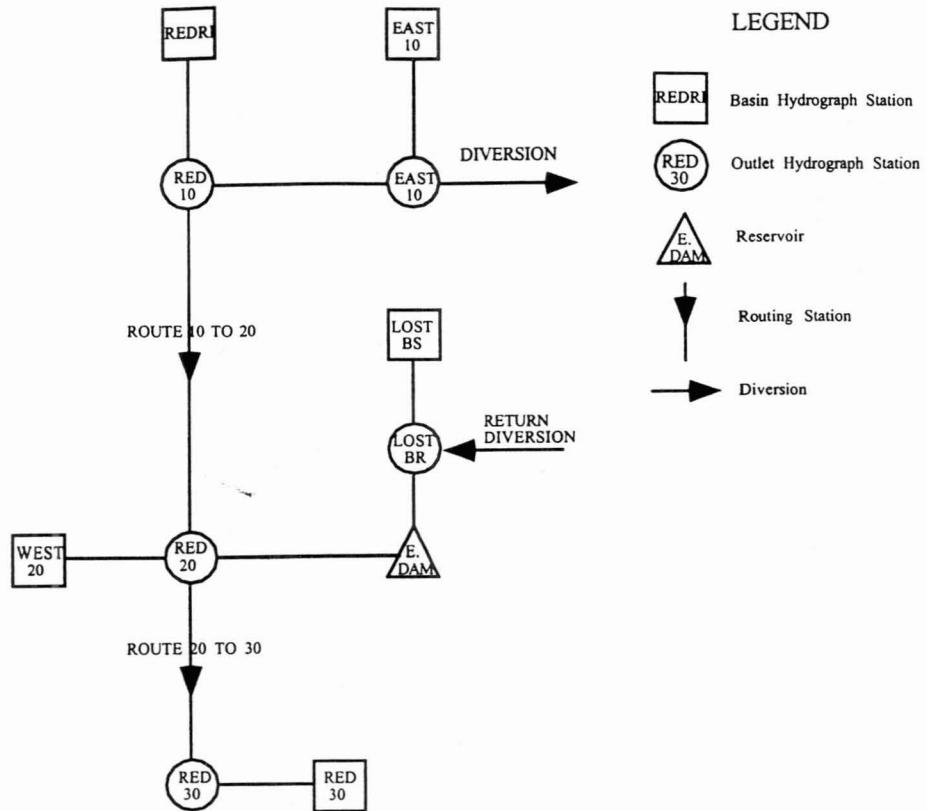


Figure 18-1 Traditional HEC-1 Tree Diagram For Red River Watershed.

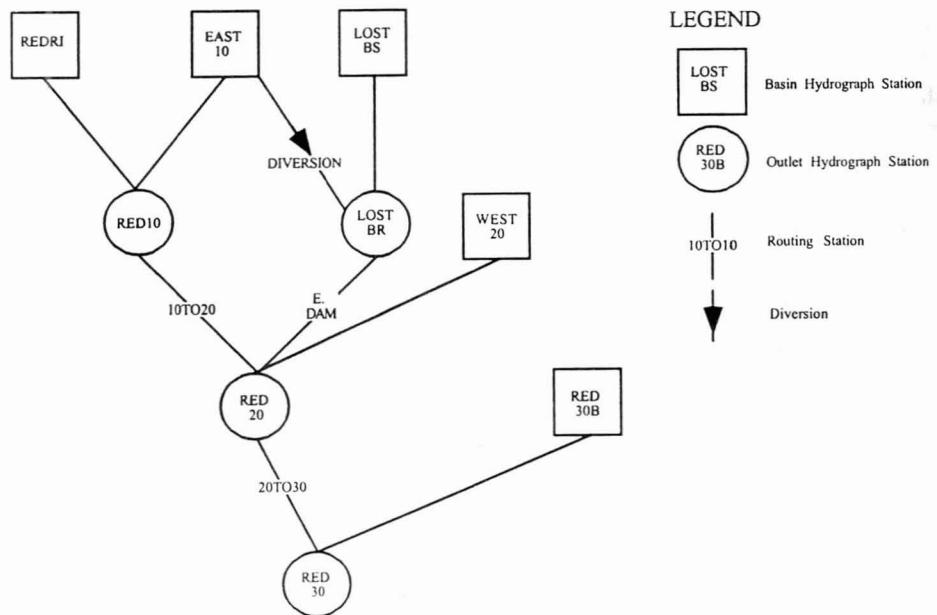


Figure 18-2 HEC-1 Tree Diagram of Red River Watershed Created by WMS.

Figure 18-3 shows an alternate tree created using WMS. The difference between this diagram and the one in Figure 18-2 is that E.DAM has been created as a reservoir rather than routing from LOSTBR to RED20 using the reservoir storage routing option.

Figure 18-4 puts the E.DAM reservoir at RED20, since there is no need to have the outlet at LOSTBR. The reservoir can be placed at the RED20 outlet. The diversion is routed from EAST10 to RED20, and the basin LOSTBS is connected directly to RED20.

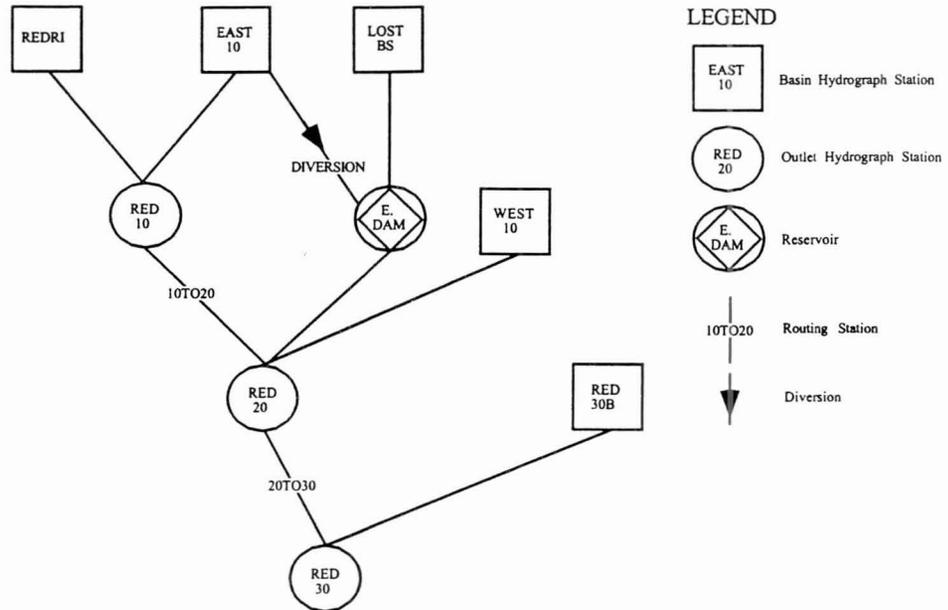


Figure 18-3 Reservoir Used in Place of Reservoir Storage Routing.

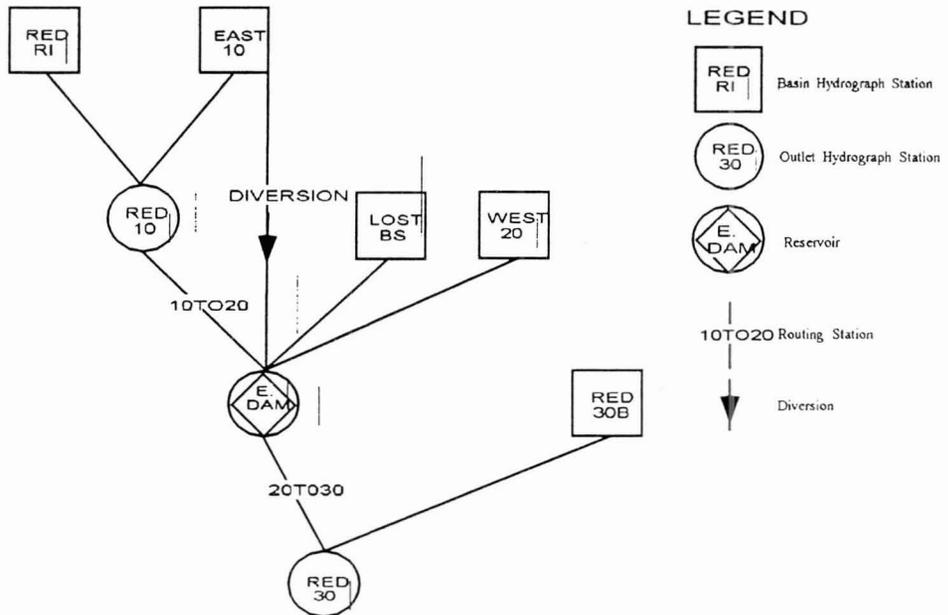


Figure 18-4 LOSTBR Outlet Eliminated with E.DAM Reservoir at RED20.

The HEC-1 input file generated from all three trees will be identical. They simply illustrate different ways in which reservoirs, and reservoir routing from outlets can be done. The tree defined in the remaining sections of this tutorial will correspond to the one in Figure 18-2.

18.2 Creating the Tree

If you are running the professional version switch to the Tree Module . If you are running the basic version you will be in the Tree Module already, unless you have turned the Demo on. If you are running the Demo turn it off now. The following commands can be used to create the topological tree for the Red River watershed as shown in Figure 18-2. All commands are in the *Tree* menu.

1. Select the *Add Outlet* command. The bottom outlet will be created and then selected. Each time you create a new outlet it becomes the selected outlet.
2. Select the *Add Basin* command. Don't worry that the names don't match those in Figure 18-2, you can change them later.
3. Select the *Add Outlet* command.
4. Select the *Add Basin* command.
5. Select the *Add Outlet* command.
6. Select the *Add Basin* command.
7. Click on the Outlet point named 2C to select it.
8. Select the *Add Outlet* command.
9. Select the *Add Basin* command.
10. Select the *Add Basin* command.

Except for the names, your tree should now look like the one in Figure 18-2. If it does not you can start over by selecting the *New* command in the *File* menu, or you can use a combination of the *Delete* and *Add* commands in the *Tree* menu to correct your tree.

18.3 Changing Hydrograph Station Names

WMS requires that all hydrograph stations have a unique six character identification name. In order to help avoid this problem default names are provided whenever an outlet, basin, diversion, or reservoir is created (i.e. 1C, 2C, 1B, 2B, etc.). You can change the name of these stations to make them more identifiable, however you must remember these two limitations:

- Names can not be longer than 6 characters.
- Names must be unique.

To change the outlet names do the following:

1. Click on the outlet point 1C and select the *Edit HEC-1 Parameters* command from the *HEC-1* menu, or double-click on the outlet point (this automatically brings up the *Edit HEC-1 Parameters* dialog).

The Edit HEC-1 Parameters dialog contains several buttons which can be used to define/edit data for hydrograph stations. Only the buttons which pertain to the selected hydrograph station are active, all others are dimmed. The text window below the group of buttons shows the actual HEC-1 cards as they are currently defined for the selected object(s). You can define/edit data by either: 1) Clicking on the appropriate button, or 2) Clicking on the appropriate HEC-1 card in the text-window.

1. Select the *Routing Data* button, or click on the KK card in the text-window.
2. Change the combining name from 1C to RED30 and click on the *OK* button. There is no routing from this outlet so you don't need to worry about changing the routing name.
3. Select the next upstream outlet 2C. You may wish to reposition your *Hydrologic Modeling Window* and/or the *Edit HEC-1 Parameters* dialog so that you can see both the tree and the dialog. You don't need to quit the *Edit HEC-1 Parameters* dialog, data for the next selected outlet point will now be displayed.
4. Select the *Routing Data* button, or click on the KK card in the text-window.
5. Change the combining name from 2C to RED20. There is routing from this outlet so change the routing name from 2R to 20TO30.
6. Repeat the last three steps changing 3C to LOSTBR, 3R to E.DAM, and 4C to RED10, 4R to 10TO20.

To change Basin names do the following:

1. Click on the basin 1B. If you accidentally closed the *Edit HEC-1 Parameters* dialog then double-click on 1B to bring it back up.
2. Select the *Basin Data* button, or click on the KK card in the text-window.
3. Change the basin name from 1B to RED30B and click on the *OK* button.
4. Click on basin 2B.

5. Select the *Basin Data* button, or click on the KK card in the text-window.
 6. Change the basin name from 2B to WEST20 and click on the *OK* button.
 7. Repeat the last three steps changing 3B to LOSTBS, 4B to EAST10, and 5B to REDRI.
1. Select the *Done* button.

18.4 Creating the Tree with Feature Objects

In more recent versions of *WMS*, the hydrologic modeling tree can be created directly from feature arcs (streams), and polygons (basins) (see chapter 4). This means that a tree can be created even without a map from which actual stream networks and watershed boundaries are digitized. In such cases the feature objects represent a "picture" or "drawing" of the watershed and are not used to calculate areas and lengths. However, the drawing itself may be much more instructive to clients and reviewers than just the modeling tree. If you want you can read in a map file that contains the same watershed definition you have just created by following these three steps:

1. Select the *New* command from the *File* menu and confirm that you want delete all data.
2. Select the *Open* command from the *File* menu.
3. Find and open the file named *redri.map*.

If you choose to do this your model will replace (but be equivalent to) the one you have just defined and should look like Figure 18-5.

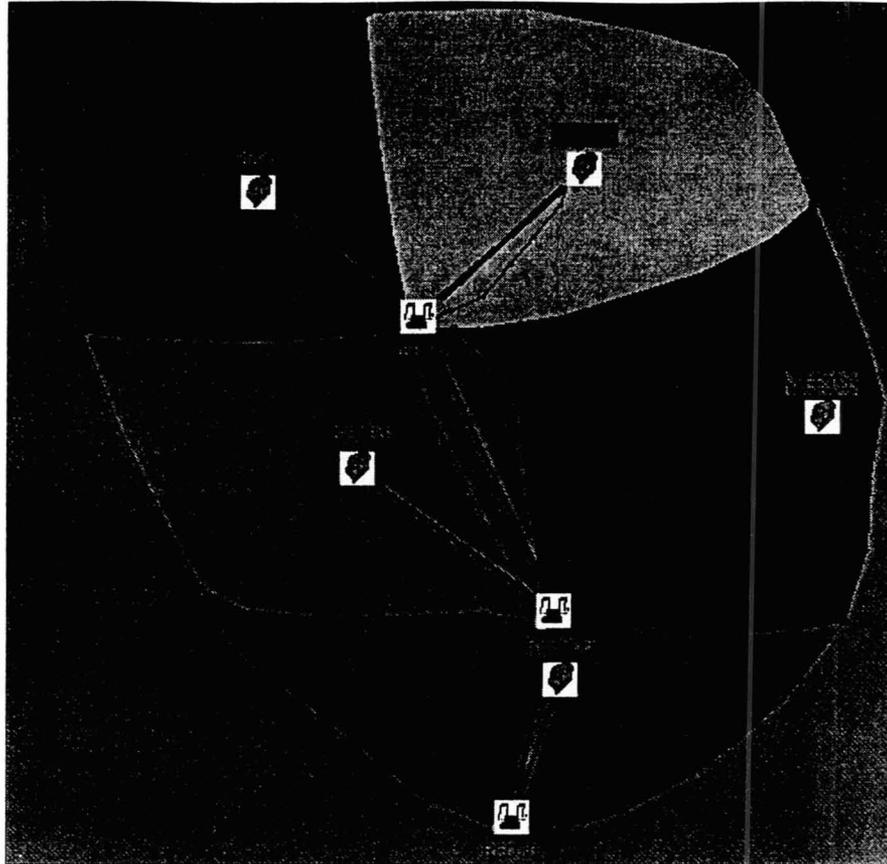


Figure 18-1 Red River Watershed Tree Defined from Feature Objects.

Diversions cannot be saved with feature objects (TINs, or DEMs either) so after creating a watershed using feature objects diversions still need to be entered using the commands in the *Tree* menu of the *Hydrologic Modeling Module*.

1. Select the *Hydrologic Modeling* module icon .
2. Select the *Select Basin* icon tool .
3. Click on the basin named EAST10 to select it.
4. Select the *Add Diversion* command from the *Tree* menu.
5. Click on the Outlet point named E.DAM to select it.
6. Select the *Retrieve Diversion* command from the *Tree* menu.

To change the diversion name do the following:

1. Click on the diversion D1.
2. Select the *Edit HEC-1 Parameters* command from the *HEC-1* menu.
3. Select the *Diversion Data* button.

4. Change the name from D1 to DIVERT.
5. While in this dialog change the outflow hydrograph name to EASTDV and the inflow hydrograph name to LOSTDV. These are the names used on the KK records where the diversion is created (DT record) and retrieved (DR record). Click on the OK button.

18.5 Job Control Data

Job control data includes computational starting time and data, interval, and length, general output specification and other custom features of HEC-1. To modify this data do the following:

1. Select the *Job Control* command from the *HEC-1* menu.
2. Change the day to 12.
3. Change the month to 6.
4. Change the year to 68.
5. Change the computational time interval to 15.
6. Change the beginning time to 715.
7. Change the number of hydrograph ordinates to 58. The total length of the simulation will be 58×15 min or 14 hours and 30 minutes.
8. If you wish, you can change the ID cards.
9. Select the *OK* button.

18.6 Defining Gages

The Red River watershed model uses 5 total storm stations and one recording station to define the precipitation for the simulation. This is done in *WMS* with the following steps:

1. Select the *Gages* command from the *HEC-1* menu.
2. Select the *New* button.
3. Turn the *Define rain gage location* toggle off.
4. Change the Name to 60.
5. Define the precipitation to be 4.68 (inches).

- Repeat the last three steps defining gages 61,62,63, and 64 using the data in Table 2.1 to define the precipitation.

Name	Precipitation
60	4.68
61	4.65
62	4.85
63	4.90
64	5.10

At least one recording station must be defined to set up the temporal distribution data for the gages. For the Red River watershed a single recording station is defined and then used for all storm total stations. Set this gage up with the following steps:

- Select the *New* button.
- Change the Name to 400.
- Toggle off the *Storm Total Station* button, and toggle on the *Temporal distribution station* button.
- Click on the *Define Series* button.

This will bring up the XY Series editor dialog. WMS uses this dialog to define several different series of data. Rainfall distributions are one type of series data. For additional information see the chapter in the reference manual on using the XY series editor.

By default the rainfall distribution values are considered to be cumulative. The Red River model however uses incremental. This model also uses a time step of 60 minutes rather than the default of 15. To change these two defaults to match the Red River model use the following steps.

- Select the *XY Opts* button.
- Toggle on the *Unit Time* box.
- Change the *beginning time* to 435 minutes (This corresponds to 7 hours and 15 minutes and will be converted appropriately when exiting the dialog).
- Change the *increment* to 60.
- Select the *Delta* button in the *Distribution* box.
- Select the *OK* button.

7. Starting with time 7:15 in edit field 1 enter the following ten incremental values: .04, .35, .01, .03, .73, .21, .02, .01, .03, .01.
8. Change the name of the *Current XY Series* to "Rainfall Distribution".
9. Select the *OK* button of both dialogs.

18.7 Defining Basin Parameters

HEC-1 requires that all basins have an appropriate area, precipitation, loss method, and unit hydrograph method defined. Just about any combination of these methods will work. This tutorial will describe how these parameters can be defined for one basin and then a table of values will be provided so that the parameters can be defined for the remaining basins.

Since the base flow parameters are identical for all basins this will be defined first.

1. Click on the basin named RED30B.
2. Select the *Edit HEC-1 Parameters* command from the *HEC-1* menu. Once again you may wish to reposition windows.
3. Select the *Select All* command from the *Edit* menu. All basins will be selected and their data displayed in the text window.
4. Select the *Basin Data* button. Notice that the name and area fields are dimmed since you wouldn't want to change these values with all basins selected.
5. Change the *STRTQ* value to be 10.
6. Change the *QRCSN* value to be -.25.
7. Change the *RTIOR* value to be 1.2.
8. Select the *OK* button.
9. Click on the RED30B basin once again so that it is the only one selected.
10. Select the *Basin Data* button or click on the BA card in the text window.
11. Change the *Area* to .19 and select the *OK* button.
12. Select the *Precipitation* button.
13. Select the *Gage* button.

14. Select the *PT Gage Weights* (These are weights for total storm stations).
15. In the first two *Gage Name* fields enter 64 and 63, and in the first two *Weight* fields enter .65 and .35.
16. Select the *OK* button.
17. Select the *PR Gage Weights* (These are weights for distribution station).
18. In the first *Gage Name* field enter 400, and in the first *Weight* field enter 1.0.
19. Select the *OK* buttons for both the *Weight* and *Precipitation* dialogs.
20. Select the *Loss Method* button or click on the LS card in the text window.
21. The SCS method is on by default (for some of the other basins you may need to change the method), but change the curve number (CRVNBR) field to 79.
22. Select the *OK* button.
23. Select the *Unit Hydrograph Method* button or click on the UD card.
24. Once again the SCS method is on by default, but you need to change the lag time (TLAG) to 1.04.
25. Select the *OK* button.

Basin parameters for RED30B should now be defined. You can now select the other basins one at a time and repeat the above steps for defining area, precipitation and weights, loss method, and unit hydrograph method using the data given in Table 2.2. For some basins you will need to change the Loss and Unit Hydrograph Methods before the appropriate parameters can be defined. The HEC-1 card identification (LS, LE, etc. is always part of the method button so you can identify which method to choose from the HEC-1 cards in Table 2.2. Where there may be some confusion parameter names are given in the table. If only one parameter exists, no name is given and if a parameter is left out of the table just leave its value at zero.

Table 2.2. Basin Parameters for the Red River Watershed.

Name	Area	Gage	Weights	Loss	Parameters	Unit	Params
RED30B	.19	63	.35	LS	CRVNBR=79	UD	1.04
		64	.65				
		400	1.0				
WEST20	.80	63	.60	LH	FC=.04	UD	.94
		64	.40		GIA=.4		
		400	1.0		SAI=.3		
					BEXP=1.4		
LOSTBS	.36	62	.5	LU	STRTL=.3	UC	TC=.8
		63	.5		CNSTL=.04		R=1.2
		400	1.0				
EAST10	.66	61	.6	LE	STRKR=.6	US	TP=1.3
		62	.3		DLTKR=1.0		CP=.8
		63	.1		RTIOL=1.0		
		400	1.0		ERAIN=0.0		
REDRI	.82	60	.75	LS	CRVNBR=80	UD	1.47
		61	.25				
		400	1.0				

18.8 Defining Routing Parameters

Routing parameters are defined in a similar fashion as basins. For the Red River model channel routing needs to be defined from RED10 to RED20 (10TO20) and from RED20 to RED30 (20TO30). This can be done with the following steps:

1. If the *Edit HEC-1 Parameters* is still active select the outlet E.DAM. If you have closed it already select E.DAM and then select the *Edit HEC-1 Parameters* command from the *HEC-1* menu.
2. Select the *Routing Data* button or click on the RN card in the text window.
3. Select the *Storage (RS)* routing option.
4. Set the nsteps to 1, the type to FLOW and RSVIC to -1.0.
5. Leave the channel button selected and click on the *Define* button.
6. Select the *Modified Puls* option and click on the *Define Volume* button. The *XY Series* editor can now be used to define the volume table.
7. Change the *Current XY Series Name* to "RED20 Volume".
8. Starting with the first table field enter the following values: 0, 17, 42, 67, 100, 184, 274, 386, 620.
9. Click on the *OK* button.

10. Click on the *Define Outflow* button.
11. Select the *New* button.
12. Change the *Current XY Series Name* to "RED20 Outflow".
13. Starting with the first table field enter the following values: 0, 500, 1000, 1500, 2000, 3000, 4000, 5000, 7000
14. Click on the *OK* button of all three dialogs.
15. Select the outlet RED10.
16. Select the *Routing Data* button or click on the RN card in the text window.
17. Select the *Storage* routing option.
18. Set the steps to 1, the type to FLOW and RSVIC to -1.0.
19. Leave the channel button selected and click on the *Define* button.
20. Select the *Modified Puls* option and click on the *Define Volume* button. The *XY Series* editor can now be used to define the volume table.
21. Select the *New* button.
22. Change the *Current XY Series Name* to "RED10 Volume".
23. Starting with the first table field enter the following values: 0, 18, 36, 54, 84, 110, 138, 174, 228, 444.
24. Click on the *OK* button.
25. Click on the *Define Outflow* button.
26. Select the *New* button.
27. Change the *Current XY Series Name* to "RED10 Outflow".
28. Starting with the first table field enter the following values: 0, 500, 1000, 1500, 2150, 2600, 3000, 3450, 4000, 6000.
29. Click on the *OK* button of all three dialogs.

18.9 Defining Diversion Data

Both an inflow and outflow table must be defined to set up the diversion data. This is done by following the steps below:

1. If the *Edit HEC-1 Parameters* is still active select the diversion arrow named DIVERT. If you have closed it already, select DIVERT and then select the *Edit HEC-1 Parameters* command from the *HEC-1* menu.
2. Select the *Diversion Data* button or click on the DR card in the text window.
3. Click on the *Define DI* button. The *XY Series* editor can now be used to define the inflow table.
4. Change the *Current XY Series Name* to "Divert DI".
5. Starting with the first table field enter the following values: 0, 100, 300, 600, 900.
6. Click on the *OK* button.
7. Select the *Define DQ* button. The *XY Series* editor can now be used to define the outflow table.
8. Select the *New* button.
9. Change the *Current XY Series Name* to "Divert DQ".
10. Starting with the first table field enter the following values: 0, 25, 100, 180, 270.
11. Click on the *OK* button of both dialogs.

18.10 Defining a Reservoir

As seen in Figure 18-2 to Figure 18-4 a reservoir can be defined in a couple of different ways: 1) by using the storage-reservoir routing method, or 2) by explicitly creating a reservoir and defining the storage parameters. For the model in this tutorial we have chosen to represent it using the second alternative.

1. If the *Edit HEC-1 Parameters* is still active select the outlet E.DAM. If you have closed it already select E.DAM and then select the *Edit HEC-1 Parameters* command from the *HEC-1* menu.
2. Select the *Reservoir Data* button.

3. Select the *Storage (RS)* routing option.
4. Set the *nsteps* to 1, the Initial condition type to *ELEV* and *RSVRIC* to 851.2.
5. Select the *reservoir* button and click on the *Define* button.
6. Select the *known volume* option.
7. Select the *SV* toggle and click on the *Define* button. The *XY Series* editor can now be used to define the volume table.
8. Select the *New* button.
9. Change the *Current XY Series Name* to "E.DAM Volume".
10. Starting with the first table field enter the following values: 21, 100, 205, 325, 955.
11. Click on the *OK* button.
12. Select the *SE* toggle and click on the *Define* button. The *XY Series* editor can now be used to define the volume table.
13. Select the *New* button.
14. Change the *Current XY Series Name* to "E.DAM Elevation".
15. Starting with the first table field enter the following values: 850, 851.5, 853.3, 856.5, 858.
16. Click on the *OK* button.
17. Select the *Compute Weir Spillway* button.
18. Select the *SL* toggle and define the parameters as follows: *ELEVL*=851.2, *CAREA*=12, *COQL*=.6, *EXPL*=.5.
19. Select the *SS* toggle and define the parameters as follows: *CREL*=856, *SPWID*=60, *COQW*=2.7, *EXPW*=1.5.
20. Click on the *OK* button for both dialogs.
21. Click on the *Done* button of the *Edit HEC-1 Parameters* dialog.

18.11 Running and Saving HEC-1 Files

You have now completed the definition of the Red River watershed model and are ready to run the analysis (If you haven't done all the work outlined in the

previous section you can read in the already defined model redri.hc1). HEC-1 is a stand-alone program, but can be launched from within *WMS* using the following steps:

1. Select the *Check Model* option from the *HEC-1* menu.

This will perform a check on the data and try to determine problems that should be addressed before HEC-1 can be successfully run. If you followed the steps in the preceding sections correctly there should be no problems. If you discover some potential problems you should correct them before moving on.

Just because the model checker does not find any errors, does not insure that HEC-1 will compute the correct result, but it should help.

2. Select *DONE*.
3. Select the *Run HEC-1* option from the *HEC-1* menu.

HEC-1 needs the names of three files: 1) the input file. 2) an ASCII output file outlining the processes used and the final results. 3) a TAPE22 solution file containing hydrograph information which can be read back into *WMS* for display.

4. Enter *redriv* in the prefix for all files edit field and select the *Update all prefixes* button.
5. Select *OK*.

When running HEC-1 the input file is automatically saved before HEC-1 is launched.

The *View File* command in the *File* menu can be used to examine the ASCII output file should problems occur. On PC's the view file command brings up a simple view only program, whereas on UNIX operating systems you can specify an editor (*vi* by default) to bring the file up in.

18.12 Displaying Hydrograph Results

The TAPE22 file generated by HEC-1, and stored in the third file specified when running HEC-1 from within *WMS* contains hydrographs for each basin of the watershed, for the routed hydrographs from outlet points, and for the combination of basin and routed hydrographs at each outlet. You can read this file and display the hydrographs in the hydrograph window. To read the hydrograph file, do the following:

1. Select the *Show Hydrograph Window* from the *Display* menu (If the hydrograph window is already mapped you won't need to use this command).

2. Select the *Open* command from the *Hydrographs* menu.
3. Find and *Open* the TAPE22 file (it should be named redriv.sol if you actually ran HEC-1 as outlined above). If you did not actually run HEC-1, or are having difficulties you can read in redri.sol.

After you have successfully read the hydrograph file, you should see several small hydrographs displayed in the upper right corner of each outlet and basin in the HEC-1 *Tree Window*. You can examine the hydrographs in greater detail by selecting them for display in the hydrograph window. Two hydrographs for some of the outlets will be displayed representing the hydrograph before and after channel routing. To view the hydrographs in the hydrograph window, do the following:

1. Select one of the hydrographs by clicking on the miniature.
2. Hold the SHIFT key down while selecting more hydrographs. All currently selected hydrographs should be displayed in the hydrograph window.
3. Select the *Display Options* command from the *Display* menu.
4. Select the *Plot Options* button.
5. Turn on the *Display X grid* option.
6. Turn on the *Display Y grid* option.
7. Set any other parameters you wish to experiment with.
8. Click on the *OK* button for both dialogs.

HEC-1

Supplementary Exercise for HEC-1-Session 10

The following set of exercises builds on the previous supplementary exercises. If you haven't completed those exercises you can skip to step two and use the sf.tin file.

All files for this exercise are located in the *Supplimentary 10* directory.

1. Delineate the South Fork watershed using a TIN created from two dems: *bridal.dem* and *wallsrdg.dem*
The key data needed to create a boundary polygon and streams prior to creating a TIN are:
 - a) The watershed outlet should be at the UTM coordinates (451340, 4467015). Remember the base units for UTM coordinates are meters.
 - b) You should use the contours, shading, etc. to define streams for South Fork, Bunnells Fork, Left Fork, Middle Fork, and Right Fork as indicated on the accompanying map. You may add additional stream branches if you wish, but these must be included.
 - c) After creating the TIN and delineating the watershed you should turn in a printout showing the watershed boundary, the polygon boundary used to create the initial TIN, and the area of the watershed (as a single basin) in sq. miles.

2. Assume that a 150 meter high dam is at the outlet point of the basin. Compute the reservoir surface area, in acres, for each 10 meter elevation increment and plot the elevation-area curve. Compute the reservoir storage, in acre feet, for each elevation increment and plot the elevation-storage curve. Turn in a plot of the two curves. What are the area (acres) and storage (acre-feet) values when the reservoir is completely full (at 150 meters)?
 - a) Area _____
 - b) Storage _____

3. Divide the watershed into **two** sub-basins by creating a second outlet at the junction of Left Fork and the combined Middle Fork and Right Fork. If necessary merge the sub-basins together so that only a single basin exists upstream of the newly created outlet. What are the areas of the upper and lower sub-basins in sq. miles. Turn in a printout of your basins.
 - a) Upper Basin Area _____
 - b) Lower Basin Area _____

4. Using the gage data location in the table below to enter three gages for your watershed.
Set up an HEC-1 rainfall runoff simulation (without snowmelt) using the Gage method and Thiessen polygons for your two basins. The SCS type II 24-hr precipitation distribution should be used. Use a CN=75 for the upper basin and a CN=82 for the lower basin. Use the SCS Unit Hydrograph method for both basins, computing the lag time with the Tulsa Rural method. What is the peak flowrate and time to peak at the basin outlet?

Gage Location	24 hr. rainfall
457650, 4469140	2.1 inches
461775, 4463230	1.3 inches
450675, 4460845	1.1 inches

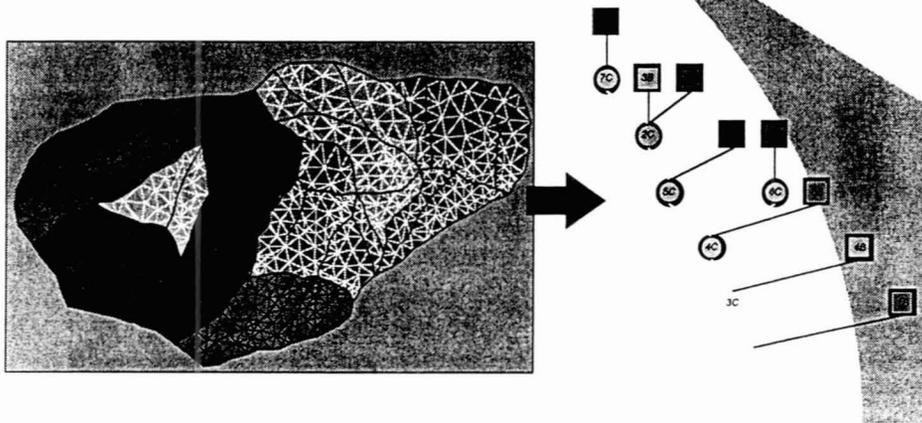
- a) Peak flowrate_____.
- b) Time to peak_____.
5. Modify your HEC-1 simulation to route the hydrograph from the upper basin through the lower basin along the South Fork. Use the Muskingum method of routing where $x = 0.2$ and K is determined from the stream reach length and an average stream velocity of 3.5 fps. What values for K and NSTPS did you use? What are the peak flowrate and time to peak at the basin outlet?
- a) K (AMSKK)_____.
- b) NSTPS_____.
- c) Peak flowrate_____.
- d) Time to peak_____.
6. If the watershed were in the wet condition (Wanielista, Table 4.6 attached) when this storm occurred and the CN values were changed appropriately, what would the peak flowrate and time to peak at the outlet be?
- a) Peak flowrate_____.
- b) Time to peak_____.
7. What is the percent change in **composite** CN and what is the % change in the outlet flowrate?
- a) % change composite CN_____.
- b) % change flowrate_____.
8. Use the Degree-Day Method to determine snowmelt runoff from the total basin for 3 days. The minimum and maximum temperatures are 38, 65, 42, 75, 38, 63, 35 F. respectively. The lapse rate is 3 F/300 meters. The elevation increments (zones) are each 300 meters. The weather station is located at the outlet. The watershed is covered with at least 5 inches of snow water equivalent. Use the Uniform Loss Method with an initial loss of .2 inches, and a loss rate of .1. For the snow loss rate (STRKS) use a value of .04 inch / hr. The snowmelt coefficient should be 0.07. You will also need to enter a very small amount of rainfall (as we had to with the homework problem) and define a lag time for the SCS Unit Hydrograph method with the Tulsa Rural method to compute the lag time. What is the peak flowrate during the simulation period? What is the time to peak? Turn in a printout of your hydrograph.
- a) Peak flowrate_____.
- b) Time of peak_____.

What if I Don't Have Digital Terrain Data?

- WMS allows you to define topologic models without having a TIN present
 - from scratch
 - reading existing files
 - Not all options for different models supported

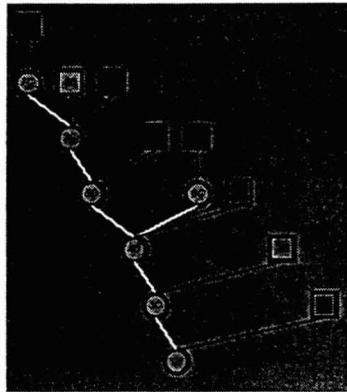
Topologic Trees from TINs

- Topologic representation of the watershed



Topologic Trees

- Can be created without the presence of a TIN



Differences/Limitations

- Reservoirs
- Diversions
- All areas and geometric parameters must be defined manually

TR-20

- Developed by the NRCS (SCS)
- Uses SCS dimensionless hydrograph
- Uses SCS curve number for rainfall losses
- Precipitation defined from synthetic or natural events
- Lumped parameter, event based model
- Typically used more often in the eastern US
- Subset of HEC-1 options

How Does TR-20 Work in WMS?

- 1 Delineate a watershed
 - Create a topologic model without a TIN
- 2 Define basin (RUNOFF) values
- 3 Define routing (REACH) values
- 4 Add diversions/reservoirs etc.
- 5 Compute hydrograph

TR-55

- Developed by the NRCS (SCS)
- Uses a “tabular” hydrograph method.
Hydrographs were initially computed and routed using TR-20
- Uses SCS curve number for rainfall losses
- Precipitation defined using Type I, IA, II, or III synthetic rainfall distributions and total precipitation for a 24-hour storm
- Extremely fast and easy

Capabilities

- Fully implemented in WMS 5.1
- Number of sub-basins and outlets limited only by computer’s memory
- Computes and routes basin and sub-basin hydrographs
- Computes a peak flow and several other sub-basin parameters without computing a hydrograph

Limitations

- Limited to a certain range of values for Time of Concentration, Travel Time, and Curve Number:
 - Tc should be less than 3.1 hours (less than 2.1 hours if no hydrograph is computed)
 - Tt should be less than 2.1 hours
 - CN should be greater than 40
- One of four 24-hour rainfall distributions may be used. There can be no user-defined rainfall distribution

Using TR-55 in WMS

The screenshot displays the TR-55 software interface. The main window is titled "TR-55" and contains several input fields and buttons. On the right side, there is a "TR-55 data window (click to get help)" which displays the following information:

Warnings for basin 4B:
* No warnings were detected in this basin.
Computations for basin 4B:
* Peak discharge (Op):
10.567 cubic feet per second

TR-55 help window
for the current basin in CFS.
The equation used to compute this value is:
 $Qp = Qu^*Am^*Q^*Fp$

Buttons: Copy to clipboard, Compute hydrograph(s), Data window display options...

Input fields in the main window:
Graphical peak discharge/tabular hydrograph data
Current basin name: 4B
Time of concentration: 8.000
Drainage area: 0.240
Rainfall: 2.300
Curve number: 78.0
Pond and swamp area: 0.000
Travel time: [empty]
Rainfall distribution: Type II

Buttons: Compute Tc - Basin Data, Compute Tc - Map Data, Compute travel time, Done

Using TR-55 in WMS

- 1 Delineate a watershed
 - Create a topologic model without a TIN or DEM.
Or, use a TIN, DEM, or Feature objects to define your model.
- 2 Define basin values (Online help messages are available)
- 3 Define outlet travel time values
 - Time computation coverage
- 4 Compute the hydrographs

Creating Topologic Models For TR-20

This chapter, if you are familiar with TR-20, will help you understand how *WMS* compares to and how it differs from traditional TR-20 modeling. It is useful for new users of TR-20 as well, since it will help you become familiar with what types of structures can be modeled with TR-20, and how they are set up in *WMS*.

The example problem illustrated here is the Red River watershed. It is chosen since it illustrates many of the basic capabilities available in TR-20. This tutorial will not lead you through every step to completely set up the model. Instead you will be shown how to create and enter data for each type of object (i.e. basins, outlets, reservoirs, diversions) and then data for other similar objects will be provided so that you can repeat the steps. A completed model of the Red River watershed is provided in the tutorial directory. If you don't wish to enter all of the necessary data, or if you want to skim through this section and look at the results, you can read this file in using the *Open TR-20 File* command in the *TR-20* menu.

The Red River watershed demonstrates the following capabilities available in TR-20:

- Calculating hydrographs using base flow, losses, and unit hydrograph methods.
- Flood hydrograph routing.
- Reservoir routing.
- Diversions.

19.1 The Topologic Tree Representation

The schematic shown in Figure 19-1 illustrates how different components in the Red River watershed are modeled using TR-20. Figure 19-1 shows a completed tree diagram for the same watershed created using WMS. A few important differences need to be noted.

- Names for all hydrograph computation units (outlets, basins, etc.) must be unique.
- Reservoirs are defined with separate icons (triangles) in the traditional TR-20 schematic, whereas in WMS they are defined at outlet points.

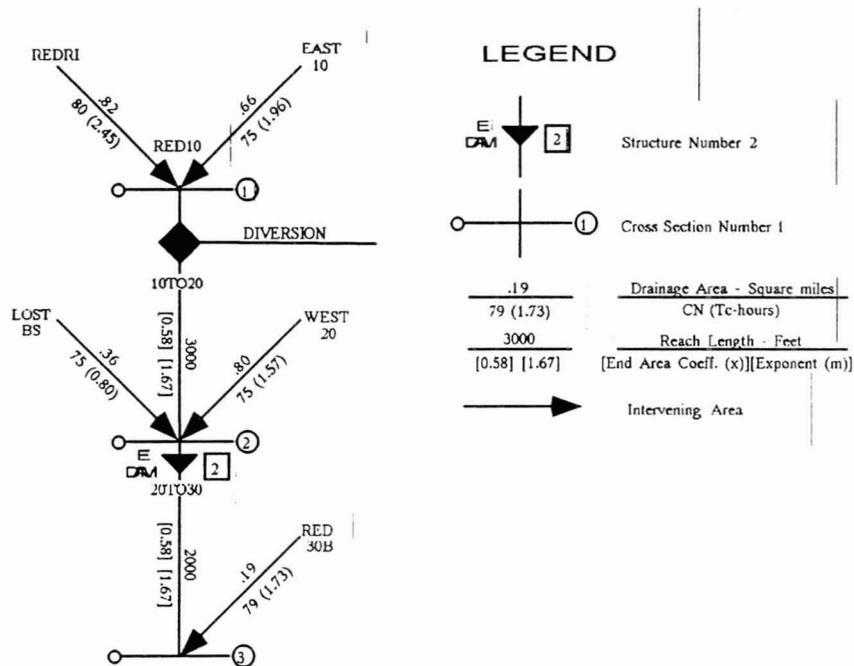


Figure 19-1 Traditional TR-20 Topological Tree Diagram For Red River Watershed.

Figure 19-2 puts the E.DAM reservoir at RED20, since there is no need to have the outlet at LOSTBR. The reservoir can be placed at the RED20 outlet.

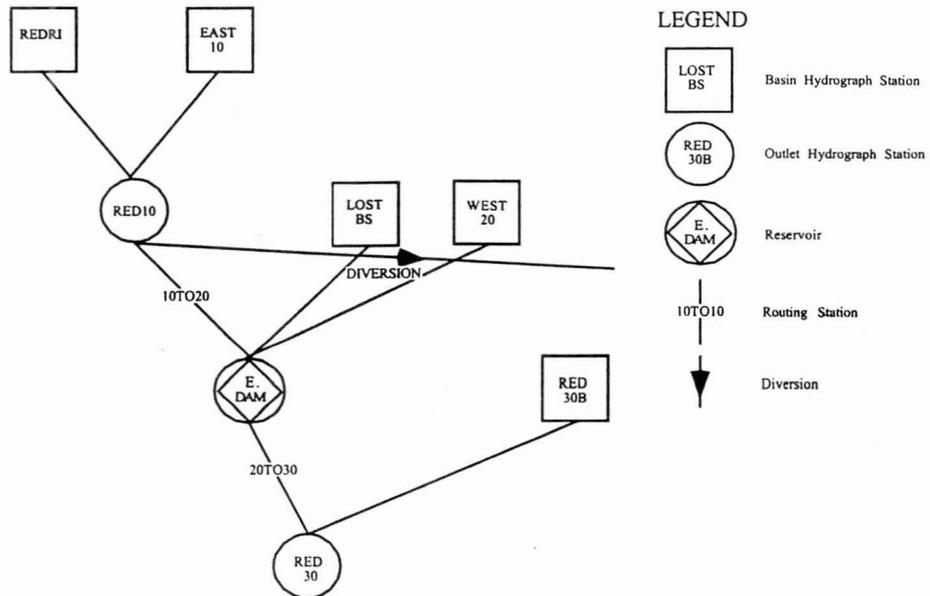


Figure 19-2 LOSTBR Outlet Eliminated with E.DAM Reservoir at RED20.

The tree defined in the remaining sections of this tutorial will correspond to the one in Figure 19-2.

19.2 Creating the Tree

Switch to the Hydrologic Modeling Module . If you are running the Demo version turn it off now. The following commands can be used to create the topologic tree for the Red River watershed as shown in Figure 19-2. All commands are in the *Tree* menu.

1. Select the *Default Model* command and change the current model to TR-20. Select the OK button.
2. Select the *Add Outlet* command. The base outlet will be created and highlighted meaning that it is currently selected. Each time you create a new outlet it becomes the selected outlet.
3. Select the *Add Basin* command. Don't worry that the names don't match those in Figure 19-2, you can change them later.
4. Select the *Add Outlet* command.
5. Select the *Add Reservoir* command.

6. Select the *Add Basin* command.
7. Select the *Add Basin* command.
8. Select the *Add Outlet* command.
9. Select the *Add Basin* command.
10. Select the *Add Basin* command.
11. Click on the Outlet named 3C to select it.
12. Select the *Add Diversion* command.

Except for the names, your tree should now look like the one in Figure 19-2. If it does not you can start over by selecting the *New* command in the *File* menu, or you can use a combination of the *Delete* and *Add* commands in the *Tree* menu to correct your tree.

19.3 Changing Hydrograph Station Names

WMS requires that all hydrograph stations have a unique six character maximum identification name. In order to help avoid this problem default names are provided whenever an outlet, basin, diversion, or reservoir is created (i.e. 1C, 2C, 1B, 2B, etc.). You can change the name of these stations to make them more identifiable, however you must remember these two limitations:

- Names can not be longer than 6 characters.
- Names must be unique.

To change the outlet names do the following:

1. Click on the outlet point 1C and select the *Edit TR-20 Parameters* command from the *TR-20* menu, or double-click on the outlet point (this automatically brings up the *Edit TR-20 Parameters* dialog).

The *Edit TR-20 Parameters* dialog contains several buttons which can be used to define/edit data for hydrograph stations. Only the buttons which pertain to the selected hydrograph station are active, all others are dimmed. You can define/edit data by clicking on the appropriate button.

1. Select the *Routing Data* button.
2. Change the *Routing combine* name from 1C to RED30 and click on the OK button.

3. Select the next upstream outlet 2C. You may wish to reposition your *Hydrologic Modeling Window* and/or the *Edit TR-20 Parameters* dialog so that you can see both the tree and the dialog. You don't need to quit the *Edit TR-20 Parameters* dialog. Data for the next selected outlet point will now be displayed.
4. Select the *Routing Data* button.
5. Change the *Routing combine* name from 2C to E.DAM. Toggle the *Define reach routing* toggle box. There is routing from this outlet so change the routing name from 2R to 20TO30.
6. Click on the *OK* button.
7. Repeat the last four steps changing 3C to RED10 and 3R to 10TO20. Be sure to turn reach routing on for outlet 3C so that the resulting hydrograph may be routed downstream from this point.

To change Basin names do the following:

1. Click on the basin 1B. If you accidentally closed the *Edit TR-20 Parameters* dialog then double-click on 1B to bring it back up.
2. Select the *Basin Data* button.
3. Change the basin name from 1B to RED30B and click on the *OK* button.
4. Click on basin 2B.
5. Select the *Basin Data* button.
6. Change the basin name from 2B to WEST20 and click on the *OK* button.
7. Repeat the last three steps changing 3B to LOSTBS, 4B to EAST10, and 5B to REDRI.

To change the diversion name do the following:

1. Click on the diversion arrow D1.
2. Select the *Diversion Data* button.
3. Change the *Diversion name* from D1 to DIVERT and click on the *OK* button.
4. Select the *Done* button.

19.4 Creating the Tree with Feature Objects

In more recent versions of WMS, the hydrologic modeling tree can be created directly from feature arcs (streams), and polygons (basins) (see chapter 4). This means that a tree can be created even without a map from which actual stream networks and watershed boundaries are digitized. In such cases the feature objects represent a "picture" or "drawing" of the watershed and are not used to calculate areas and lengths. However, the drawing itself may be much more instructive to clients and reviewers than just the modeling tree. If you want you can read in a map file that contains the same watershed definition you have just created by following these three steps:

1. Select the *New* command from the *File* menu.
2. Select the *Open* command from the *File* menu.
3. Find and open the file named *redri.map*.

If you choose to do this your model will replace (but be equivalent to) the one you have just defined and should look like Figure 19-3.

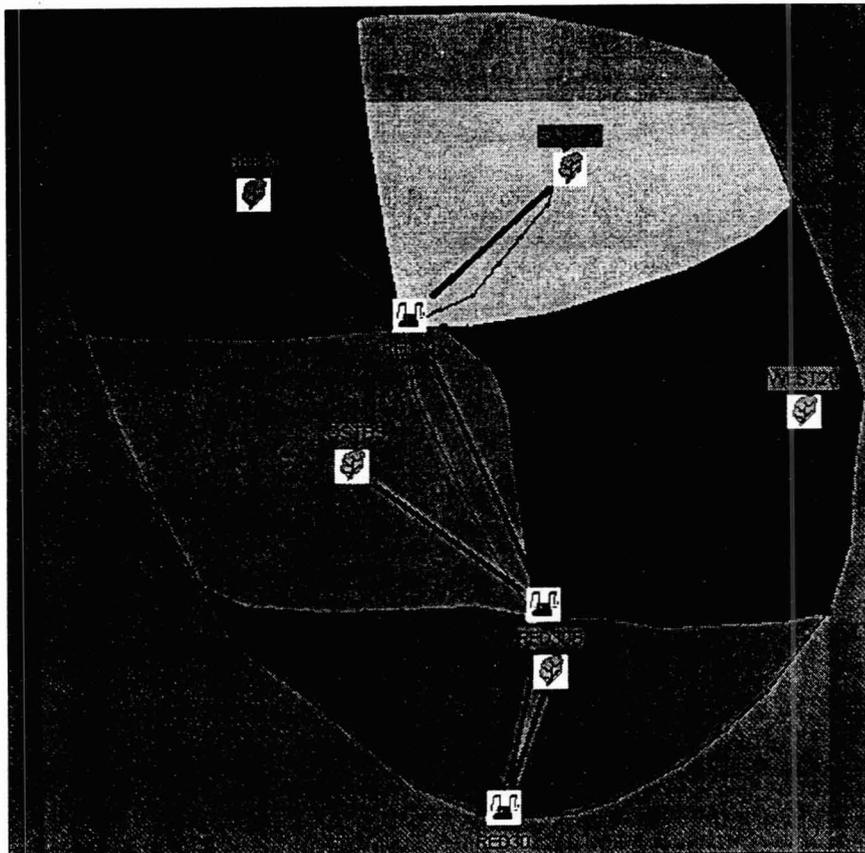


Figure 19-3 Red River Watershed Defined from Feature Objects.

19.5 Job Control Data

Job control data includes computational starting time and data, interval, and length, general output specification and other custom features of TR-20. To modify this data do the following:

1. Select the *Job Control* command from the *TR-20* menu.
2. Change the main time increment to 0.25.
3. Select the *Define Precipitation* button.
4. Select *Use standard SCS rainfall distribution* for the precipitation distribution and *Emergency spillway and freeboard design* for the SCS rainfall table type.
5. Toggle the options to define rainfall depth and rainfall duration on.
6. Enter 5.0 inches for the rainfall depth.
7. Enter 4.0 hours for the rainfall duration.
8. Select the *OK* button on both dialogs.

19.6 Defining Basin Parameters

TR-20 requires that all basins have an appropriate area, curve number, and time of concentration defined. This tutorial will describe how these parameters can be defined for one basin and then a table of values will be provided so that the parameters can be defined for the remaining basins.

1. Click on the basin named RED30B.
2. Select the *Edit TR-20 Parameters* command from the *TR-20* menu. Once again you may wish to reposition windows.
3. Select the *Basin Data* button.
4. Change the *area* to .19.
5. Input 79 for the SCS Curve number.
6. Input 1.73 for the time of concentration.
7. Select the *OK* button.

Basin parameters for RED30B should now be defined. You can now select the other basins one at a time and repeat the above steps for defining area, curve number, and time of concentration using the data given in Table 3.1

Name	Area	Curve Number	Time of Concentration
RED30B	.19	79	1.73
WEST20	.80	75	1.57
LOSTBS	.36	75	0.80
EAST10	.66	75	1.96
REDRI	.82	80	2.45

19.7 Defining Routing Parameters

Routing parameters are defined in a similar fashion as basins. For the Red River model channel routing needs to be defined from RED10 to E.DAM (10TO20) and from E.DAM to RED30 (20TO30). This can be done with the following steps:

1. If the *Edit TR-20 Parameters* is still active select the outlet RED10. If you have closed it already select RED10 and then select the *Edit TR-20 Parameters* command from the *TR-20* menu.
2. Select the *Routing Data* button.
3. Select the option to *Define reach routing*.
4. Enter 3000 feet for the reach length.
5. Notice that there are two methods for defining reach routing. We will use the second (x and m) method. Keep the default x and m values.
6. Click on the *OK* Button.
7. Select the outlet E.DAM.
8. Select the *Routing Data* button.
9. Select the option to *Define reach routing*.
10. Enter 2000 feet for the reach length.
11. Click on the *OK* Button.

19.8 Defining Diversion Data

Diversion data can be defined by following the steps below:

Diversions cannot be saved with feature objects (TINs, or DEMs either) so after creating a watershed using feature objects diversions still need to be entered using the commands in the *Tree* menu of the *Hydrologic Modeling Module*.

1. Select the Hydrologic Modeling module icon .
2. Select the Select Basin icon tool .
3. Click on the basin named EAST10 to select it.
4. Select the Add Diversion command from the Tree menu.
5. Click on the Outlet point named E.DAM to select it.
6. Select the Retrieve Diversion command from the Tree menu.

To change the diversion name do the following:

1. If the Edit TR-20 Parameters is still active select the diversion icon named D1. If you have closed it already, select D1 and then select the Edit TR-20 Parameters command from the TR-20 menu.
2. Select the Diversion Data button.
3. Change the name from D1 to DIVERT.
4. Enter 860 for the constant discharge above which the flow will be diverted.
5. Click on the OK Button.

19.9 Defining a Reservoir

In *WMS*, a reservoir is defined in TR-20 by adding a reservoir at an outlet.. In this section, we will define the reservoir data at the E.DAM outlet.

1. If the *Edit TR-20 Parameters* is still active select the outlet E.DAM. If you have closed it already select E.DAM and then select the *Edit TR-20 Parameters* command from the *TR-20* menu.
2. Select the *Reservoir Routing* button.

3. Toggle *Define reservoir routing* to "on".
4. Select the *Define Reservoir Data* button.
5. Select the *New* button.
6. Enter the following values for elevation, discharge, and storage:

Elevation	Discharge	Storage
850.00	0.00	21.00
851.50	15.00	100.00
853.30	54.00	205.00
856.50	200.00	325.00
858.00	496.00	955.00

- Click on the *OK* button for both dialogs.
- Click on the *Done* button of the *Edit TR-20 Parameters* dialog.

19.10 Running and Saving TR-20 Files

You have now completed the definition of the Red River watershed model and are ready to run the analysis (If you haven't done all the work outlined in the previous section you can read in the already defined model *redri.dat*). TR-20 is a stand-alone program, but can be launched from within *WMS* using the following steps:

1. Select the *Check Model* option from the *TR-20* menu.

This will perform a check on the data and try to determine problems that should be addressed before TR-20 can be successfully run. If you followed the steps in the preceding sections correctly there should be no problems. If you discover some potential problems you should correct them before moving on.

If the model checker does not find any errors, it does not insure that TR-20 will compute the correct result, but it should help.

2. Select *Done*.
3. Select the *Run TR-20* option from the *TR-20* menu.

TR-20 needs the names of the TR-20 input file, the ASCII output file, and the solution file containing hydrograph information which can be read back into *WMS* for display.

4. Enter *redriv* in the *prefix for all files* edit field and select the *Update* button. If necessary select a directory to save the TR-20 files in by clicking on the file browser button. When running TR-20 the input file is automatically saved before TR-20 is launched.
5. Click on the *OK* button. TR-20 will run and the hydrographs will be read in at the hydrograph stations.

The *View File* command in the *File* menu can be used to examine the ASCII output file should problems occur. On PC's the view file command brings up a simple view only program, whereas on UNIX operating systems you can specify an editor (*vi* by default) to bring the file up in.

19.11 Displaying Hydrograph Results

The ".THY" file generated by TR-20, and stored in the third file specified when running TR-20 from within *WMS* contains hydrographs for each basin of the watershed, for the routed hydrographs from outlet points, and for the combination of basin and routed hydrographs at each outlet. You can read this file and display the hydrographs in the hydrograph window. To read the hydrograph file, do the following:

1. Select the *Show Hydrograph Window* command from the *Display* menu (if the hydrograph window is already mapped you will not need to do this).
2. Select the *Open* command from the *Hydrographs* menu.
3. Find and *Open* the ".THY" file (it should be named *redri.THY* if you actually ran TR-20 as outlined above). If you did not actually run TR-20, or are having difficulties you can read in *redri.thy*.

After you have successfully read the hydrograph file, you should see several small hydrographs displayed in the upper right corner of each outlet and basin in the TR-20 *Tree Window*. You can examine the hydrographs in greater detail by selecting them for display in the hydrograph window. Two hydrographs for some of the outlets will be displayed representing the hydrograph before and after channel routing. To view the hydrographs in the hydrograph window, do the following:

1. Select one of the hydrographs by clicking on the miniature.
2. Hold the *SHIFT* key down while selecting more hydrographs. All currently selected hydrographs should be displayed in the hydrograph window.
3. Select the *Display Options* command from the *Display* menu.

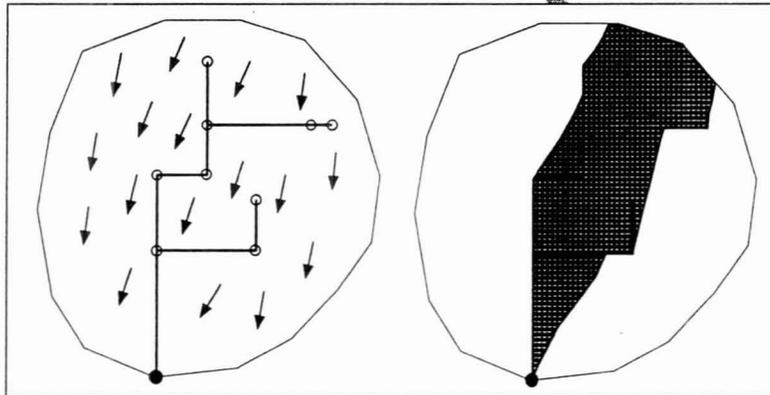
4. Select the *Hydrologic Modeling* tab.
5. Select the *Plot Options* button.
6. Turn on the *Display X grid* option.
7. Turn on the *Display Y grid* option.
8. Set any other parameters you wish to experiment with.
9. Click on the *OK* button for both dialogs.



Urban Drainage in WMS

- Uses geographic data but allows for the following to act as “streams”
 - Roads
 - Railroads
 - Canals
 - Any feature which “channels” water
- Interface to the rational method

Runoff



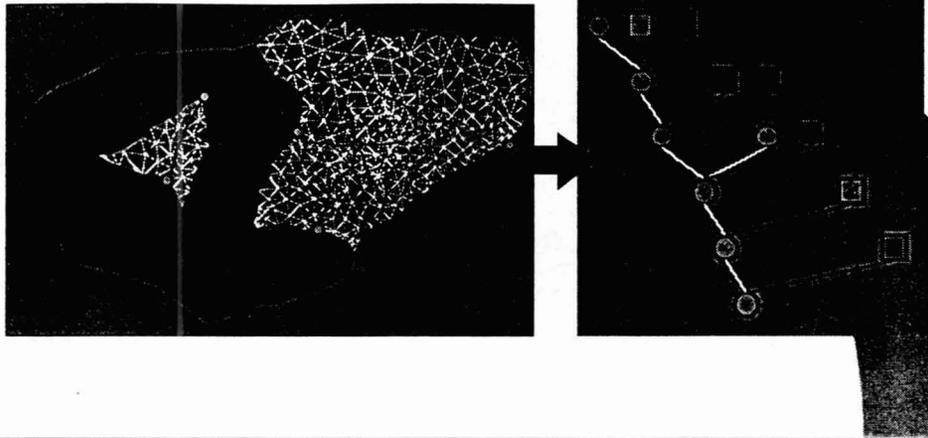
Inserting Roads

- Currently created as “stream” feature arcs
- Future versions will contain a highway feature arc type
 - it will act like a stream
- “Underground” pipes may also be placed a TIN

Does it Work?

- Background elevation data provides general overland flow characteristics
- Roads, railroads, canals, etc. provide stream flow
- You do not have to have the roads sloping downstream along every segment
 - BE CAREFUL!

Topologic Tree for Rational Method



Rational Method Equation

- $Q = CiA$

- C - runoff coefficient (between 0.0 and 1.0)

- Typical values can be found on 12-2 of WMS Reference Manual

- i - rainfall intensity (determined using t_c)

- A - drainage area in acres

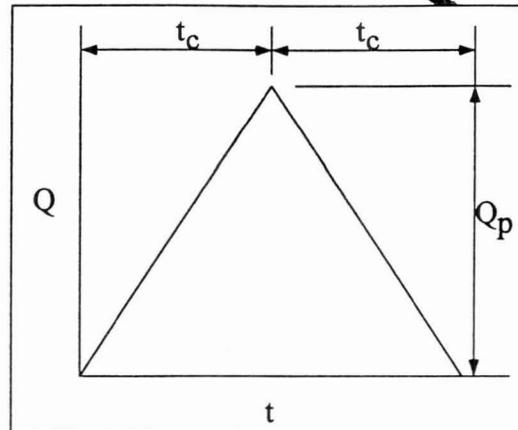
Limitations

- Basins should be smaller than 1 mi²
 - Analysis of larger regions can be done if the effects of routing are properly considered
- All catchment losses are incorporated through C
 - Area weighted runoff coefficients can be used
 - No catchment will have a constant loss, but works well for impervious regions (urban)

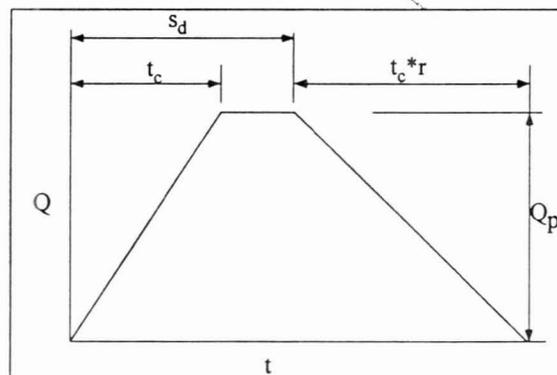
Creating Hydrographs

- Rational
- Modified
- Universal
- Dekalb
- User defined

Rational Hydrograph

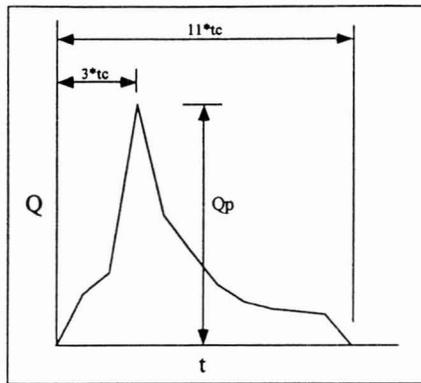


Modified Rational Hydrograph



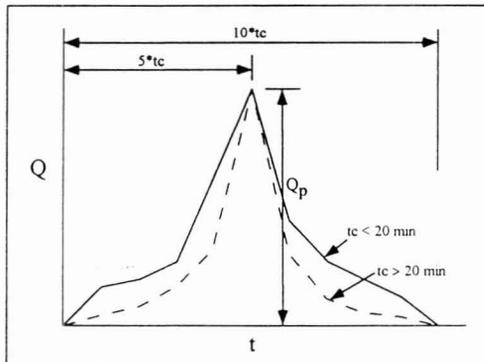
S_d - Storm duration
 r - Falling limb coefficient

Universal Hydrograph



t/t_c	Q/Q_p
0	0.00
1	0.21
2	0.30
3	1.00
4	0.54
5	0.39
6	0.25
7	0.18
8	0.15
9	0.14
10	0.13
11	0.00

Dekalb Hydrograph



t/t_c	Q/Q_p $t_c < 20$	Q/Q_p $t_c \geq 20$
0	0.00	0.00
1	0.16	0.04
2	0.19	0.08
3	0.27	0.16
4	0.34	0.32
5	1.00	1.00
6	0.45	0.30
7	0.27	0.11
8	0.19	0.05
9	0.12	0.03
10	0.00	0.00

User Defined Hydrograph

- Number of ordinates specified
- Values for t/t_c and Q/Q_p defined

How Does Rational Method Work in WMS?

- 1 Delineate a watershed
- 2 Initialize a Rational method analysis
- 3 Define runoff coefficients
- 4 Define rainfall intensities
 - Hydro 35, NOAA Atlas 2
 - t_c
- 5 Compute Q
- 6 Generate a hydrograph

Rational Method Interface

The Rational Method is one of the simplest and best known methods routinely applied in urban hydrology. Peak flows are computed from the simple equation:

$$Q = kCiA$$

where:

- Q - Peak flow
- k - conversion factor
- C - Runoff coefficient
- i - Rainfall intensity
- A - Area

In this tutorial you will learn how to solve problems using a digital terrain model and the Rational Method.

12.1 Reading in Terrain Data

The terrain model used in this tutorial is for a small portion of a city. The elevation data was obtained by digitizing a contour map.

1. Select the *TINs* module icon .

2. Select *Open* from the *File* menu.
3. Find and *Open* the TIN file named “*afpts.tin*”.
4. Select the *Triangulate* command from the *TINs* menu.
5. Select the *Display Options* command from the *Display* menu.
6. Turn off *Vertices* and *Triangles*.
7. Select *OK*.

12.2 Creating a TIN for Basin Delineation

The TIN that has been read in and triangulated will be used as a background elevation map for the creation of a TIN that will be used for basin delineation.

12.2.1 Opening a DXF file

1. Select the *Map* module icon .
2. Select the *Import* command from the *DXF* menu.
3. Find and *Open* the file named “*af.dxf*”.

12.2.2 Converting DXF Data to Feature Objects

The DXF data represents a small street network and a boundary for the study area. In order to use the DXF data to help create a TIN it must be converted to feature objects.

1. Select the *DXF->Feature Objects* command in the *DXF* menu.
2. Turn off *Feature points from dxf points*.
3. Select *OK*.
4. Select the *Delete* command from the *DXF* menu.
5. Confirm that you want to delete the DXF data by selecting *OK*.

12.2.3 Preparing the Feature Objects to Create a TIN

The way line segments are connected in DXF files is often random. However, in order to create a network of stream feature arcs, line segments must be connected from downstream to upstream. WMS can automatically reorder feature arcs by:

1. Select the Select Points/Nodes tool .
2. Select the point indicated in Figure 12-1.

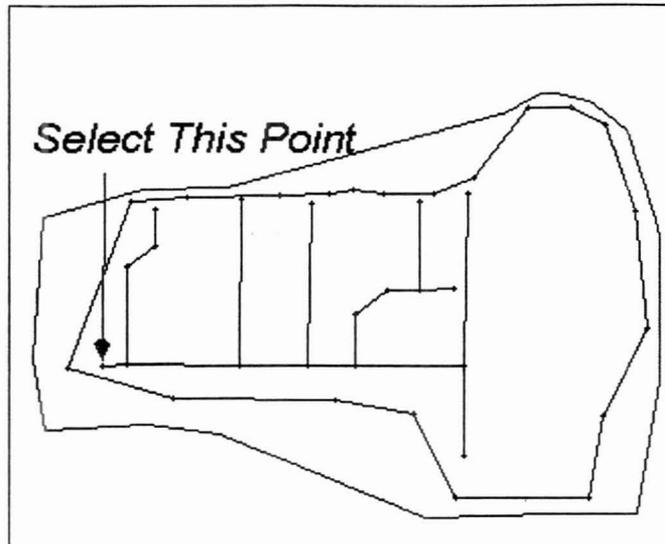


Figure 12-1 Location of Watershed Outlet Used for Stream Reordering.

1. Select the *Reorder Streams* command from the *Feature Objects* menu.

Now that the arc segments are properly ordered they can be converted to stream arcs. First, you must make the coverage in which you are working a Drainage coverage (right now it is a General coverage).

1. Select *Coverages...* from the *Feature Objects* menu.
2. Make sure the coverage named "default coverage" is selected in the window in the upper left of the dialog.
3. Select *Drainage* from the *Attribute Set* drop-down box in the lower right area of the dialog.
4. Select OK.

Now that the coverage is a drainage coverage, you are ready to convert the arcs to streams and basins.

1. Select the *Select Network* tool .
2. Select any arc that is part of the stream network.
3. Select the *Attributes* command from the *Feature Objects* menu.
4. Select *Stream*.

5. Select *OK*.
6. Select *Refresh* from the *Display* menu.

12.2.4 Redistributing Arc Vertices

The vertices along the arcs need to be redistributed in order to create a TIN more suitable for basin delineation.

1. Select the *Select Arc* tool .
2. Select the *Select All* command from the *Edit* menu.
3. Select the *Redistribute* command from the *Feature Objects* menu.
4. Specify the *Subdivide each edge uniformly* option.
5. Use a *specified spacing* of 100 (feet in this case).
6. Select *OK*.

12.2.5 Creating the TIN

1. Select *Build Polygon* from the *Feature Objects* menu.
2. Select *OK* to use all feature arcs.
3. Select the *Select Polygon* tool .
4. Click anywhere inside the boundary polygon so that it becomes selected.
5. Select the *Create TIN...* command from the *Feature Objects* menu.
6. Select *No* when asked if you want to save your data.
7. Make sure the option to *Interpolate from existing TIN* is selected.
8. Select *OK*.

12.3 Defining Catchment Areas

Catchment areas need to be defined in order to determine areas. The computed areas will later be used in the Rational Method equation.

12.3.1 Adding Outlets and Assigning Triangles to Basins

1. Select the *TINs* module icon .
2. Select the *Smooth Pits* command from the *TINs* menu.
3. Using the *Select Vertices* tool , and while holding the SHIFT key down select the four TIN vertices at the stream branching points indicated in Figure 12-2.

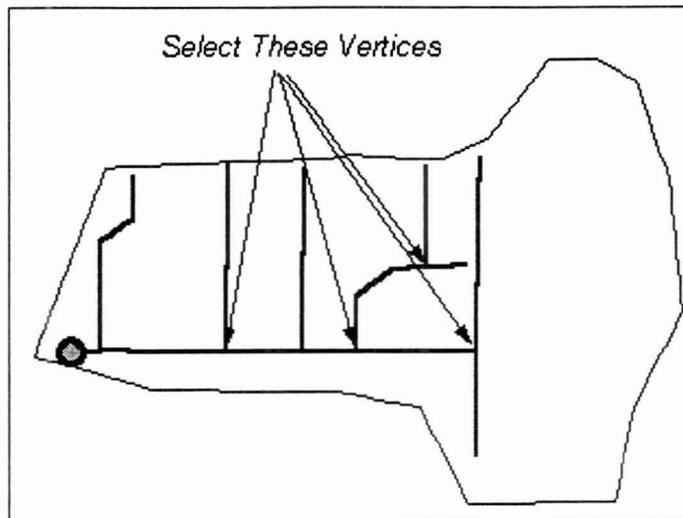


Figure 12-2 Location of Outlet Vertices.

1. Select the *Add Outlets* command from the *Streams* menu.
2. Select the *Define Basins* command from the *Drainage* menu.

Because of the way vertices are connected by triangle edges, locations where flow splits may occur. If you receive a message about triangles belonging to more than one basin you need to do the following two commands.

1. Select the *Correct Split Flow* command from the *Drainage* menu.
2. Repeat the *Define Basins* command from the *Drainage* menu.

Since basins are defined based on flow from triangle centroids you will need to split triangles which straddle basin boundaries. After refining boundaries you will also want to eliminate triangles which do not belong to any of the catchment areas.

1. Select the *Refine Boundaries* command from the *Drainage* menu.
2. Select the *Delete Null Basin Triangles* command from the *Drainage* menu.

3. Select *OK* to verify deletion.

12.3.2 Merging Catchment Areas

By default WMS creates a separate catchment area (basin) for each upstream branch of the stream. However, we only want to consider a single basin upstream from the outlet points so we must merge the pairs of adjacent basins together.

1. Select the *Display Options* command from the *Display* menu.
2. Select the tab labeled *Drainage*.
3. Turn off the *Stream networks*.
4. Select *OK*.
5. Select the *Select Basin* tool .
6. While holding down the *SHIFT* key select both basins labeled *A* in Figure 12-3.
7. Select the *Merge Basins* command in the *Drainage* menu.

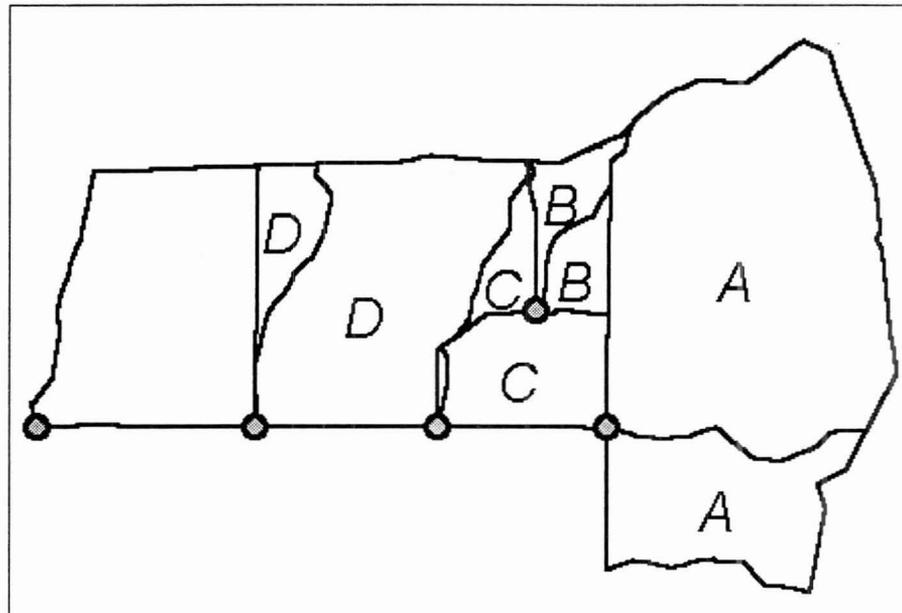


Figure 12-3 Delineated Catchment Areas Before Merging.

1. Repeat the last two steps for basins labeled *B*, *C*, and *D*.
2. Select the *Display Options* command from the *Display* menu.
3. Select the *Drainage* tab.

4. Turn *Stream networks* back on.
5. Turn on *Fill drainage basins*.
6. Select *OK*.

12.3.3 Defining Basin Data

Now that the delineation of catchment areas is complete you can compute the contributing area (as well as many other parameters) for each.

1. Select the *Compute Basin Data* command in the *Drainage* menu.
2. Set the *Basin Areas* option to be *Acres*.
3. Select *OK*.

12.4 Running a Rational Method Simulation

The areas computed from the TIN can now be used in setting up a Rational Method simulation of the urban development. Each of the outlet points represents an inlet to a storm drain.

1. Select the *Hydrologic Modeling* module icon .
2. Select basin "*Basin 5*" as labeled in Figure 12-4 by clicking on the basin icon inside the basin.
3. Select the *Run Simulation* command from the *Rational* menu.

The *Rational Method* dialog should appear. While this dialog is up you can continue to interact with the menus, tools, and windows of *WMS*. As you select different basins, the current parameters will be updated in the dialog.

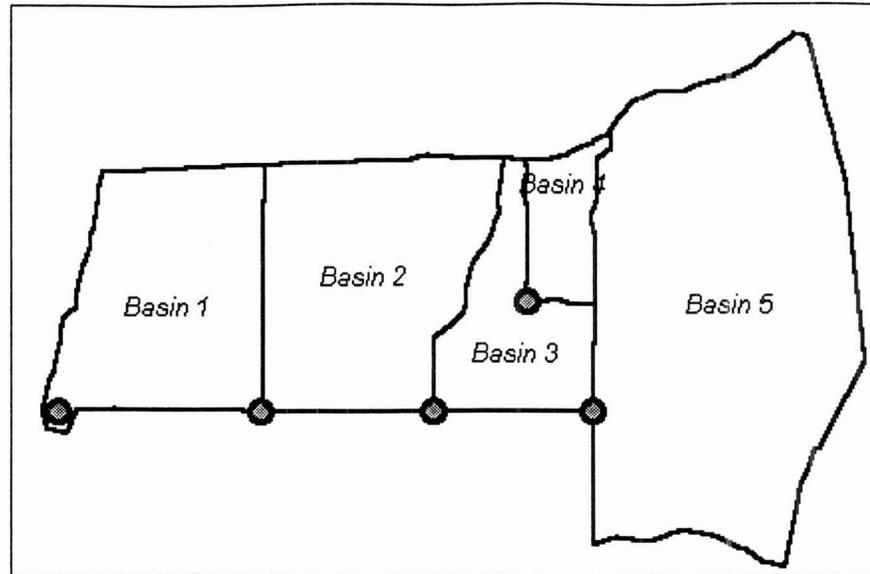


Figure 12-4 Final Delineated Basins.

12.4.1 Defining the Runoff Coefficients and Time of Concentration

The runoff coefficient, C , is used to account for losses between rainfall and runoff. The more developed a catchment is, the higher the C value it will have. The *WMS* reference manual contains a table of typical C values for different land use conditions.

1. Enter a value of .60 for C .
2. Enter a value of 35 for Time of Concentration.
3. Select the basin labeled "Basin 4" in Figure 12-4 by clicking on its square icon in the *Hydrologic Modeling Tree Window*.
4. Enter a value of .75 for C .
5. Enter a value of 14 for Time of Concentration.
6. Repeat this process for the other three basins, using the table below to fill in values for C and t_c .

Basin Name	Runoff Coefficient C	Time of Concentration t_c
Basin 1	.85	22
Basin 2	.85	20
Basin 3	.8	18
Basin 4	.75	14
Basin 5	.6	35

12.4.2 Defining the Rainfall Intensity (i)

As part of the WMS interface to the Rational Method you can compute IDF curves using either HYDRO-35, NOAA, or user defined data. For this tutorial we will use HYDRO-35 data and a recurrence interval of 10 years.

1. Select the "Basin 5" once again.
2. Select the *IDF Curves* button from the basin section of the *Rational Method* dialog.
3. Make sure the HYRO-35 radio group button is selected and select the Define Data button.
4. Enter the following values to define IDF curves using HYDRO-35

Time	Intensity
2 yr. 5 min.	.47
2 yr. 15 min.	.97
2 yr. 60 min.	1.72
100 yr. 5 min.	.81
100 yr. 15 min.	1.75
100 yr. 60 min.	3.60

1. Select the OK button after correctly entering the rainfall values.

The IDF curves for the 2, 5, 10, 25, 50, and 100 year recurrence intervals will be drawn, and values listed for selected times given in the windows on the right of the *IDF Computation* dialog.

1. From the text window in the upper right hand part of the dialog click on the line of data for the 10-yr recurrence interval as shown in Figure 12-5.

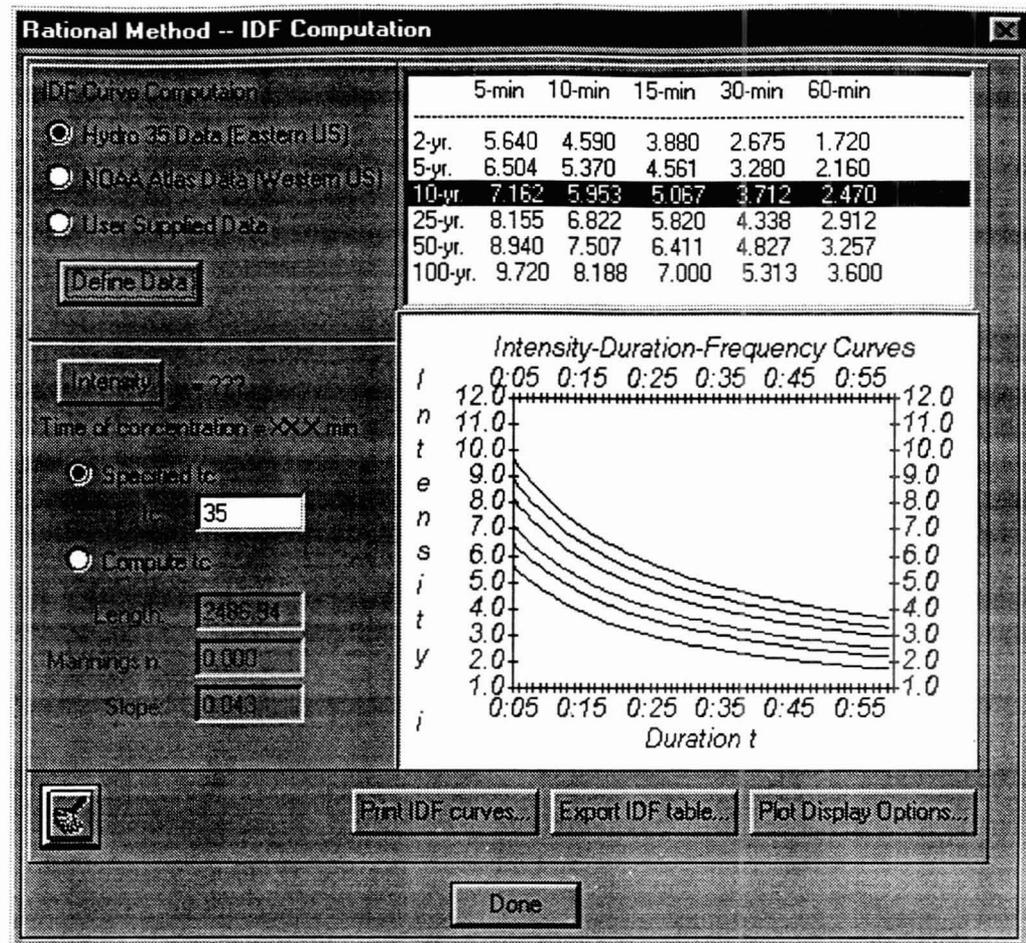


Figure 12-5 IDF Computation Dialog with the 10-yr Recurrence Interval Selected.

The rainfall intensity is determined from the selected interval by using the previously defined value for time of concentration.

1. Compute i by selecting the *Intensity* button.
2. Select the *Done* button. The value of i computed using the *IDF-Computation* dialog will be placed in the edit field for this basin.
3. Note also that the input for this basin is complete and a value for runoff Q has been computed. a summary of the rational equation and results is in the upper right window of the dialog.

The HYDRO-35 data only needs to be entered once (unless different data is to be used for different basins), so the rainfall intensity for the remaining basins can be defined using the following steps.

1. Select the icon for "Basin 4".
2. Select the *IDF Curves* button.

3. Select the line of text for the *10-yr* recurrence interval.
4. Select the *Intensity* button.
5. Select *Done*.
6. Repeat these steps for Basins 3, 2, and 1.

12.4.3 Defining Hydrographs

As the data entry for each basin is completed, a peak flow (Q) is computed and listed in the text display window. The Rational Method equation does not produce a hydrograph. However, several different unit-dimensionless hydrographs can be used to distribute the peak flow through time to create a runoff hydrograph.

1. Select the *Show Hydrograph Window* command from the *Display* menu (unless the hydrograph window is already visible).
2. Select the basin labeled "*Basin 5*" in Figure 12-4.
3. Select the *Define Hydrographs...* button.
4. Select the *Universal hydrograph method* from the drop down list.
5. Select *DONE*.
6. Click on the small hydrograph box of *Basin 5* in the *Graphics* window.

You should see the hydrograph displayed in the *Hydrograph Window*. You may want to resize this window to inspect the chart.

12.4.4 Combining Runoff from Multiple Basins

WMS allows you to define lag times (time of travel) in order to compute runoff from multiple upstream basins and account for the time of travel between sub-basin outlets. Once the lag times have been defined, a single runoff hydrograph for the entire watershed can be computed using one of two different methods.

1. Select one of the upper-most outlet icons.
2. Enter a value of 5 minutes for the *Routing lag time*.
3. Select the other upper-most outlet.
4. Enter a value of 5 minutes for the *Routing lag time*.
5. Select the next downstream outlet.

6. Enter a value of 7 minutes for the Routing lag time.
7. Select the next downstream outlet.
8. Enter a value of 6 minutes for the Routing lag time.

The last or bottom-most outlet does not need to have a Routing lag time defined since the hydrograph accumulations will occur at this point.

1. Select the bottom-most outlet point. Note the results for this outlet and all upstream outlets and basins are displayed in the window in the upper right of the dialog.
2. Select the *Define Hydrographs...* button.
3. Choose the *Route by summing* method.
4. Choose the *Universal hydrograph* method.
5. Select *DONE*.
6. Select the hydrograph box for the bottom-most outlet.
7. Inspect the hydrograph displayed in the *Hydrograph Window*.

12.4.5 Defining Rainfall Intensities at the Outlet to Compute Runoff Hydrographs

Besides the method of summing hydrographs at an outlet and then lagging to the next outlet by the lag time, WMS can also be used to determine composite rational method parameters at the outlet for computing hydrographs. With this method, the time of concentration is defined as the longest flow time from upstream basins (times of concentration) combined with any lag times from channels. The area is the cumulative upstream area, and the runoff coefficient is determined as an area weighted value from the upstream basins. With the time of concentration at the outlet defined you will need to determine the appropriate rainfall intensity. Then instead of summing the upstream hydrographs to develop the hydrograph at the outlet, WMS will use the rational method equation to compute a peak flow and multiply this peak flow by the selected hydrograph method (the same as is done for basins) to determine the hydrograph at the outlet. To do this:

1. Select one of the upper-most outlet icons in the *Hydrologic Modeling Tree* window.
2. Select the *IDF Curves* button (for outlets).
3. Select the line of text for the *10-yr*-recurrence interval.
4. Select the *Intensity* button.

5. Select *Done*.
6. Repeat these steps for the remaining outlets.

The last or bottom-most outlet does not need to have a Routing lag time defined since the hydrograph accumulations will occur at this point.

1. Select the bottom-most outlet point.
2. Select the *Define Hydrographs...* button.
3. Choose the *Traditional* method.
4. Choose the *Universal hydrograph* method.
5. Select *DONE*.
6. Select *DONE* in the *Rational Method* dialog also.
7. Select the hydrograph box for the bottom-most outlet.

You should get some kind of an idea about the difference in the resulting hydrographs from the two methods. You may want to inspect other hydrographs at different locations. Just click on the box in the *Graphics Window* to do so.

12.5 Adding a Detention Basin

If you compute runoff using the route by summing method then you can route hydrographs through detention basin structures defined at any of the outlet locations.

1. Select the outlet that defines Basin 5 in Figure 12-4.
2. Select the *Run Simulation* command from the *Rational* menu.
3. Select the *Define Reservoir* button.

You will now define a hypothetical detention basin facility from approximate geometric parameters. WMS can compute a storage capacity curve for a rectangular basin. You could also use a TIN to develop a storage capacity curve, if the TIN had sufficient resolution to accurately define the storage-depth relationship. Further, you could enter a pre-computed storage capacity curve.

4. Select the *Define...* button in the *Storage Capacity* box.
5. Select the *Known Geometry* option.

6. Enter *500* feet for *Length*.
7. Enter *500* feet for *Width*
8. Enter a *Depth* of *30* feet.
9. Enter a *Side slope* of *2*
10. Leave the *Base elevation* at *0.0* (It will be assumed on-grade at the outlet location).
11. Select *OK*.

You will now define a standpipe and spillway (weir) for outlet structures and WMS will compute the elevation-discharge relationship automatically. In addition to standpipes and weirs you can define low-level outlets, or you can enter a pre-computed elevation-discharge relationship.

1. Select the Define... button in the Elevation Discharge box.
2. Select the Discharge Structures option.
3. Select the Add Standpipe button.
4. Set the pipe diameter to 4 feet.
5. Set the Standpipe elevation to 15 feet.
6. Select the Add Weir button.
7. Set the Weir width to 20 feet.
8. Set the Weir elevation to 25 feet.
9. Select OK.
10. Select OK.

You have now defined a detention facility that has a standpipe and a spillway for control structures. The incoming hydrograph to this outlet point will be routed through the detention facility before being routed downstream and combined with the hydrographs of other basins.

1. Select the bottom-most outlet point.
2. Select the *Define Hydrographs...* button.
3. Choose the *Route by summing* method.
4. Choose the *Universal hydrograph* method.

5. Select *DONE*.
6. Select the hydrograph box for the bottom-most outlet.

12.6 Conclusions

In this tutorial you have learned some of the options available for using the rational method in *WMS*. You will want to continue experimenting with the different options so that you can become familiar with all the capabilities in *WMS* for doing Rational Method simulations.



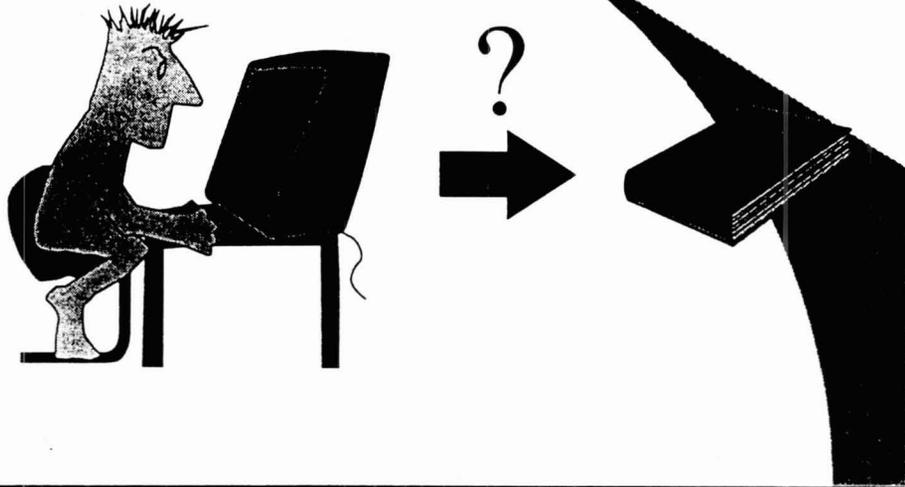
Drainage Calculators

- FHWA has sponsored some work
- Goal is to replace HYDRAIN
 - Hydrology
 - WSPRO
 - HY8
 - NFF
 - HEC12
 - HYDRA
 - Others

WMS Implementations

- Channel calculator
 - Manning's Equation
- Weir calculator
- Detention Basin calculator
- Curb & Gutter calculator
- Culvert calculator

Creating Reports



Methods...

- Printing
- Copy to Clipboard
- Exporting
- Screen Image Capture

Printing

- WMS Uses standard Microsoft Windows printing utilities
 - Same drivers supported
 - Same headaches involved
- Many times you will want to print to a postscript file

Copy to Clipboard

- Under Edit menu
- Convenient for moving graphics from WMS to Word/WordPerfect or other packages
- Only available on PC versions

Export

- DXF
 - text, some other capabilities not supported
- ArcView Shape files
- DLG
- Hydrographs
 - WMS format
 - Spreadsheet format
- Text files for output summaries

Screen Capture

- Several different tools available
 - Paint Shop Pro
 - Downloadable from ECGL web pages
 - Corel
 - Harvard
 - HiJack
- Several different image formats
 - TIFF, GIF, PCX, etc.

File Types

- WMS native file types
 - opened using the File Open command
 - exceptions
 - Hydrograph files
 - DEM files
- Hydrologic model input files
 - opened/saved from respective menus
- Other file types
 - opened using the File Import command
 - some exceptions
 - TIFF Images
 - DXF Files

WMS Native Files

- Super
- Map
- Image
- TIN
- *DEM
 - very large, not something you want to use often
- Hydrograph
- Others (scatter point, 2D grid, data set, image)

Hydrologic Model Input Files

- HEC-1
 - TR20
 - TR55
 - NFF
 - Rational
- } Strictly Defined
- } WMS defined for read/write

Foreign File Formats

- The File Import command
 - DEMs, Shape files, tabular data, grids, misc.
- TIFF (Images Import command)
 - Some advice from Alan
- DXF (DXF Import command)
 - AutoCAD release 13 compliant (most cases)

Time Series Files

- WMS Defined
- You can save your time series and then import again later

Defaults File

- WMS has built in defaults
- Customized defaults can be saved in two ways
 - as a *.ini file in the windows system directory
 - as a *.ini file which is specific to a model and included in a super file

Scattered Data and 2D Grids

Estimated Time: 20 minutes

The Scattered Data module and Grid module can be used in combination to visualize any set of scalar values which can be assigned to unique x-y positions. The scalar value used in this tutorial is rainfall intensity generated from NEXRAD radar files, but they could just as easily represent rain gages or other locations for which rainfall intensities or accumulations are known.

16.1 Objectives

In this tutorial you will learn how to import scattered data with an accompanying data set and interpolate from the scattered data to a grid for visualization by:

- Reading a scattered data file.
- Creating a bounding grid around the scattered points.
- Interpolating from scattered data.
- Contouring a grid.
- Mapping scalar values to elevations.
- Using virtual gages to examine the time variance of the data.
- Animating the transient rainfall data.

16.2 Reading the Data File

The scattered data file and data points used in this tutorial were derived from a portion of a NEXRAD radar scan. After reading the scattered points, a data set representing rainfall intensities will be imported. To read the file, do the following:

1. Select the *Scattered Data* module icon .
2. Select *Open* from the *File* menu.
3. Use the file browser to find and open the file "rainfall.xy". This file should be found in the tutorial directory.
4. Select the *Data Browser* command in the *Data* menu.
5. Select the *Import* button, and accept the default of *generic* scalar file.
6. Use the file browser to find and *Open* the file "rainfall.scl". This file should be found in the tutorial directory.

This file contains a transient (values for multiple times) data set of rainfall intensities.

7. Select the data associated with time step 15.00. You may need to use the scroll bar to locate this time step in the data browser dialog.
8. Select the *Done* button.

16.3 Interpolation from Scattered Data to a Grid

In order to contour or animate the rainfall data set, the values must be interpolated to a grid. To interpolate to a grid do the following:

1. Select the *Bounding Grid* command in the *Data* menu.
2. Specify 25 cells in both the x and y directions, and a mesh centered grid type. All other fields can be left as defaults.
3. Select the *OK* button.
4. Select *OK* for the default z value.

A grid will be created which just bounds the scattered data points.

5. Select *Interpolation Options* from the *Interpolation* menu.
6. Select the *Data Set* button.

7. Turn the *All time steps* toggle on.
8. Select the *Select* button.
9. Turn on the *Truncate values* toggle.

This will keep the interpolation functions from assigning values outside the absolute minimum and maximum of the data set.

10. Select the *OK* button.
11. Select *Interpolation to Grid* from the *Interpolation* menu.
12. Accept the default name for the data set being created for the grid.

WMS will now interpolate the values for each time step from the scatter points to the grid nodes. The progress dialog box informs you of the progress for interpolation.

16.4 Contouring the Grid

The interpolation process created a new transient data set for the grid. In this section options for contouring individual time steps will be explored.

1. Select the *Display Options* command from the *Display* menu.
2. Turn off the *scatter point symbols* and select *OK*.
3. Change to the *2D Grid* module .
4. Select the *Display Options* command from the *Display* menu.
5. Turn off *Nodes* and *Cells* and turn on *contours*.
6. Select the *OK* button.

Linear contours of rainfall intensity are drawn for the grid for time step 15.00. Since time value 15.00 was selected as the active time step in the data browser for scattered data, it is also the active time step for the data set of the grid.

7. Select the *Contour Options* command from the *Display* menu.
8. Select the *Color fill between contours* option.
9. Turn on the *Display Legend* option.
10. Select the *OK* button

The contours are now drawn using a ramp of color intensities. A legend is drawn in the upper left hand portion of the screen, displaying the correlation between color and rainfall intensity.

11. Select the *Contour Options...* command from the *Display* menu.
12. Select the *Color Ramp* tab.
13. Select the *reverse* button.
14. Select the *OK* button.

16.5 Virtual Gages

A virtual gage can be used to show a two dimensional plot of a transient data set for any point in the grid domain.

1. Select *Show Hydrograph Window* from the *Display* menu.
2. Select the *Create Gage* tool .
3. Click on any point within the boundary of the grid (the gage plot will be more interesting if the gage is in a region of the grid where the rainfall intensity is high).
4. Click on another point to create a second gage.

By default a gage is selected immediately after being created (and all other gages are unselected). Therefore the plot currently being displayed corresponds to the second gage. Both gages can be displayed simultaneously using multi-selection.

5. Select the *Select Gage* tool .
6. Select both gages, holding the SHIFT key down as the second gage is selected.

The hydrograph window shows a plot of the selected gages. Several options can be changed to control how gages are displayed. These options are accessible by selecting the *Gages* command or *Gage Plot Manager* command in the *Data* menu. You are encouraged to experiment with these options as much as you want. When you are finished delete the gages by:

7. Select the *Select Gage* tool  if it is not already selected.
8. Select both gages.

9. Choose the DELETE or BACKSPACE key.

16.6 Animations

Animation sequences can be generated from transient data sets. Animations are comprised of a series of images of the current grid where all display options remain the same while the time step is moved ahead.

1. Select the *Film Loop* command from the *Data* menu.

The *Film Loop* dialog comes up over the edit window and stays up while you create and play your film loop animations. Icons are similar to a VCR and can be used to run, stop, or step through an animation sequence.

2. Select the *Setup* button.
3. Change the *Size (% Screen)* to 30%.
4. There are several other options which you may experiment with, but for now select *OK*.

WMS will draw a contoured image of each time step in the data set, saving the image after each one. Once an image for each time step is created, the video buttons and other controls will undim and the animation sequence can be replayed at a much higher refresh speed.

5. Select the *play* button (the VCR-like right arrow button).
6. Experiment with the speed setting.
7. Stop the animation by selecting the stop button (the VCR-like box).
8. Use the *Frame* scroll bar to move the animation forward/backwards in time.
9. If you wish to save the series of images to a file so that they can be replayed later without having to load data files or run through setup, you can use the *Save* button to save an animation loop file. In a subsequent session of *WMS* the *Read* dialog in the *Film Loop* dialog can be used to reload and run the animation. In this case you would not have to read in the grid, data set or run through the time consuming process of setup.

10. Select the *Done* button.

16.7 Mapping Elevations

Any function from a data set can be mapped to the elevations of the grid cells so that the function can be represented as a 3-dimensional surface. This can be done for the current grid and data set by doing the following:

1. Select the *Display Options* from the *Display* menu.
2. Turn on the display of *Cells*.
3. Select *OK*.
4. Select the *Map Elevations* command from the *Data* menu.
5. Select the data set created when interpolating from the scattered data (rainfall_all would have been the default) and once again choose time step 15.0.
6. Select the *Select* button.
7. Select the *Oblique View* macro .
8. Select the *Z Magnification* command from the *View* menu.
9. Set the value to 250 (in order to better visualize the variations in elevations) and select *OK*.

You may wish to rotate  or experiment with other display commands.

16.8 Conclusions

In this tutorial you have learned how scatter points and data sets can be used to interpolate sparse data to a grid for visualization purposes. You should know how grids can be contoured, and how virtual gages, animations, and mapped z values can be used to display transient data.

Hydraulic Calculators in WMS

- Use results from hydrologic models to design and/or analyze
 - Weirs
 - Channels
 - Detention basins
 - Culverts (HY8)
 - Curb and Gutter (FHWA)
- Use calculators to set up hydrologic model input

Weir Calculator

The screenshot shows a window titled "Weir Calculations". It contains several input fields and checkboxes. The "Weir Type" dropdown is set to "Cipolletti". The "Weir Length (ft)" field contains "20.000". The "Weir Crest Elevation (ft)" field contains "3.367". The "Weir Crest Width (ft)" field contains "30.000". There are two checkboxes, both of which are checked. The interface is dark-themed with white text and input fields.

Channel Calculator

Channel Calculations

Trapezoidal

1.00

1.00

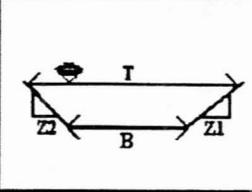
10.00

0.0014

0.0140

30.00

0.85



Detention Basin Calculator

Detention Basin

1000.00

1000.00

20.00

1.00

20.00

17.00

78.00

0.50

0.00

0.00

150.00

Drainage Calculation Tools

Estimated Time: 30 minutes

Several drainage calculation tools are included in WMS to aid in analysis and design of hydraulic structures. These tools include Channel Calculator, a Weir Calculator, a Detention Basin Calculator, a Curb and Gutter Calculator, and an interface to HY-8, a culvert analysis program.

13.1 Channel Calculator

The channel calculator can use information (hydrographs) determined from a hydrologic model, to design a channel capable of supporting a given flow. It may also be used to find stage depth in an existing channel or in a natural cross-section. An example of both of these scenarios will be given in this section. In this section of the tutorial, we will use a HEC-1 simulation file to demonstrate the functionality of the channel calculator.

13.1.1 Using the Channel Calculator with Hydrologic Model Results

In this section of the tutorial, we will use the results from a HEC-1 simulation to demonstrate the functionality of the channel calculator.

1. Select *New* from the *File* menu.
2. Confirm that you want to delete everything.
3. Select the *Hydrologic Modeling* module icon .

4. Select *Open HEC-1 File* from the *HEC-1* menu.
5. Find and *Open* the file named "*redri.hc1*".
6. Select *Open...* from the *Hydrographs* menu.
7. Find and *Open* the file named "*redri.sol.*".
8. Select *Show Hydrograph Window* from the *Display* menu.
9. Select the hydrograph corresponding to the outlet "RED30" in the *Hydrologic Modeling* window.

The hydrograph for the outlet point RED30 is now displayed in the *Hydrograph* window.

Bring up the *Channel Calculations* dialog by going to the *Calculators* menu and selecting *Channels*. Note the channel geometry options on the left side of the dialog. You may want to click through them and note the different parameters required for each. The trapezoidal channel option will be used for this tutorial. To set up the channel calculator follow these steps:

1. Select the *Trapezoidal channel* option.
2. Enter 1.5 in the edit fields for *Side slope 1* and *Side slope 2*.
3. Enter 5.0 in the *Channel width* box.
4. Enter 0.001 for *Longitudinal slope* and 0.02 for *Manning's roughness*.
5. Make sure *Enter Flow* is selected above the channel diagram; note that the maximum flow from the hydrograph selected above appears in the edit field.
6. Select *Calculate*.

The calculated values for several channel parameters will appear on the left side of the dialog. It may be useful to experiment with different channel shapes to determine the channel that would best fit the given circumstances. For example, you may want to try calculating the flow using a rectangular channel with a depth of 12 feet. When done experimenting, select *OK* to close the calculator.

13.1.2 Using the Channel Calculator with Cross-Section Data

In this section, you will learn how to use WMS to create a cross section of a natural channel, then use the Channel Calculator to find stage (depth) in that channel.

The cross section you will create is located immediately downstream from Deer Creek Dam, Utah. To create this cross section:

1. Select *New* from the *File* menu.
2. Select the *Map* module icon .
3. Select *Open* from the *File* menu. Find and open the *deercrk.sup* from the tutorial directory.

This will read in and display a contoured DEM. You will note a red line in the lower right of the DEM representing the location of Deer Creek Dam. By tracing that line with a cross-section arc, you will be able to create a natural channel cross-section.

1. Select *Coverages* from the *Feature Objects* menu.
2. In the *Coverages* dialog, click on the *New* button to create a new coverage (data layer).
3. Enter "*Cross Sections*" in the *Name* box to rename the coverage.
4. Select *Cross-Section* from the *Attribute Set* drop down box.
5. Click on *OK*.

You now have an active cross-section coverage. Any arcs created in this coverage will represent cross sections and will derive station and elevation information from the DEM or TIN in WMS. To create the cross section at the dam:

1. Select the *Create Arc* tool .
2. Create an arc parallel to the red line representing Deer Creek Dam on the DEM. Click to start at one end, click several times (9-10 times) as you proceed to the other end, double-click to end. Your arc should look similar to Figure 13-1.

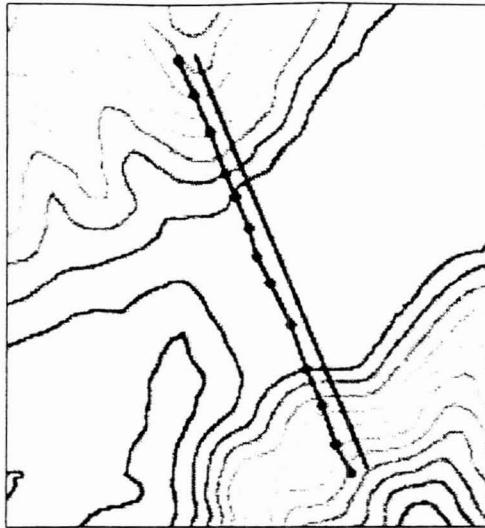


Figure 13-1 Cross section at Deer Creek.

Upon creating a cross section arc, WMS interpolates station and elevation data to the cross section. To view and/or change this information:

1. Select the *Select Arc* tool .
2. Double-click on the cross section arc to bring up the *Cross Section Editor*.

In this dialog, you will see a profile plot of the cross section arc. You will also see the station and elevation for each point (vertex) along the arc. Note also that a Manning's roughness values may be set for each section of the arc.

You need not change any values in this dialog, but you may want to experiment with the *Z-scale* option to view the cross section at a different *Z*-scale.

3. Enter "Deer Creek" in the *Cross section name* box.
4. Click OK.

You are now ready to use the Channel Calculator to analyze the Deer Creek cross section.

1. Select the *Hydrologic Modeling* module icon .
2. Select *Channels...* from the *Calculators* menu.
3. In the *Channel Calculator* dialog, select *Cross Section* from the *Channel type* drop-down box.
4. Make sure that "Deer Creek" is the cross section selected in the *Cross Section* drop-down box.

Note that the cross section is plotted in the window in the lower right of the dialog. This plot may be exaggerated in the Z direction for better visualization if desired.

You will now determine the stage in the channel given the maximum spillway and outlet capacity (12,000 cfs) of Deer Creek Dam.

1. Select the Enter Flow option in the upper right of the dialog. Enter 12000 in the corresponding box.
2. Ensure that English is selected in the Units box.
3. Enter 0.001 in the Longitudinal slope box.
4. Click Calculate.

Note in the window on the left of the dialog, the results of the calculation are shown. The depth in the channel should be around 6 feet, which is acceptable for this channel. Note that a blue line representing the water surface elevation is plotted in the window in the lower right of the dialog. You may want to change the Z-scale to view the plot better.

This concludes the section on using the Channel Calculator. However, you may continue to experiment with different flows, depth, or channel types.

13.2 Weir Calculator

The weir calculator can be used to find the head over a weir at a given flow or the flow over a weir given a head value. The following section will demonstrate how to set up such a calculation. Following on the last example of Deer Creek Dam, you will find out how much head is on the spillway at peak capacity.

13.2.1 Using the Weir Calculator

1. Select *Weirs...* from the *Calculators* menu.
2. On the left side of the dialog, look at the options available and select the *Broad-Crested* weir. Note that in the lower right corner, a default weir coefficient for this weir type appears in the *Weir coefficient* edit field. This value is a generally accepted coefficient for this weir type, but may be edited.
3. Select *Calculate head* in the lower right of the dialog.
4. Enter 12000 in the *Weir Flow* box.
5. Enter 42.0 for the *Weir length*.

6. Select *Calculate*.

Note the answer for the head is located in the lower right corner of the dialog box (about 20 feet). It may be useful to experiment with different weir types to see how they affect the head passing over the weir. For example, try a Cipolletti with the same flow. It may also be of interest to calculate a flow using a given head. Using the same weir type, select *Calculate Flow* and enter head of 10.0. Select *Calculate* again to see the flow that would cause a 10.0 ft head over the weir. Select *OK* to close calculator.

13.3 Detention Basin Calculator

The detention basin calculator may be used as purely a design tool or it may be used to determine the effects of a reservoir in a drainage unit. Storage capacity curves and/or outflow hydrographs may be calculated using this tool. In this example, the detention basin calculator will be used to find the effects of a reservoir created in a drainage unit.

13.3.1 Opening a TIN file

1. Select *New* from the *File* menu.
2. Select the *TINs* module icon .
3. Select *Open* from the *File* menu.
4. Find and *Open* the file named "*aspencal.tin*".

13.3.2 Creating a Reservoir

We will create a reservoir to define detention basin geometry.

1. Select the *Select TIN Vertices* tool .
2. Select the downstream most outlet of the TIN.
3. Select *Create Reservoir* from the *Streams* menu. The *Reservoir* dialog will appear.
4. Enter 6190.0 in the *Water surface elevation* edit field, then click on the *feet* radio button. Note that at the top of the dialog, the current model units are shown to be meters; by clicking on the *feet* radio button, you have indicated that the value you have entered is in feet and that *WMS* will need to convert this value to be consistent with the model units when doing computations. The option to select which units you are entering allows you to work with whichever units you desire.

5. Make sure the *Create reservoir* toggle is on, then select the *Create storage capacity* toggle.
6. Enter 10 for the number of divisions.
7. Select the *English units (acre-ft)* radio button. This will instruct *WMS* to create a storage capacity curve of acre-ft versus feet. The storage capacity units options are meant to allow you to work with units which you are most familiar.
8. Select *OK*.

You will see the Detention Basin Calculator dialog come up with a plot of your storage capacity curve. You will now need to open a hydrograph file to get the inflow to the reservoir.

9. Select *OK* in the *Detention Basin Hydrograph Routing* dialog.

13.3.3 Opening the Hydrograph File

An inflow hydrograph, to be used in conjunction with the detention basin geometry, must also be imported for detention basin calculations.

1. Select the *Tree* module icon .
2. Select *Open* from the *Hydrographs* menu.
3. Find and Open the file named "*aspencal.sol*".
4. Select *Show Hydrograph Window* from the *Display* menu.

Hydrographs will appear on the TIN. Select the hydrograph corresponding to the lowest outlet on the tree, where the reservoir was created. It will appear in the *Hydrograph* window.

13.3.4 Using the Detention Basin Calculator

Now that a reservoir has been created and the hydrograph has been selected, the Detention Basin calculator may be opened again.

1. Select *Detention Basins...* from the *Calculators* menu.

The storage capacity curve should now be plotted in the dialog. To change the format of the plot, click on the *Plot Options* button. You may then format the plot as desired. If a hard copy of the curve is needed, the *Print* button will allow you to print to a file or to a printer.

The next step is to define a discharge relationship for the reservoir:

2. Select the *Define* button from the *Elevation Discharge* area on the right side of the *Detention Basin Hydrograph Routing* dialog.
3. Select the *Discharge Structures* option.
4. Select the *Add weir* button; this will add spillway-type discharge component.
5. Enter 10.0 in the *Weir width* box. (This represents the width of the spillway).
6. Enter 3.367 in the *Weir coefficient* box (typical for a Cipolletti weir).
7. Enter 6185.0 in the *Weir elevation* box.
8. Select the *Add Outlet* button; this will add a low-level outlet discharge.
9. Make sure the outlet is selected in the *Discharge structures* window.
10. Enter 50.0 in the *Outlet area* box.
11. Enter 6100.0 in the *Outlet elevation* box.
12. Click OK.

You should now see an elevation-discharge curve plotted in the Detention Basin calculator. You are now ready to perform the hydrograph routing through the reservoir.

1. Click on the *Compute Outflow Hydrograph* button.

You should now see the routing hydrograph plotted on top of the original hydrograph in the Hydrograph Window. Note that the reservoir has cut the peak off the hydrograph and lengthened it. You may want to experiment with different initial storage values (between 0.0 and 200.0) to see how that might affect the outflow hydrograph.

This concludes the section on the Detention Basin calculator.

13.4 Culvert (HY-8) Calculator

In this section you will learn how to design a culvert using the Federal Highway Administration Culvert Analysis computer program HY8. The following example will guide you through the interface that permits easy data entry, editing and computation.

Problem Statement

Given the following site conditions, find a reinforced concrete box culvert that will pass the 50-year flow rate of 400 cfs with a design headwater elevation of 195 ft and with a maximum discharge of 500 cfs.

The natural stream channel consists of a main channel in rock having a Manning's roughness of 0.03 and overbanks having a roughness of 0.08. The channel is on a 5% slope. A typical cross-section is shown below:

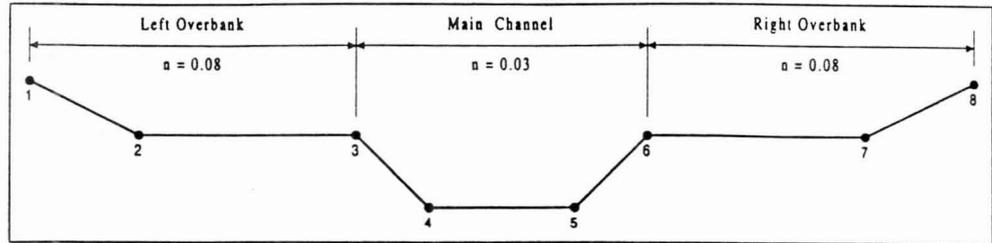


Figure 13-2 Sample cross section.

The cross-section can be described by the following coordinates:

<u>Horizontal Station (ft)</u>	<u>Elevation (ft)</u>
12	180
22	175
32	174.5
34	172.5
39	172.5
41	174.5
51	175
61	180

The roadway profile near the culvert is a sag vertical curve. The road is paved, with an embankment width of 50 ft. The following is a list of coordinates which describe the vertical curve:

<u>Horizontal Station (ft)</u>	<u>Elevation (ft)</u>
0	199.2
100	197.5
200	196.5
300	196.0
360	196.0
400	196.2
500	197.0
600	198.5
720	201.0

A profile of the culvert with the culvert invert data is shown below:

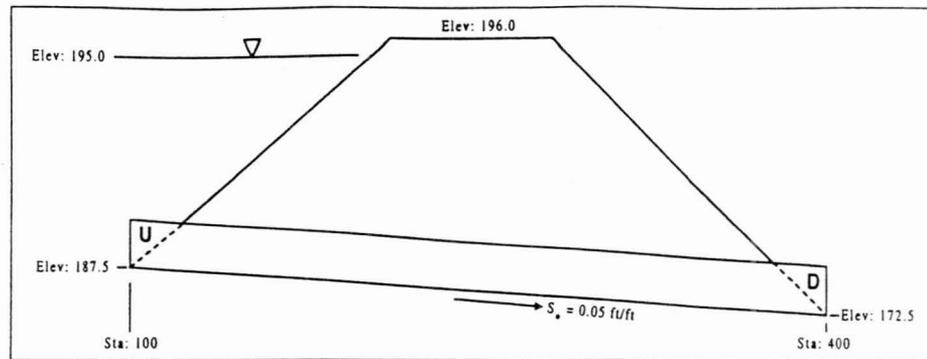


Figure 13-3 Culvert profile.

As an initial size estimate a 5'x5' box concrete culvert assuming a conventional inlet with 1:1 bevels and 45 degree wingwalls.

13.4.1 Entering Culvert Information

1. Select the *HY8 Culvert Analysis* command from the *Calculators* menu. This will bring up the HY8 Culvert Analysis dialog.
2. Change the *Discharge data*, minimum to 0.0, maximum to 500 and design to 400.
3. Click on the *Create/Edit culvert data* button. This will bring up the HY8 Culvert Data dialog.
4. Make sure that the *Culvert invert data* option is selected.
5. Enter culvert data for upstream. Change *Station* to 100 and *Elevation* to 187.5.
6. Enter culvert data for downstream. Change *Station* to 400 and *Elevation* to 172.5.
7. Choose *Box-Concrete* from *Culvert Shape* selection list.
8. Change *Span* to 5 and *Rise* to 5.
9. Accept the default *Number of barrels* i.e. 1.
10. Accept the default *Manning's n*, which would be 0.012.
11. Click on the *Inlet data* button. This will bring up the HY8 Culvert Inlet dialog.
12. Choose *conventional inlet type*.
13. Choose 1:1 bevels with 45 degree wingwalls from the *Inlet Condition/Edge* selection list.

14. Click on the *OK* button to close the HY8 Culvert Inlet dialog.
15. Click on the *OK* button to close the HY8 Culvert Data dialog.

13.4.2 Entering the Tailwater Channel Information

1. Click on the *Create/Edit tailwater data* button. This will bring up the HY8 Tailwater Data dialog.
2. Select the *Irregular Channel* rating curve option.
3. Click on the *Irregular Channel* button to specify the irregular channel properties. This will bring up the HY8 Irregular Channel dialog.
4. Change *channel slope* to 0.05.
5. Change *number of cross-section coordinates* to 8.
6. Choose *Lt_Rt* over banks and main channel as the *subchannel option*.
7. Change *left overbank* coordinate to 3.
8. Change *right overbank* coordinate to 6.
9. Change Manning's roughness for *left* section to 0.08.
10. Change Manning's roughness for *main* section to 0.03.
11. Change Manning's roughness for *right* section to 0.08.
12. Click on the *Define/Edit Cross-Section* button to specify the irregular channel geometry. This will bring up the Cross Section Editor.
13. Enter 8 pairs of station-elevation values. The values are (12,180), (22,175), (32,174.5), (34,172.5), (39,172.5), (41,174.5), (51, 175), and (61, 180).
14. Click on the *OK* button to close the Cross Section Editor.
15. Click the *View rating curve data* button to compute the rating curve and display the information.
16. Click the *Save tailwater file* button to bring up the file manager. Specify *hy801.tw* as the file name and save the file.
17. Click on the *OK* button to close the *HY8 Irregular Channel* dialog.
18. Click on the *OK* button to close the *HY8 Tailwater Data* dialog.

13.4.3 Entering the Roadway Information

1. Click on the *Create/Edit roadway data* button. This will bring up the HY8 Roadway Data dialog.
2. Choose irregular *profile* option.
3. Change the *number of coordinates* to 9.
4. Change the *roadway top width* to 50.
5. Click on the *Profile* button to specify the irregular roadway profile. This will bring up the Cross Section Editor.
6. Enter 9 pairs of station-elevation values. The values are (0,199.2), (100,197.5), (200,196.5), (300,196.0), (360,196.0), (400,196.2), (500, 197), (600, 198.5) and (720, 201.0).
7. Click on the *OK* button to close the Cross Section Editor.
8. Choose *paved roadway surface*.
9. Click on the *OK* button to close the HY8 Roadway Data dialog.

13.4.4 Performing Culvert Analysis

The data already entered in the previous sections must be saved in order to perform any computation.

1. Click on the *Save input file* button. This will bring up the file manager. Specify hy801.inp as the file name and save.
2. Choose *Full Flow* option for outlet control.
3. Choose *Single culvert output table* for computation.
4. Click on the *Compute* button.
5. Click on the *View* button to display the result.

In the result, find the headwater elevation at 400 cfs and compare with the design value of 195 feet. It may turn out that the 5'x5' culvert is inadequate for the site conditions. You are encouraged to test with different sizes, suppose a 6'x6' culvert or larger in order to determine the required culvert size.

13.5 Conclusions

This tutorial has demonstrated the basic functionality of the drainage calculators in *WMS*. Design and/or modeling of channels, weirs, detention basins, and culverts is possible using these calculators.



Flood Plain Delineation

- Easily adapted to a TIN model
- Flood plain delineation is somewhat of a misfit
 - calculations usually come from a surface water program such as WSPRO or HEC-2
 - resolution of surface water and hydrologic models are different
 - surface water doesn't include much of the flood plain
 - hydrologic doesn't include a good definition of channels

How Does WMS Delineate Flood Plains?

- TIN
- Stream network
- Define stage values at positions along the stream
- Flood plain determined from a combination of the channel (overall topography) slope and the slope of the water surface elevation

Steps to Delineate Flood Plains

1. Create a TIN with a stream network
2. Edit stage at locations along stream
3. Interpolate stage between known values
4. Delineate Flood Plain

Flood Plain Delineation

Estimated Time: 25 minutes.

In addition to drainage basin analysis, *WMS* also contains flood plain delineation features that can be used to define flood plain boundaries on any TIN. Flood plains are delineated from stage values that are entered at various locations on a TIN. Stage values are defined as the difference between the flood water surface elevation and the normal TIN elevation.

This tutorial will guide you through the steps for entering, saving, and reading stage values from a file. It will also show how to delineate flood plains from stage values, adjust the flood plain delineation parameters, and save or read stage values from a file

14.1 Delineating Flood Plains

In this tutorial you will learn the steps for entering stage values at points on a TIN and then how to delineate the flood plain boundary represented by these stage values. The topics covered include the following:

- Editing the stage value at TIN vertices.
- Reading stage values from a file.
- Using the Interpolate Stage command.
- Delineating a flood plain from TIN stage values.
- Comparing stream-based and flat flood plain delineation.

- Changing the flood plain delineation parameters.
- Saving and reading stage values to and from files.

14.2 Editing Stage Values

To begin editing stage values, do the following:

1. Select the *TINs* module icon .
2. Select *Open* from the *File* menu.
3. Locate and select the file called "*ding.tin*". It is located in the tutorial directory by default.
4. Select the *Display Options* command from the *Display* menu.
5. Be sure that *Triangles*, *Vertices*, and *Boundaries* are toggled on, and everything else is off.
6. Select the *Drainage* tab.
7. Make sure that the display of *Outlets* and *Stream networks* are on.
8. Select *OK*.

With the TIN file read in and the display of vertices and stream networks on, you are ready to define stage values on the TIN. Stage values can be defined by selecting vertices on the TIN and entering a stage value, as shown by the following example:

1. Click on the *Select Vertices*  tool.
2. Click on the outlet of the stream network. By default, outlets are displayed as a black circle with a yellow center.
3. Select the *Edit stage* command from the *Flood* menu.
4. Enter the value 4.0 in the *Enter new vertex stage* field.
5. Select *OK*.
6. While holding the SHIFT key down, select four or five additional vertices on the stream.
7. Select the *Edit stage* command from the *Flood* menu.
8. Enter the value 3.5 in the *Enter new vertex stage* field.

9. Select *OK*.

For each stream node that was given a stage value, a vertical red bar will be drawn with a length proportional to the stage value at that node.

14.3 Interpolating Stage Values

When stage values are defined at locations on a stream, stage values for other locations on the stream can be linearly interpolated. By editing the stage values at the outlet and branching nodes, we have entered stage values for points that are on the stream network.

1. Select the *Interpolate Stage* command from the *Flood* menu.

The interpolated stage values for stream nodes that lie between stream nodes where stage was defined will now be displayed with a pink bar. Note that this interpolation can only be applied to stream nodes that lie along a stream between two stream nodes where stage has been defined.

14.4 Flood Display Options

A display option can be set to view both fixed and interpolated stage values by following these steps:

1. Select the *Display Options* command from the *Display* menu.
2. Select the *Flood* tab.
3. Turn on the display of *fixed* and *interpolated stage* values.
4. Click on the *OK* button.

This allows you to check the values that will be used in the delineation of the flood plain. If any value is incorrect, that node can be selected and the stage value edited as described above.

With display of both fixed and interpolated values, it is often hard to read all of the stage values since the values overlap each other. The zoom tool can be used to inspect the values on the stream a portion at a time. Additionally, viewing the display of fixed and interpolated values separately may eliminate the overlapping problem.

The flood display options dialog box allows many additional changes to be made in the display of the stage values, including line thickness, style and color. This can be done by clicking on the color box next to each display option. The relative height of the stage bars in terms of # of pixels/unit of stage value can also be changed. You are encouraged to experiment with these

parameters until you feel comfortable with the different display options that are available to you.

14.5 Delineating Flood Plains

Once stage values have been defined on a TIN, you are ready to delineate the flood plain. To delineate the flood plain, do the following:

1. Select the *Display Options* command from the *Display* menu.
2. Select the *Flood* tab.
3. Turn off the display of *fixed* and *interpolated stage* values.
4. Select *OK*.
5. Select the *Delineate Flood* command from the *Flood* menu.

The flood plain was delineated using the stage values found at the stream nodes where stage values were input, or stages that were interpolated along the stream. Using streams to help define the flood plain gives the best results. In order to be able to perform stream-based flood plain delineation, WMS must have at least two stream nodes with defined stage values. If there are vertices on a stream without stage values defined and no downstream stream vertex has a defined stage value, no stage values for these stream nodes can be interpolated. The closest upstream stage value is assumed to represent the water surface elevation for the downstream remainder of the TIN. To see an example of this effect, do the following:

1. Select on the *Select Vertices*  tool.
2. Click on the outlet of the TIN (black circle with yellow center).
3. Select the *Remove Stage* command from the *Flood* menu.
4. Select the *Delineate Flood* command from the *Flood* menu.

The flood plain will look similar to the previous flood plain, except for the portion of the stream near the outlet are omitted.

14.6 Reading and Saving Stage Files

Stage values can be saved to and read from files.

1. Select the *Remove Stage* command from the *Flood* menu. Confirm that all stage values will be removed.

2. Select the *Read Stage* command from the *Flood* menu. Locate and *Open* the file called "*ding.stg*".

Stage values that you define can be saved to a file using the *Save Stage* command in the *Flood* menu.

14.7 Reading a Super File with Pre-Defined Flood Files

The *blackriv.sup* file is a *WMS* super file that contains a *TIN* file, a stage file, and a map file. The *TIN* is a portion of a watershed in the western Adirondack Mountains. The stage file contains several key locations along the Black River for which stage is known during a flood event. This file can be read in by doing the following:

1. Select the *New* command from the *File* menu.
2. Confirm that you want all data deleted.
3. Select the *Open* command from the *File* menu.
4. Using the file browser find and *Open* the "*blackriv.sup*" file.

This model shows a basin delineation of the Black River in northwestern New York on the western slopes of the Adirondacks. The polygon identified as the flats is subject to flooding during spring runoff. A separate, more detailed *TIN* of the flats was created and used for flood plain delineation. To see this *TIN*:

1. Select the *New* command from the *File* menu.
2. Confirm that you want all data deleted.
3. Select the *Open* command from the *File* menu.
4. Using the file browser find and *Open* in the "*flood.sup*" file.

In addition to the *TIN*, a map file is read in which shows the main channel as well as major roads and railroads in the flood plain. Once the flood plain has been delineated you will be able to see the portions of the road and railroad endangered by the high waters.

1. Select the *Delineate Flood* command from the *Flood* menu.

14.8 Display Options

So far, only an outline of the flood plain has been shown. Besides drawing the boundary you can also have the flood plain filled with a solid color or contoured according to depth.

1. Select the *Display Options* command from the *Display* menu.
2. Select the *Flood* tab.
3. Turn on the *Flood plain color filled* option.
4. Select *OK*.

Besides showing the extents of the flood plain, you can also display it using contours.

1. Select the *Display Options* command from *Display* menu.
2. Select the *Flood* tab.
3. Turn off the *Flood plain color filled* option.
4. Turn off the *Fixed* and *Interpolated stage* options.
5. Turn on the display option for *Flood plain contours*.
6. Select the *Contours...* button to access the *Contour Options* dialog.
7. Make sure the *Color fill between contours* is turned on.
8. Select *OK*.
9. Select *OK*.

14.9 Conclusions

This concludes the Flood plain delineation tutorial. You are encouraged to experiment with all of the commands explained in this section until you feel comfortable with the tools available to you for delineating flood plains.



DEM - DIGITAL ELEVATION MODULES
OR MODELS

3 WAYS TO DEMONSTRATE

CREATE OWN

DEM

TIN