
Watershed Modeling System

WMS v6.1

TUTORIALS

WMS 6.1

Copyright © 1999 Brigham Young University - Environmental Modeling
Research Laboratory

All Rights Reserved

Unauthorized duplication of the *WMS* software or documentation is strictly
prohibited.

THE BRIGHAM YOUNG UNIVERSITY ENVIRONMENTAL MODELING
RESEARCH LABORATORY MAKES NO WARRANTIES EITHER
EXPRESS OR IMPLIED REGARDING THE PROGRAM *WMS* AND ITS
FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF
THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *WMS* is a product of the Environmental Modeling Research
Laboratory of Brigham Young University. For more information about this
software and related products, contact the EMRL at:

Environmental Modeling Research Laboratory
Rm. 242, Clyde Building
Brigham Young University
Provo, Utah 84602
Tel.: (801) 378-2812
Fax: (801) 378-2478
e-mail: wms@byu.edu
WWW: <http://www.emrl.byu.edu/wms.html>

For technical support, contact your *WMS* reseller.

TABLE OF CONTENTS

1	INTRODUCTION	1-1
1.1	SUGGESTED ORDER OF COMPLETION.....	1-2
1.2	DEMO VS. NORMAL MODE.....	1-2
2	BASIC FEATURE OBJECT MANIPULATION	2-1
2.1	FEATURE OBJECT DEFINITIONS	2-1
2.2	DISPLAY AND MANIPULATION OF COVERAGES	2-2
2.3	USING TIFF IMAGES TO CREATE FEATURE OBJECTS	2-5
2.3.1	<i>Importing and Registering a TIFF Image</i>	2-5
2.3.2	<i>Displaying and Manipulating Images</i>	2-7
2.3.3	<i>Using an Image to Define Feature Objects</i>	2-7
2.3.4	<i>Saving a File and Deleting the Feature Objects</i>	2-8
2.4	CONCLUSION	2-9
3	ADVANCED FEATURE OBJECT MANIPULATION	3-1
3.1	CLEANING AND EDITING FEATURE OBJECTS.....	3-1
3.1.1	<i>Importing and Cleaning Feature Object Data</i>	3-1
3.1.2	<i>Creating Feature Objects from Images</i>	3-5
3.1.3	<i>Building Polygons</i>	3-8
3.2	IMPORTING FEATURE OBJECTS FROM ARC/INFO (ARCVIEW) SHAPE FILES	3-10
3.2.1	<i>Reading a WMS Map Drainage File</i>	3-10
3.2.2	<i>Importing a Soil Type Shape File</i>	3-10
3.2.3	<i>Importing a Land Use Shape File</i>	3-12
3.2.4	<i>Assigning Attributes to Feature Objects</i>	3-12
3.3	CREATING FEATURE OBJECTS FROM DXF DATA	3-15
3.3.1	<i>Creating a Boundary Polygon</i>	3-15
3.3.2	<i>Importing a DXF File</i>	3-15
3.3.3	<i>Digitizing Feature Objects From DXF Data</i>	3-16
3.3.4	<i>Converting DXF Data to Feature Objects</i>	3-16
3.3.5	<i>Saving the Feature Objects to a Map File</i>	3-17
3.4	CONCLUSION	3-17
4	HYDROLOGIC MODELS FROM FEATURE OBJECTS	4-1
4.1	LAUNCHING WMS	4-1
4.2	CREATING A WATERSHED FROM SCRATCH WITH FEATURE OBJECTS	4-2
4.2.1	<i>Creating Basin Boundaries</i>	4-2
4.2.2	<i>Creating the Stream Network</i>	4-3
4.2.3	<i>Building Polygons</i>	4-4
4.2.4	<i>Updating Geometric Parameters</i>	4-5
4.3	IMPORTING GIS DATA TO CREATE A WATERSHED.....	4-5
4.3.1	<i>Importing Shape Files</i>	4-6
4.3.2	<i>Running HEC-1 with Data Defined in a GIS</i>	4-7
4.4	CREATING A WATERSHED WITH A GIS STREAM AND BASIN LAYERS	4-9
4.4.1	<i>Importing the Basin and Stream Layer</i>	4-9

4.4.2	Converting Generic Arcs to Stream Arcs	4-10
4.4.3	Converting Generic Polygons to Basin Boundary Polygons	4-12
4.5	CREATING A WATERSHED WITH A GIS BASIN LAYER.....	4-13
4.5.1	Importing the Basin Layer	4-13
4.5.2	Creating a Stream Network.....	4-13
4.5.3	Converting Polygons to Basins	4-15
4.6	CREATING A WATERSHED WITH A GIS STREAM LAYER	4-15
4.6.1	Importing the Stream Layer	4-16
4.6.2	Creating Basin Polygons.....	4-16
4.7	CONCLUSIONS	4-18
5	DEM BASICS	5-1
5.1	IMPORTING A USGS DEM	5-1
5.1.1	Opening the File.....	5-2
5.2	SELECTING AN ACTIVE AREA	5-2
5.3	VIEWING THE DEM	5-4
5.3.1	Displaying Contours	5-4
5.3.2	Display Speed.....	5-4
5.3.3	Displaying a Hill Shade View	5-5
5.4	IMPORTING ASCII GRID FILES	5-6
5.4.1	Importing an ARC/INFO ASCII Grid.....	5-6
5.5	CONCLUSION	5-7
6	WATERSHED DELINEATION FROM DEMS	6-1
6.1	PREPARING THE DEM	6-2
6.1.1	Importing a USGS DEM	6-2
6.1.2	Clipping a Region for Analysis	6-2
6.2	DELINEATING STREAM NETWORKS AND WATERSHED BOUNDARIES	6-3
6.2.1	Computing Flow Directions	6-3
6.2.2	Drawing Flow Paths	6-4
6.2.3	Computing Flow Accumulations	6-4
6.2.4	Converting Raster Streams to Stream Feature Arcs	6-5
6.2.5	Delineating Watershed and Sub-basin Boundaries.....	6-6
6.2.6	Converting Raster Basin Boundaries to Feature Polygons	6-7
6.2.7	Computing Basin Geometric Parameters.....	6-7
6.2.8	Hill Shade With Stream and Watershed Boundaries.....	6-8
6.3	CONCLUSIONS	6-8
7	TIN BASICS	7-1
7.1	READING VERTEX DATA FILES INTO <i>WMS</i>	7-1
7.2	TRIANGULATING VERTICES TO MAKE A TIN	7-2
7.3	CHANGING THE VIEWING PARAMETERS.....	7-2
7.4	DISPLAYING CONTOURS ON A TIN	7-3
7.5	CREATING A SHADED IMAGE OF YOUR TIN	7-5
7.6	MAPPING AN IMAGE TO A TIN.....	7-6
7.7	ADDING BREAKLINES TO PRESERVE GEOMETRIC FEATURES.....	7-7
7.8	SWAPPING TRIANGLE EDGES	7-8
7.9	SELECTING AND DELETING VERTICES ON A TIN.....	7-8
7.10	DRAGGING VERTICES ON A TIN	7-9
7.11	INSERTING VERTICES ON A TIN.....	7-10
7.12	CHANGING DISPLAY ATTRIBUTES.....	7-11
7.13	CONCLUSION	7-12
8	WATERSHED DELINEATION FROM TINS.....	8-1

8.1	READING A REGISTERED TIFF IMAGE	8-1
8.2	READING IN A MAP FILE	8-2
8.3	IMPORTING A BACKGROUND ELEVATION MAP	8-2
8.4	CREATING A TIN AND DEFINING A WATERSHED	8-3
8.4.1	<i>Redistributing Vertices</i>	8-3
8.4.2	<i>Creating a TIN</i>	8-4
8.5	WATERSHED DELINEATION	8-4
8.5.1	<i>Eliminating Pits and Defining Boundaries</i>	8-5
8.5.2	<i>Correcting Split Flow Problems</i>	8-5
8.5.3	<i>Refining Boundaries and Eliminating Exterior Triangles</i>	8-5
8.5.4	<i>Computing Basin Geometric Parameters</i>	8-6
8.5.5	<i>Adding a New Stream</i>	8-6
8.5.6	<i>Creating Additional Outlets</i>	8-8
8.5.7	<i>Merging and Splitting Basins</i>	8-9
8.5.8	<i>Deleting an Existing Outlet</i>	8-9
8.5.9	<i>Basin Label Text</i>	8-10
8.6	ADDING ANNOTATIONS TO A PROJECT	8-10
8.7	CONCLUSIONS	8-12
9	HEC-1 INTERFACE	9-1
9.1	OBJECTIVES	9-1
9.2	READING THE DATA FILES	9-2
9.3	COMPUTING GEOMETRIC ATTRIBUTES	9-3
9.4	ENTERING GAGES	9-3
9.5	DEFINING JOB CONTROL PARAMETERS	9-5
9.6	DEFINING BASIN PARAMETERS	9-5
9.6.1	<i>Loss Method</i>	9-6
9.6.2	<i>Unit Hydrograph Method</i>	9-6
9.6.3	<i>Precipitation</i>	9-7
9.7	DEFINING ROUTING DATA	9-7
9.8	COMPUTING COMPOSITE CURVE NUMBERS	9-8
9.9	RUNNING AND SAVING HEC-1 FILES	9-9
9.10	DISPLAYING HYDROGRAPH RESULTS	9-10
9.11	CONCLUSION	9-10
10	TIME OF CONCENTRATION CALCULATIONS	10-1
10.1	READING A TIN FILE	10-2
10.2	DEFINING FLOW PATH ARCS	10-2
10.3	ASSIGNING EQUATIONS TO TIME COMPUTATION ARCS	10-5
10.4	USING THE TIME COMPUTATION ARCS TO COMPUTE TIME OF CONCENTRATION FOR A TR-55 SIMULATION	10-7
10.5	USING THE TIME COMPUTATION ARCS TO COMPUTE THE TRAVEL TIME BETWEEN OUTLET POINTS .	10-10
10.6	MORE TR-55	10-10
10.7	CONCLUSIONS	10-11
11	NATIONAL FLOOD FREQUENCY PROGRAM (NFF) INTERFACE	11-1
11.1	READING A TIN FILE	11-2
11.2	PREPARE THE TIN TO USE WITH NFF	11-2
11.3	RUNNING AN NFF SIMULATION	11-2
11.3.1	<i>Choosing a State</i>	11-3
11.3.2	<i>Defining Regions</i>	11-3
11.3.3	<i>Assigning Areas to Selected Regions</i>	11-4

11.3.4	Defining Regression Variables	11-4
11.3.5	Computing Peak Discharges	11-4
11.3.6	Exporting Flow Data	11-4
11.3.7	Computing a Hydrograph	11-5
11.3.8	Saving and Reading Simulations	11-6
11.3.9	Using the National Urban Equations Defined in NFF	11-6
11.3.10	Using Individual State Urban Equations Defined in NFF	11-7
11.4	CONCLUSIONS	11-7
12	RATIONAL METHOD INTERFACE.....	12-1
12.1	READING IN TERRAIN DATA	12-1
12.2	CREATING A TIN FOR BASIN DELINEATION.....	12-2
12.2.1	Opening a DXF file.....	12-2
12.2.2	Converting DXF Data to Feature Objects.....	12-2
12.2.3	Preparing the Feature Objects to Create a TIN	12-2
12.2.4	Redistributing Arc Vertices.....	12-4
12.2.5	Creating the TIN.....	12-4
12.3	DEFINING CATCHMENT AREAS.....	12-4
12.3.1	Adding Outlets and Assigning Triangles to Basins.....	12-4
12.3.2	Merging Catchment Areas	12-6
12.3.3	Defining Basin Data	12-7
12.4	RUNNING A RATIONAL METHOD SIMULATION	12-7
12.4.1	Defining the Runoff Coefficients and Time of Concentration.....	12-8
12.4.2	Defining the Rainfall Intensity (i)	12-9
12.4.3	Defining Hydrographs	12-11
12.4.4	Combining Runoff from Multiple Basins	12-11
12.4.5	Defining Rainfall Intensities at the Outlet to Compute Runoff Hydrographs	12-12
12.5	ADDING A DETENTION BASIN	12-13
12.6	CONCLUSIONS	12-15
13	DRAINAGE CALCULATION TOOLS.....	13-1
13.1	CHANNEL CALCULATOR	13-1
13.1.1	Using the Channel Calculator with Hydrologic Model Results.....	13-1
13.1.2	Using the Channel Calculator with Cross-Section Data	13-2
13.2	WEIR CALCULATOR.....	13-5
13.2.1	Using the Weir Calculator	13-5
13.3	DETENTION BASIN CALCULATOR	13-6
13.3.1	Opening a TIN file	13-6
13.3.2	Creating a Reservoir.....	13-6
13.3.3	Opening the Hydrograph File.....	13-7
13.3.4	Using the Detention Basin Calculator.....	13-8
13.4	CULVERT (HY-8) CALCULATOR.....	13-9
13.4.1	Entering Culvert Information	13-10
13.4.2	Entering the Tailwater Channel Information.....	13-11
13.4.3	Entering the Roadway Information.....	13-12
13.4.4	Performing Culvert Analysis.....	13-12
13.5	CONCLUSIONS	13-13
14	FLOODPLAIN DELINEATION.....	14-1
14.1	DELINEATING FLOOD PLAINS.....	14-1
14.2	EDITING STAGE VALUES.....	14-2
14.3	INTERPOLATING STAGE VALUES.....	14-3
14.4	FLOOD DISPLAY OPTIONS	14-3

14.5	DELINEATING FLOOD PLAINS	14-4
14.6	READING AND SAVING STAGE FILES.....	14-4
14.7	READING A SUPER FILE WITH PRE-DEFINED FLOOD FILES.....	14-5
14.8	DISPLAY OPTIONS	14-6
14.9	CONCLUSIONS	14-6
15	COMPUTING CURVE NUMBERS	15-1
15.1	COMPUTING COMPOSITE CURVE NUMBERS	15-1
15.2	READING A LAND USE GRID.....	15-2
15.3	READING IN A SOIL TYPE GRID.....	15-3
15.4	READING IN AND DEFINING SUB-BASINS IN A DRAINAGE COVERAGE	15-3
15.5	IMPORTING A LAND USE TABLE	15-3
15.6	COMPUTING CURVE NUMBERS FOR A DRAINAGE COVERAGE	15-4
15.7	READING IN A LAND USE POLYGON SHAPEFILE.....	15-4
15.8	COMPUTING CURVE NUMBERS FOR A DRAINAGE COVERAGE USING A LAND USE POLYGON COVERAGE AND A SOIL TYPE GRID.....	15-5
15.9	READING IN A SOIL TYPE POLYGON SHAPEFILE.....	15-6
15.10	READING IN A TIN WITH DEFINED SUB-BASINS	15-7
15.11	COMPUTING CURVE NUMBERS FOR A TIN USING A LAND USE POLYGON COVERAGE AND A SOIL TYPE POLYGON COVERAGE.....	15-7
15.12	COMPUTING COMPOSITE RUNOFF COEFFICIENTS FOR A DRAINAGE COVERAGE USING A SOIL TYPE GRID	15-8
15.13	CONCLUSIONS.....	15-9
16	SCATTERED DATA AND 2D GRIDS.....	16-1
16.1	OBJECTIVES.....	16-1
16.2	READING THE DATA FILE	16-2
16.3	INTERPOLATION FROM SCATTERED DATA TO A GRID	16-2
16.4	CONTOURING THE GRID	16-3
16.5	VIRTUAL GAGES.....	16-4
16.6	ANIMATIONS	16-5
16.7	MAPPING ELEVATIONS	16-6
16.8	CONCLUSIONS	16-6
17	TIN EDITING	17-1
17.1	OBJECTIVES.....	17-2
17.2	READING THE DATA FILE	17-2
17.3	TRIANGULATING THE DATA POINTS	17-3
17.4	REMOVING THIN TRIANGLES ON THE BOUNDARY	17-3
17.5	SWAPPING EDGES.....	17-4
17.6	ADDING BREAKLINES.....	17-5
17.7	REMOVING FALSE DAMS.....	17-6
17.8	INSERTING ADDITIONAL POINTS.....	17-8
17.9	AUTOMATED EDITING TECHNIQUES	17-9
17.10	CREATING STREAMS MANUALLY	17-10
17.11	CONCLUSIONS.....	17-12
18	CREATING TOPOLOGIC MODELS FOR HEC-1	18-1
18.1	THE TOPOLOGIC TREE REPRESENTATION.....	18-2
18.2	CREATING THE TREE	18-5
18.3	CHANGING HYDROGRAPH STATION NAMES	18-5
18.4	CREATING THE TREE WITH FEATURE OBJECTS.....	18-7
18.5	JOB CONTROL DATA	18-9

18.6	DEFINING GAGES.....	18-9
18.7	DEFINING BASIN PARAMETERS.....	18-11
18.8	DEFINING ROUTING PARAMETERS.....	18-13
18.9	DEFINING DIVERSION DATA.....	18-15
18.10	DEFINING A RESERVOIR.....	18-15
18.11	RUNNING AND SAVING HEC-1 FILES.....	18-16
18.12	DISPLAYING HYDROGRAPH RESULTS.....	18-17
19	CREATING TOPOLOGIC MODELS FOR TR-20.....	19-1
19.1	THE TOPOLOGIC TREE REPRESENTATION.....	19-2
19.2	CREATING THE TREE.....	19-3
19.3	CHANGING HYDROGRAPH STATION NAMES.....	19-4
19.4	CREATING THE TREE WITH FEATURE OBJECTS.....	19-6
19.5	JOB CONTROL DATA.....	19-7
19.6	DEFINING BASIN PARAMETERS.....	19-7
19.7	DEFINING ROUTING PARAMETERS.....	19-8
19.8	DEFINING DIVERSION DATA.....	19-8
19.9	DEFINING A RESERVOIR.....	19-9
19.10	RUNNING AND SAVING TR-20 FILES.....	19-10
19.11	DISPLAYING HYDROGRAPH RESULTS.....	19-11
20	HSPF INTERFACE.....	20-1
20.1	OPENING THE WATERSHED AND INITIALIZING THE HSPF MODEL.....	20-1
20.2	IMPORTING LAND USE AND SEGMENTING THE WATERSHED.....	20-3
20.3	AGGREGATING SEGMENTS.....	20-5
20.4	DEFINING LAND SEGMENT PARAMETERS.....	20-6
20.5	DEFINING REACH SEGMENT PARAMETERS.....	20-9
20.6	CREATING MASS LINKS.....	20-11
20.7	SAVING AND RUNNING HSPF SIMULATION.....	20-12
21	COORDINATE CONVERSIONS.....	21-1
21.1	BACKGROUND INFORMATION ON COORDINATE SYSTEMS.....	21-1
21.1.1	<i>UTM Coordinate System.....</i>	<i>21-2</i>
21.1.2	<i>State Plane Coordinate System.....</i>	<i>21-2</i>
21.1.3	<i>Typical Problems Requiring Coordinate Transformation.....</i>	<i>21-3</i>
21.1.4	<i>The Current Coordinate System.....</i>	<i>21-4</i>
21.1.5	<i>Proper Order of Reading Data.....</i>	<i>21-4</i>
21.2	CONVERTING DATA FROM ONE COORDINATE SYSTEM TO ANOTHER.....	21-5
21.2.1	<i>Setting the Current Coordinate System.....</i>	<i>21-5</i>
21.2.2	<i>Converting from UTM to State Plane.....</i>	<i>21-6</i>
21.3	CONVERTING DATA TO CONSISTENT UTM ZONES.....	21-8
21.4	CONVERTING DATA FROM MULTIPLE COORDINATE SYSTEMS.....	21-12
21.5	CONCLUSION.....	21-14

Introduction

/* The first time you say how to run WMS is in section 4.1. It should either be in the beginning of every section, or moved to this first chapter.

Chapters 5-19 have a different format than chapters 1-4. Maybe add overview, objectives, and estimated times to chapters 1-4.

In some of the tutorials, it says “if you are continuing from another tutorial, select New . . . “ (see 12.1). Either put this at the beginning of every chapter or only at the beginning. Sometimes when consecutive chapters talk about similar processes it’s unclear whether you should start over.

14.2, 2nd list, step 2 – it says outlets are displayed as a yellow circle with a black center. Either include this earlier, like the first time you talk about an outlet, or maybe make a legend of symbols at the beginning. */

This document contains tutorials for the Watershed Modeling System (*WMS*). Each tutorial provides training on a specific component of *WMS*. Since the *WMS* interface contains a large number of options and commands, you are strongly encouraged to complete the tutorials before attempting to use *WMS* on a routine basis.

In addition to this document, the *WMS Help File* also describes the *WMS* interface. Typically, the most effective approach to learning *WMS* is to complete the tutorials and use the help file when encountering a portion of *WMS* that is unclear.

1.1 Suggested Order Of Completion

Tutorials 2-7 cover the basic data structures used in WMS (GIS vector data or feature objects, DEMs, and TINs) and how watersheds can be delineated to set up hydrologic models from them. Tutorials 2-3 cover feature objects, 4-5 DEMs, and 6-7 TINs. After going through the basics, tutorials 9 and 14 cover supporting hydrologic calculations such as curve number generation from land use and soil data layers (14) and time of concentration (or lag time) computations from computed geometric values. Tutorials 8 and 10-12 cover some of the supported models, and the rest of the chapters cover advanced tools and miscellaneous topics.

It is unlikely that you will need to work through all of the tutorials, but we recommend that you go through 2-7 and then the remaining tutorials, which cover model interfaces and advanced tools, can be completed according to the interfaces you use and as you have time.

1.2 Demo vs. Normal Mode

The interface for *WMS* is divided into six separate modules. Some of the modules contain interfaces to models such as *HEC-1*. Such interfaces are typically contained within a single menu. Since some users may not require all of the modules or model interfaces provided in *WMS*, modules and model interfaces can be licensed individually. The modules and interfaces that have been licensed are enabled using the *Register* command in the *File* menu. The icons for the unlicensed modules or the menus for model interfaces are dimmed and cannot be accessed.

You can complete all of the tutorials if you have licensed the digital terrain and map modules. However, if you have only licensed one (or more) of the hydrologic models separate from the digital terrain and map modules, there will be some tutorials which cannot be completed. By switching to *Demo* mode you will be able to run all of the tutorials, but printing and saving will be disabled. This is done by selecting the *Demo Version On* command in the *File* menu. When you wish to return to normal mode, select the *Demo Version Off* command in the *File* menu. If you are using a demo copy of *WMS* and you have not licensed any modules, you will be in demo mode by default and will not be able to switch to normal mode.

It is suggested that you start WMS new at the beginning of each tutorial. If you continue from one tutorial to another without quitting, then data, display options, and other WMS settings may not be in sync with the tutorial instructions causing the exercise to become confusing.

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 format. 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).

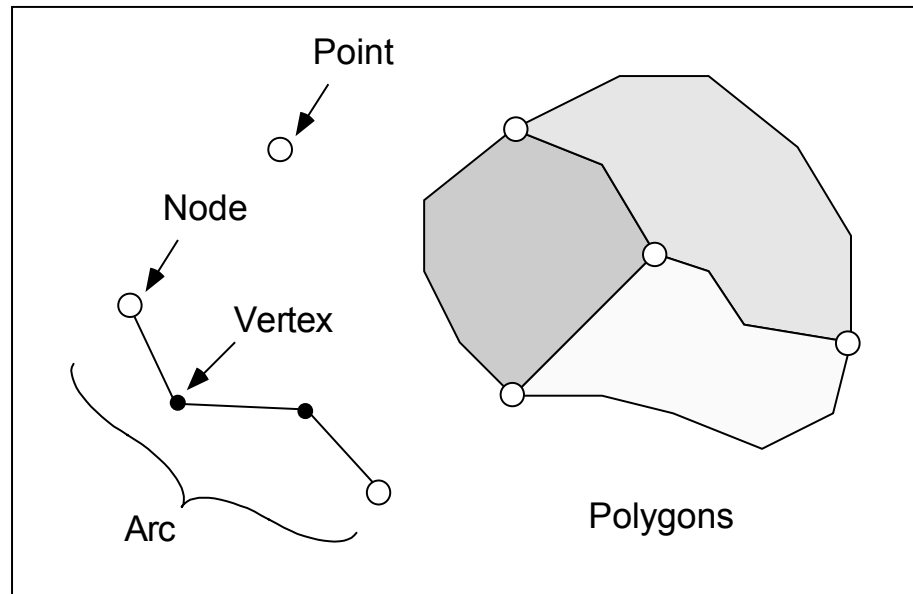


Figure 2-1 Feature Objects.

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:

- Drainage - defines basin boundaries and stream networks.
- Land Use – defines land use type polygons
- Soil – defines soil type polygons
- Time Computation – used for determining lag times and times of concentration.
- Generic
- 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*”. This file should be found in the tutorial directory.
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.

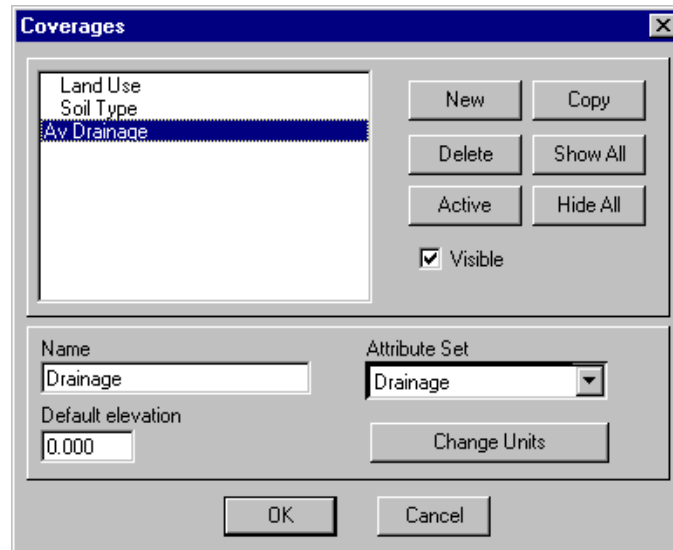



Figure 2-2 The Coverages Dialog.

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 file (DRG).

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.

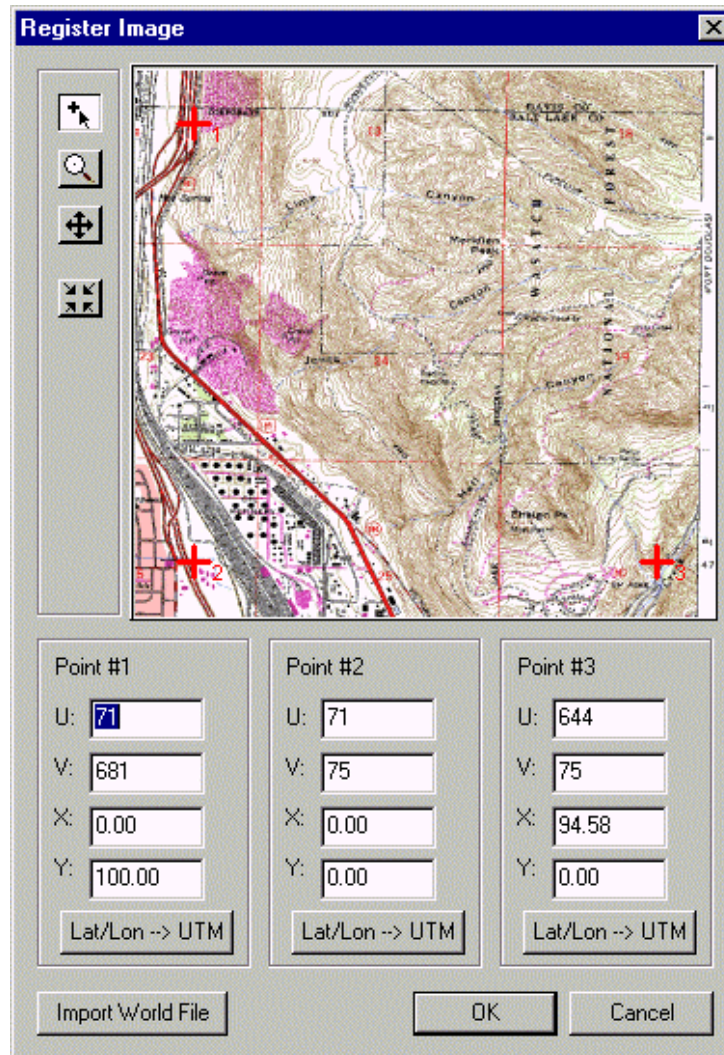


Figure 2-3 Image Registration Dialog.

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.
3. Select the pan tool .
4. Pan to a different portion of the image by dragging the cursor in the Graphics window.

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

1. Select the *Fit Entire Image* command from the *Image* menu.
2. 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 the image in order to see features more clearly.

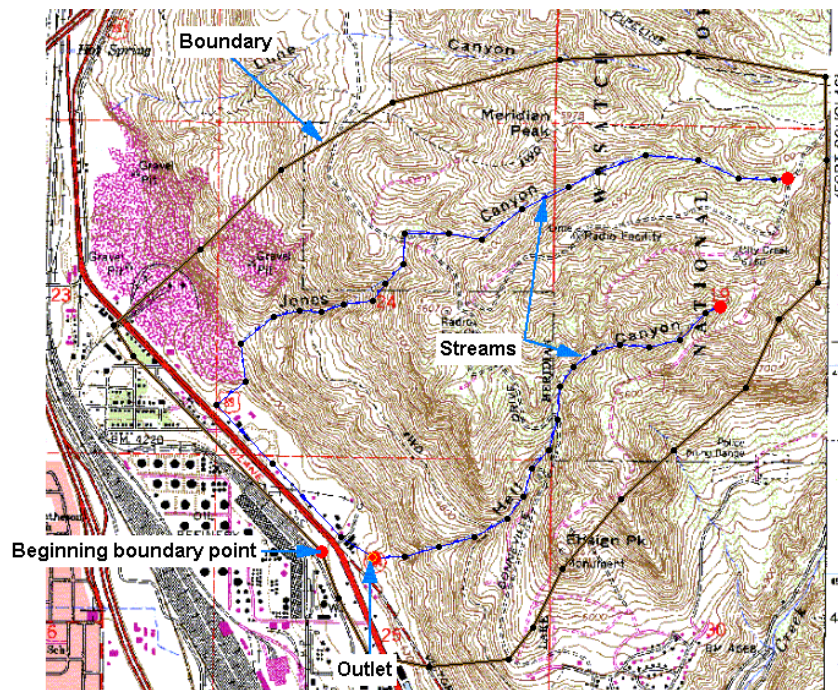


Figure 2-4 Completed Feature Objects for Jones Canyon Watershed.

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. Select *Feature Object Files (*.map)* from the *Save as type* field.
3. Click on the directory button and specify a directory where you want to save your files.
4. Enter “*jones1.map*” for the filename.
5. Hit *Save* to exit the 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*”. This file should be found in the tutorial directory. 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 *Import...* command from the *File* menu.
5. Select *Feature object arcs Shapefile (*.shp)* from the *Files of type* field.
6. Find and *Open* “*arco.shp*”.

HINT: In the arc attribute text window of the *Import Shapefile Data* dialog, 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.

7. 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 *Select Arcs* tool  in the toolbox.
3. Find and use SHIFT-select to select the two intersecting arcs (See Figure 3-1).

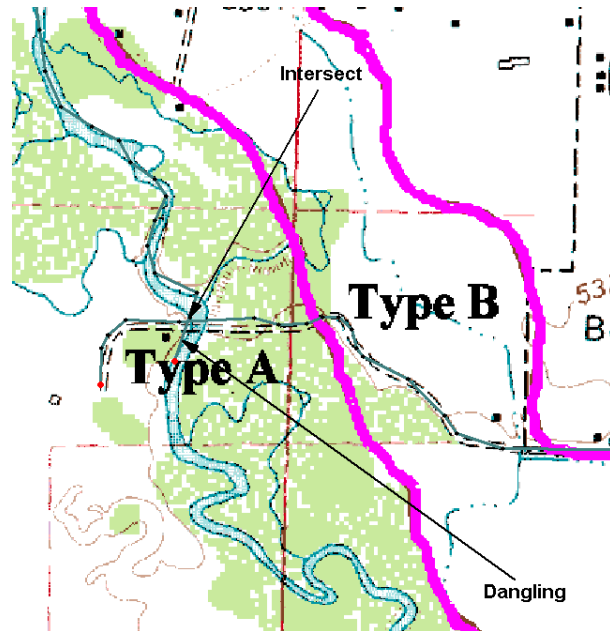


Figure 3-1 Cleaning Intersections and Dangling Arcs.

4. Select the *Clean...* command from the *Feature Objects* menu.
5. 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 *Select Point/Node* tool .
4. Select the node at the end of the arc to snap (See Figure 3-2).

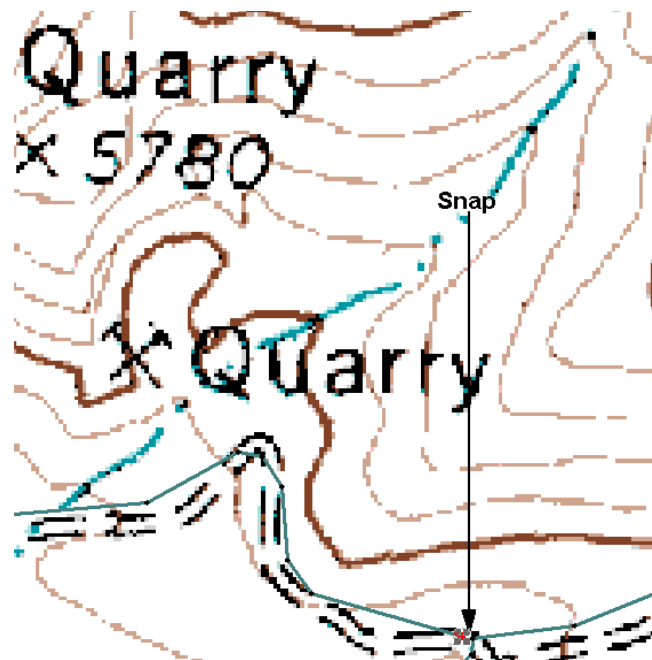




Figure 3-2 Snapping Selected Nodes

5. Select the *Clean...* command from the *Feature Objects* menu.
6. Make sure the *Snap selected nodes* option is selected and select the *OK* button. You will be prompted in the Help window 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 *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.

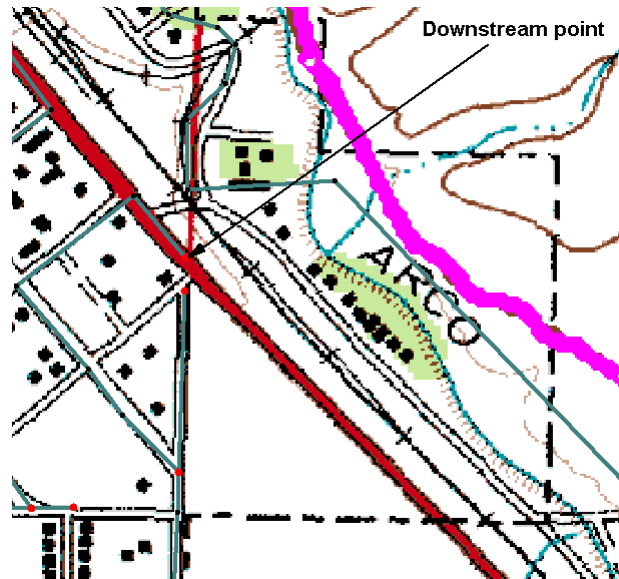





Figure 3-3 Reordering Streams.

3. Select the *Select Point/Node* tool .
4. Select the downstream node (See Figure 3-3).
5. Select the *Reorder Streams* command from the *Feature Objects* menu.
6. Select the *Frame Image* macro .
7. Select the *Select Stream Network* tool .
8. Select any location on the arc network.
9. 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.

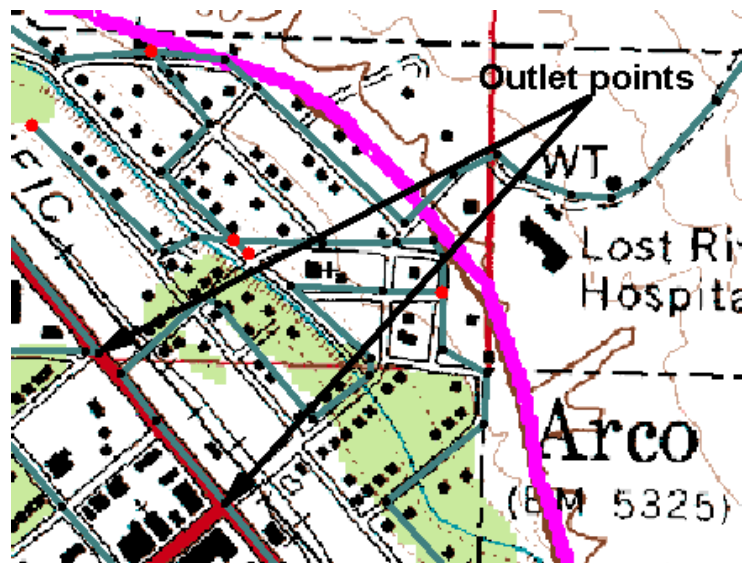




Figure 3-4 Creating Outlet Points.

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.

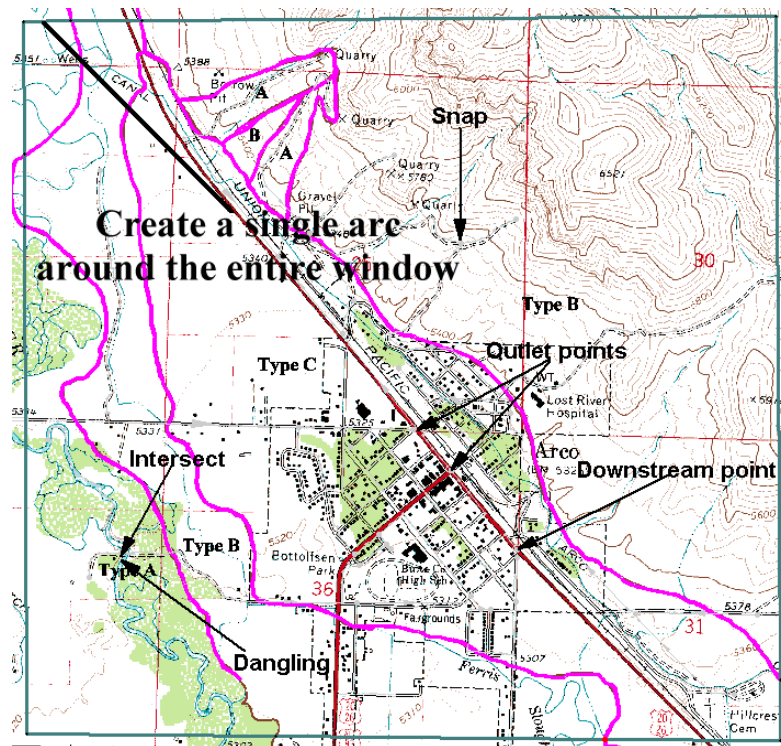


Figure 3-5 Step 1—Create a Single Arc around the Entire Window.

- 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.

- 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.

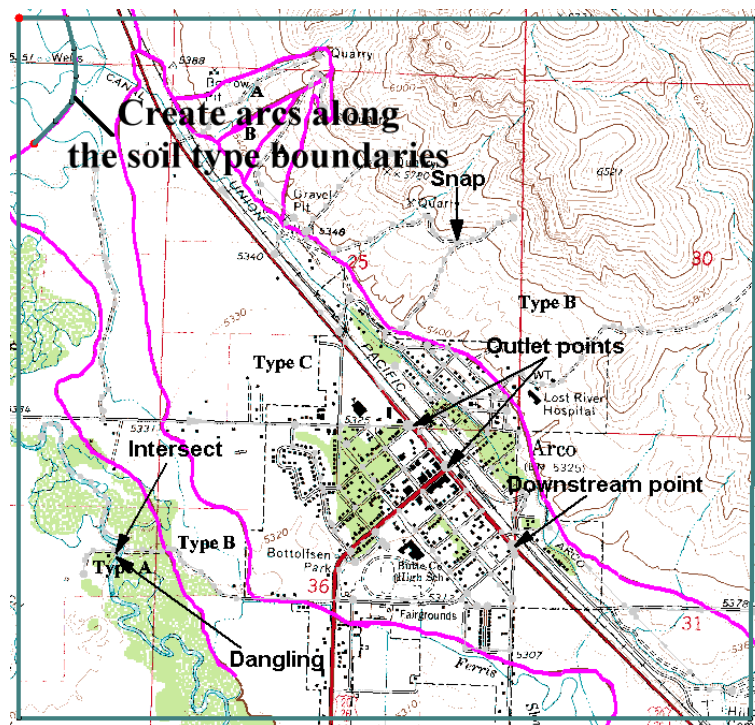



Figure 3-6 Step 2—Create Arcs Along the Soil Type Boundaries.

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.

- 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.

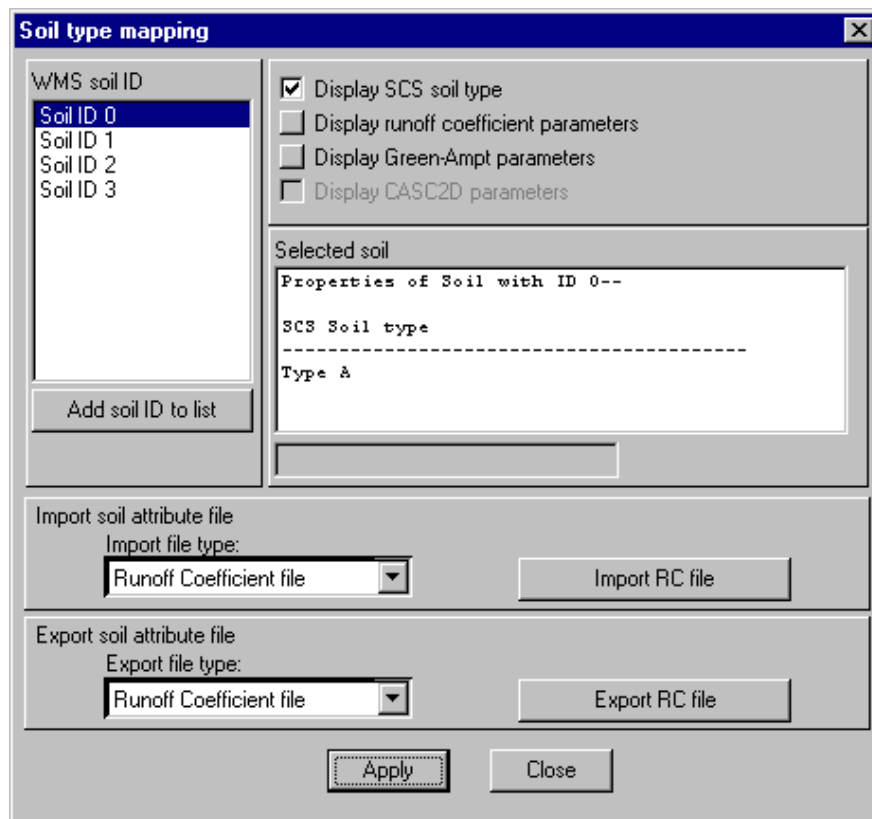


Figure 3-7 The Soil Type Mapping Dialog.

6. Perform steps 4 and 5 for soil types B, and C. For soil type B, you will need to select “Add soil ID to list”, select the *Soil Type* soil property, and change the *SCS soil ID* to *Type B*. For soil type C, you will need to select “Add soil ID to list”, select the *Soil Type* soil property, and change the *SCS soil ID* to *Type C*.
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 *Feature object polygons Shapefile (*.shp)* from the *Files of type* field.
3. Find and *Open* “*aspenso.shp*”.
4. 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.

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.

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

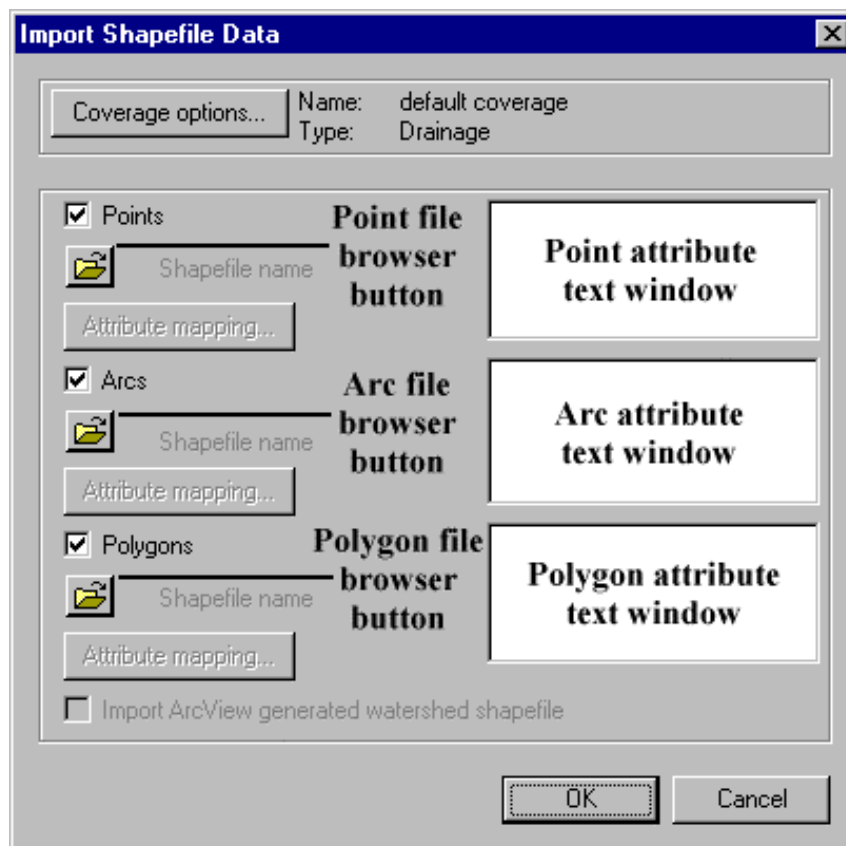


Figure 3-8 The *Import Shapefile Dialog*.

9. 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 *Feature object polygons Shapefile (*.shp)* from the *Files of type* field.
3. Find and *Open* “*aspenlu.shp*”.
4. 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.

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.

5. Select the *Coverage options...* button in the *Import Shapefile Data* dialog.
6. Select the *New* button in the *Coverages* window.
7. Change the name of the coverage to *Land Use* and change the coverage’s *Attribute set* to *Land Use*.
8. Select the *OK* button in the *Coverages* window.
9. 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 *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.

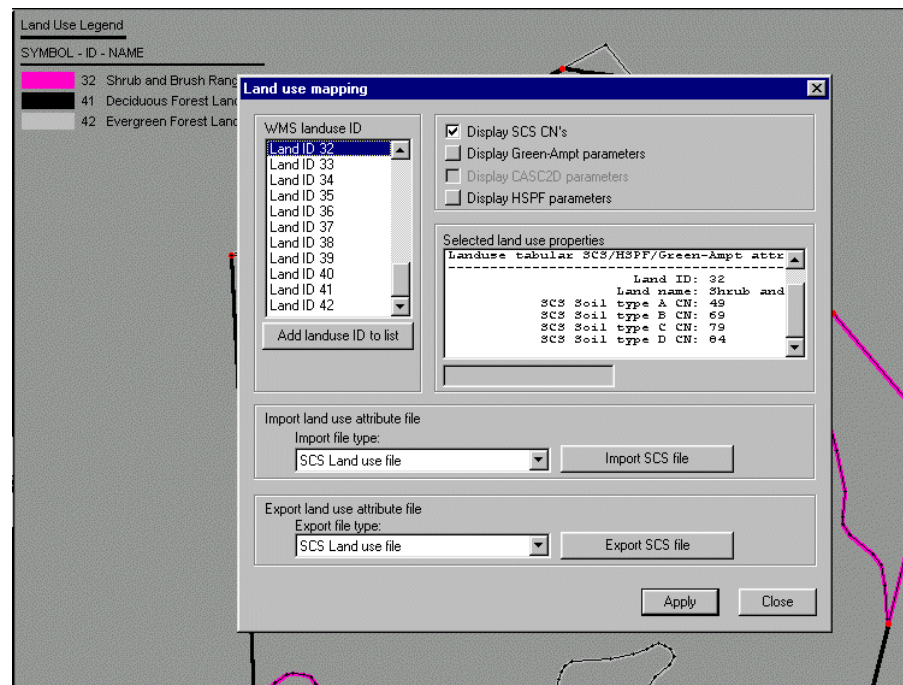





Figure 3-9 The Land Use Polygon Attributes (Land Use Mapping) Dialog.

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>
Name	Deciduous Forest Land	Evergreen Forest Land
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 “*aspen.cn.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 *Compute GIS Attributes...* 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 by selecting the *Display Options...* command from the *Display* menu and then selecting the *Drainage* tab and marking the *Basin CN's (curve numbers)* toggle box.

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. Under the *Map* tab, toggle the *Land Use Legend* option off and select the *OK* button to close the *Display Options* dialog. 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 *Create feature arcs from DXF lines* and turn off the toggle to *Create feature 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. Select *Feature Object Files (*.map)* from the *Save as type* field.
3. Specify the directory where you want to save your files.
4. Enter “*aspen3.map*” for the filename.
5. Select *Save* to exit the dialog and save the map file.
6. Select the *New* command from the *File* menu.
7. 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.

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*”. This file should be found in the tutorial directory.

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.

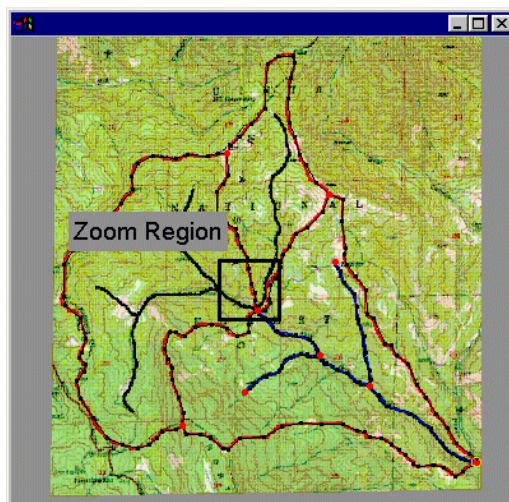



Figure 4-1 Junction of Main Channel in Aspen Grove Watershed.


3. Select the *Create Arcs* tool .
4. 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 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. 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 *Compute 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 *Basin Areas* units to *Square miles*.
4. Set the *Distances* 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 that 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.
2. Select *OK* to delete everything.
3. Select the *Map* module icon  if it isn't active already.
4. Select the *Import...* command from the *File* menu.
5. Select *Feature object polygons Shapefile (*.shp)* from the *Files of type* field.
6. Find and *Open* the polygon shape file named "*asppoly1.shp*".

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*”.

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* to close the *Import Shapefile Data* dialog.

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 hydrograph 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* twice to close the *XY Series Editor* and *HEC-1 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 it will ask if you want to exit window, select *YES*.
6. WMS will automatically open the files that you created with the HEC-1 program.

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.
2. Select *OK* to confirm that you want to delete everything.
3. Select the *Map* module icon .

4.4.1 Importing the Basin and Stream Layer

1. Select the *Import...* command from the *File* menu.
2. Select *Feature object polygons Shapefile (*.shp)* from the *Files of type* field.
3. Find and *Open* the polygon shape file named “*asppoly2.shp*”.
4. In the *Import Shapefile Data* dialog, click on the *open arcs shape file* button.



5. Find and *Open* the arc shape file named “*asparcs2.shp*”.
6. Select *OK* to close the *Import Shapefile Data* dialog.

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 in the Help window a point to snap to. 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.

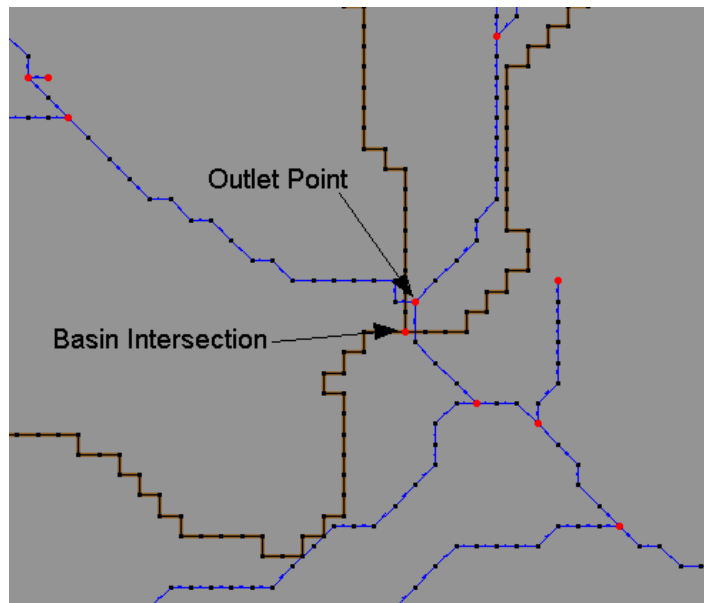





Figure 4-2 Snapping Nodes Together.

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 inside each while holding down the SHIFT key or by using the *Select All* command from 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 *Compute 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 also highlighted.

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 determine 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 that 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 *OK* to confirm that you want to delete everything.
3. Select the *Map* module icon .

4.5.1 Importing the Basin Layer

1. Select the *Import...* command from the *File* menu.
2. Select *Feature object polygons Shapefile (*.shp)* from the *Files of type* field.
3. Find and *Open* the polygon shape file named “*asppoly2.shp*”.
7. Select *OK* to close the *Import Shapefile Data* dialog.

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.

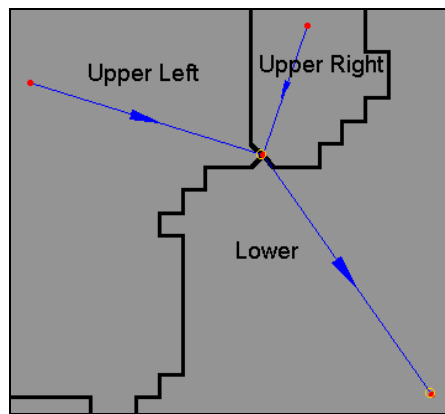



Figure 4-3 Creating Streams for Basin Polygons.

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.

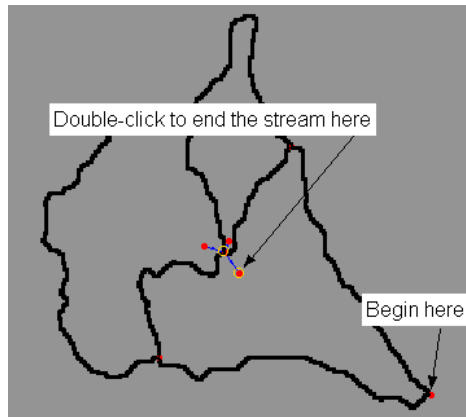




Figure 4-4 Finishing the Stream.

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 inside each while holding down the SHIFT key or by using the *Select All* command from 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 *Compute 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 that 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 *OK* to confirm that you want to delete everything.
3. Select the *Map* module icon .

4.6.1 Importing the Stream Layer

1. Select the *Import...* command from the *File* menu.
2. Select *Feature object arcs Shapefile (*.shp)* from the *Files of type* field.
3. Find and *Open* the arc shape file named “*asparcs1.shp*”.
8. Select *OK* to close the *Import Shapefile Data* dialog.

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.

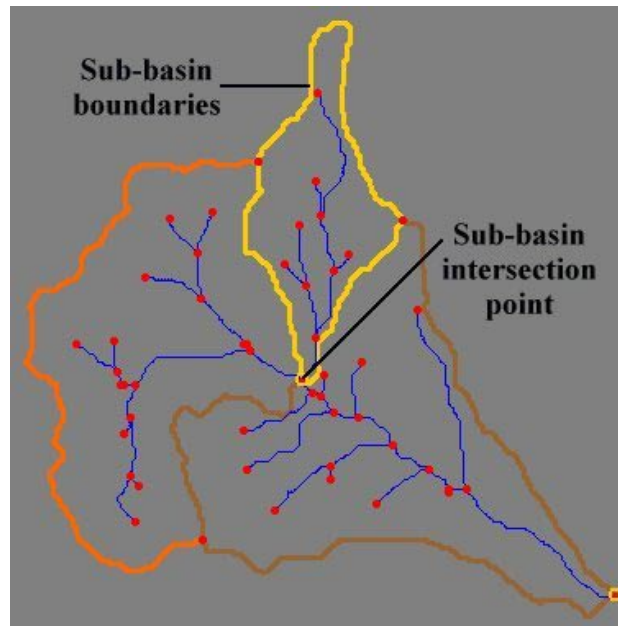






Figure 4-5 Sub-basins of Stream Network.

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. Select the *Zoom* tool .
9. Zoom in around the point on the stream where you will create the intersection of the three sub-basins (see Figure 4-5).
10. Select the *Create Arcs* tool .
11. 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.

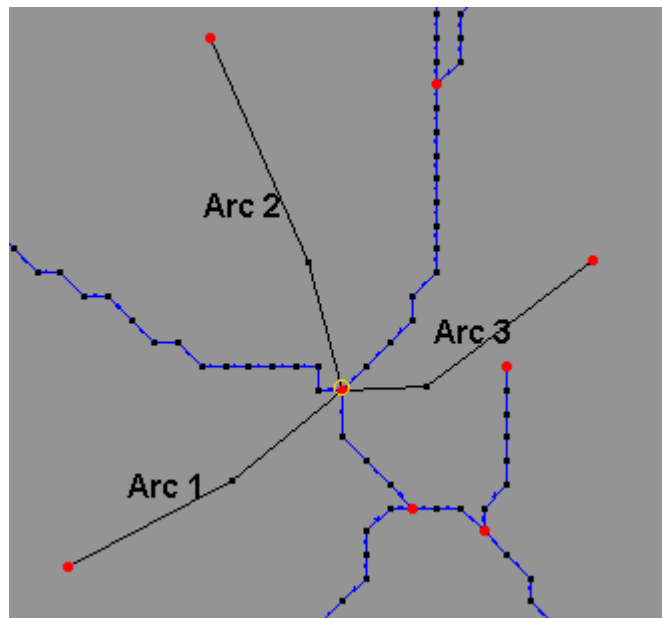





Figure 4-6 Interior Boundary Arcs.

1. Select the *Frame Image* macro .

2. Now finish defining the arcs to close the three basin boundaries by starting from the end point(s) of the three arcs at the intersection and continue tracing out the boundaries shown in Figure Figure 4-5.

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.

DEM Basics

Estimated Time: 15 minutes

A DEM (Digital Elevation Model) 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 round-off 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. Select *USGS DEM File (*.dem;*.ddf)* from the *Files of type* field.
4. 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 in the *Importing USGS DEMs*. The window at the top of the *Import USGS DEMs* dialog shows a list of the DEM files that will be imported.
5. Select *OK*.

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.
4. When asked if you want to delete the inactive region, select *No*.

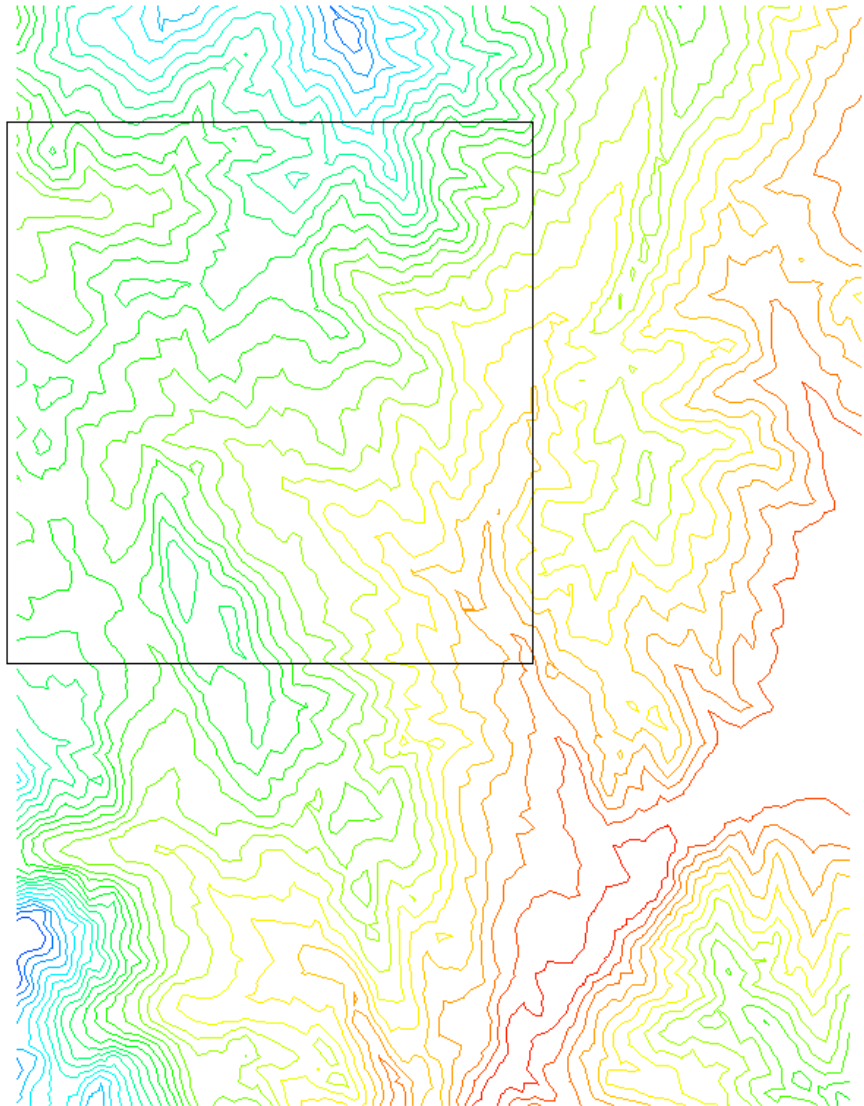


Figure 5-1 DEM Region Selection Box.

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, by doing the following:

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 option.

1. Select *Contour Options...* in the *Display* menu.
2. Turn on the *Display Legend* toggle.
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 too 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 *DEM 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.
4. Select *OK*.

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.

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 its 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 lower left portion of the dialog.
6. Select *OK*.
7. Select the *Import...* command from the *File* menu.
8. Select *Arc/Info grid (*.asc)* from the *Files of type* field.
9. In the *File name:* field type “*jonescyn.grd*”.
10. Select *Open* to import the file.
11. Select *OK* to delete existing DEM data.

The *Importing ARC Info* dialog will open. In this dialog you can add more grid files to import or adjust the coordinates of the DEM coverage.

12. Select *OK* to close the *Importing ArcInfo Grid* dialog and to import “*jonescyn.grd*”.

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 on DEM Contours and Flat DEM cells, and turn everything else off.
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 *Smooth DEM* command from the *DEMs* menu.

This opens a dialog with options that 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 *OK*.

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.

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. Select *USGS DEM File (*.dem;*.dbf)* from the *Files of type* field.
4. 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 in the *Import USGS DEMs* dialog.

The window at the top of the dialog shows a list of the DEM files that will be imported.


1. 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*”.

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. Select *Yes*, deleting the inactive DEM region.
5. Select the *Display Options...* command from the *Display* menu.

6. Turn off the display of Flow Accumulation and select *OK*.

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 automatically 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*.
3. Select *Yes* to close the application window when TOPAZ finishes.

TOPAZ creates a file called "*flovec.dat*" in which the flow direction data is stored. *WMS* then automatically reads the flow directions file, but you must change the Display Options to show the flow directions in the graphics window.


1. Select the *Display Options...* command from the *Display* menu.
2. 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:

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 “*uparea.dat*” was computed with TOPAZ. The Flow Accumulation is determined by using the flow directions to compute how many DEM points “drain” to a given DEM point for every DEM point in the analysis area. The flow accumulations will be displayed when you have either read them in or computed them in WMS. As with the flow directions, WMS has automatically read in the flow accumulation file computed by TOPAZ. All you have to do is display it.

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


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:

3. Select the *Display Options...* command from the *Display* menu.
4. Set the *Min accumulation for display* value to be 1000.
5. 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.

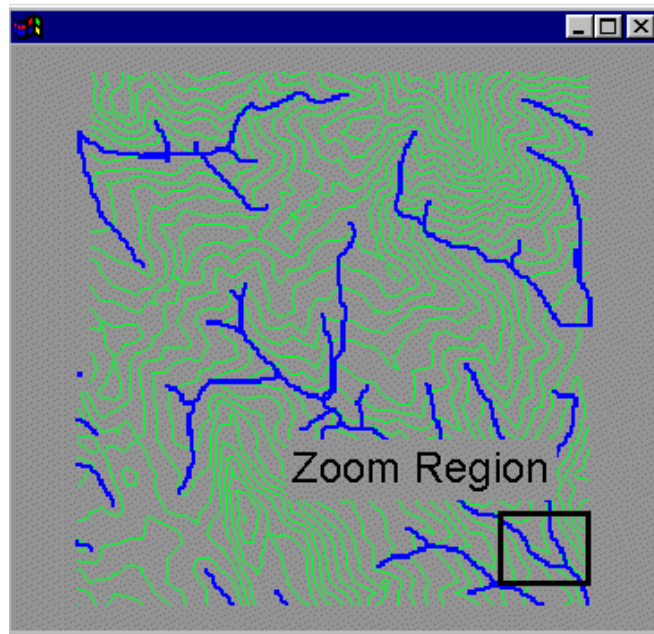



Figure 6-1 Aspen Grove Stream Network Branching Point.

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).

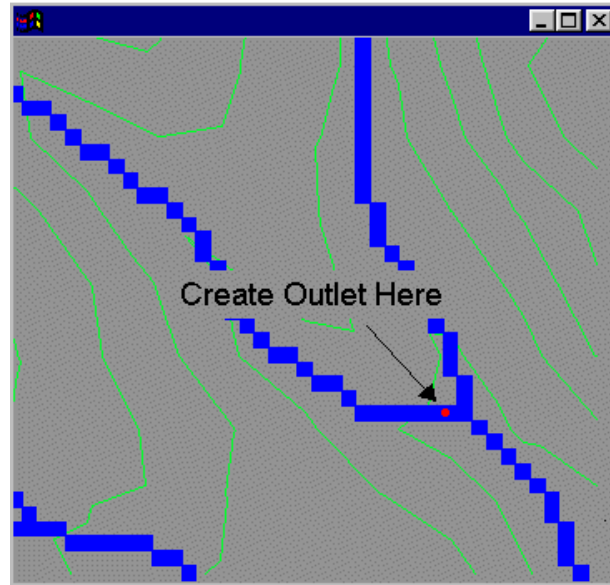




Figure 6-2 Aspen Grove Watershed Outlet Location.

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 streams* 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.

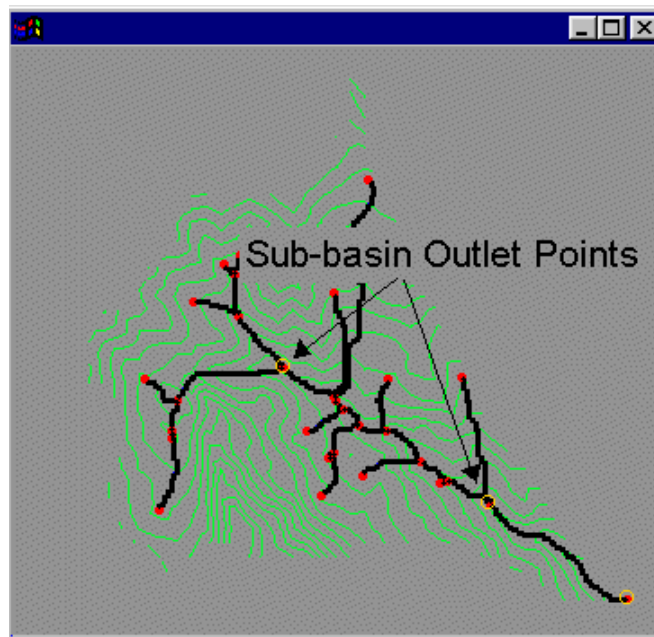


Figure 6-3 Sub-basin Outlets of Aspen Grove Watershed.

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. Select the *Drainage* tab.
6. Insure that the *Show units*, *Basin areas*, and *Basin slopes* options are toggled on.
7. 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.

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.
2. Select *OK* if you are told that retriangulating will delete existing triangles.

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* if 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 in the *View* menu. 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 *Tin 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.

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 Options* tab.
3. Select the *Intensity Ramp* 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 *Tin 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 *OK* twice to close the *Image Display Options* and *Display Options* dialogs.

9. Select the TINs module .
10. 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 . Then choose the *Shade* command from the *Display* menu.
- 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 *OK*.
6. Select the *TIN* tab.
7. Turn on *Tin Contours*.
8. Select *OK*.
9. 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, use 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 cannot 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, you are allowed 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. (Be careful not to select a vertex when you begin to drag your box.) 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.

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 *Vertices*, *Triangles*, *Tin Contours*, *Ridge Edges*, *Channel edges*, and *Pits* options.
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. If prompted, select *Lock All* to lock all the vertices.

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. Select the color box to the left of the *Contours* display option toggle.

3. Change the width of the line to 2.
4. Select the color box from the *Attributes* dialog.
5. A color palette appears from which you may assign a new contour color.
6. Click on the *OK* button for the *Color Palette* dialog.
7. Click on the *OK* button for the *Attributes* dialog.
8. Click on the *OK* buttons for the *Display* options dialog.

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 that 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.

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 that 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. Select the *Map* module icon .

2. Select *Open...* from the *File* menu.
3. Type **.img* in the file name edit field (this will cause the file browser to only list file names ending with *img*).
4. Find and *Open* the file named “*jonescyn.img*”. This file should be found in the tutorial directory.

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 images, 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. Find and select, but do not open, “*jonescyn.grd*”.
3. Select *Arc/Info Grid* from the *Files of type* field.
4. Select the *Open* button.

5. Select *OK* to close the Importing ArcInfo Grid.
6. Select the *DEM* module .
7. Select the *Smooth DEM...* command from the *DEMs* menu.
8. Set the number of iterations to 3.
9. Select *OK*.

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 asked 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 in the *Create Tin Options* dialog (this will allow you to see, and better understand, how the TIN creation process works).
6. Select *OK*.
7. Select the *Display Options...* command from the *Display* menu.
8. Select the *Image Display Options* button.
9. Turn off the *Draw on xy plane behind all objects* option.
10. Select *OK* to the *Image Display Options* and *Display Options* dialogs.
11. Select the *Delete* command from the *Drawing Objects* menu.
12. 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 *Flat triangles* and *Pits*.
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 that 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:

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 *Tin 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.

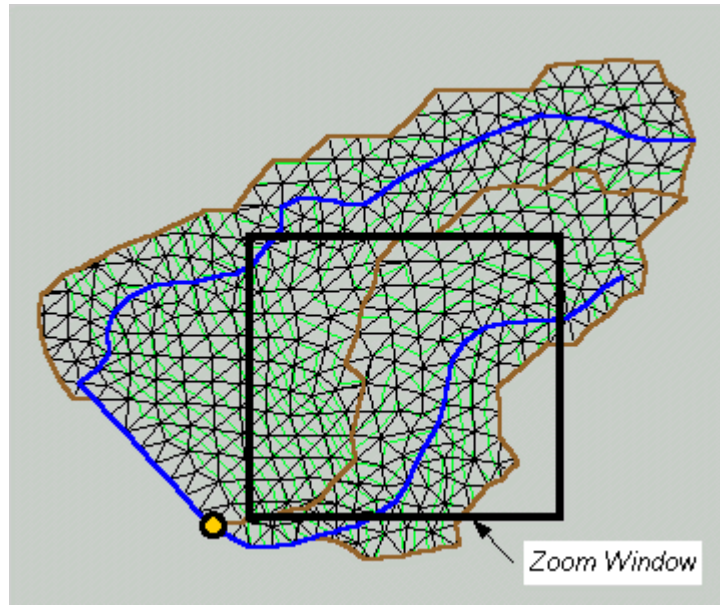



Figure 8-1 Zoom Window.

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.

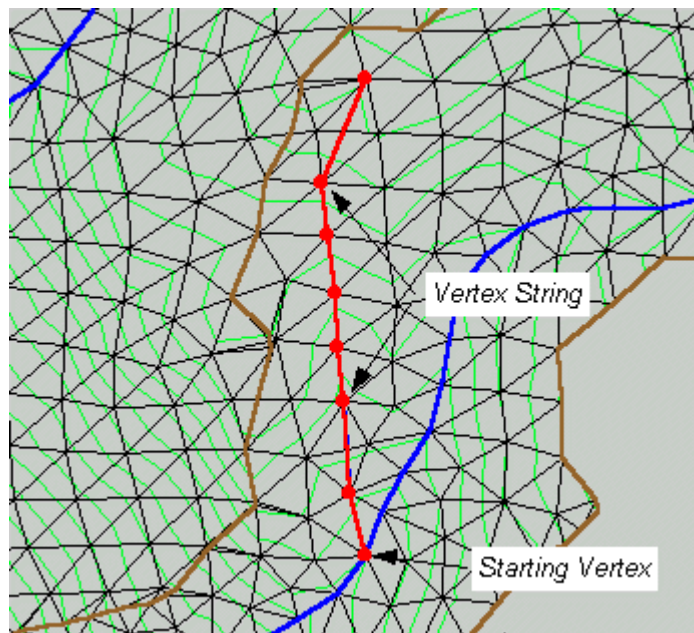



Figure 8-2 Entering a Vertex String to Create a New Stream.

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.

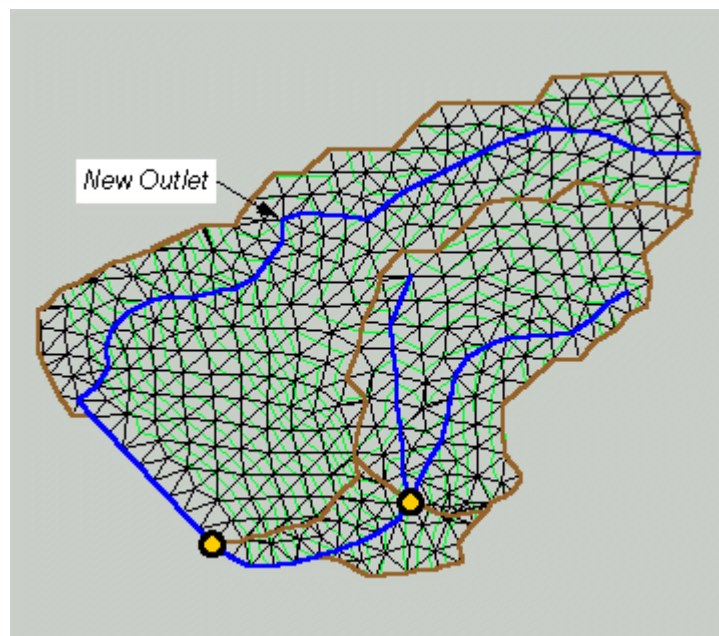


Figure 8-3 Vertex for New Stream


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 *Tin Contours*.
3. Select the *Drainage* tab.

4. Turn on *Fill drainage basins*.
5. Select *OK* to close the *Display Options* dialog.
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 small, central basin.
3. Hold the SHIFT key down and click inside the basin to the right of that.
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 newly created basin.
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 an 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 and SHIFT-select the outlet that you created on the upper stream.
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 slopes* and *Average overland flow*.
4. Select the *OK* button.

Text showing these values, in addition to the area 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 include text, lines (with or without arrows), 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 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.

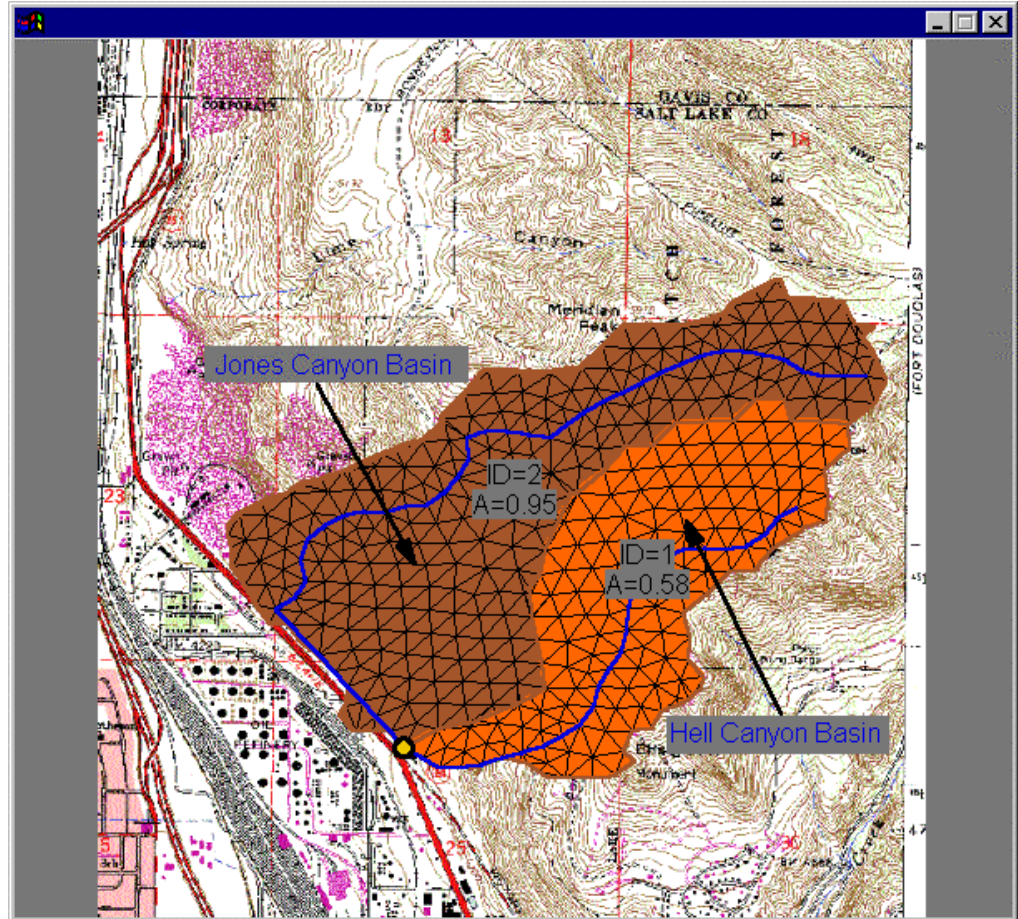


Figure 8-4 Annotations for Jones Canyon Watershed.

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.

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 that 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 WMS map (*.map) 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. Change to the *Hydrologic Modeling* module .
3. Select *Open...* command from the *File* menu.
4. 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.

5. Select *Display Options...* from the *Display* menu.
6. Turn the display of all entities off.
7. Select the *Drainage* tab.
8. Be sure the display of Outlets, Outlet names, Stream networks, Storm gages, Temporal distribution gages, and Drainage basin boundaries are all turned on.
9. 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. Change to the *TINs* module .
2. Select the *Compute Basin Data...* command from the *Drainage* menu.
3. Specify *Feet* for the *Model units*, *Square miles* for *Basin Areas*, and *Feet* for *Distances* and then select the *OK* button.

After the computation, the basin areas will be displayed at the centroids of the drainage basins. 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.

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 that 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.

- 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.

- 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:

- Select the *Job Control...* command from the *HEC-1* menu. A dialog will appear with several different job control options.
- 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.
- Set the computational time interval to 10.
- Set the date fields to today's date.
- Set the number of hydrograph ordinates to 75.
- 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:

- Click on the *Select Drainage Basin* tool .
- 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 that 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 (UD)* 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 user's 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(PG)* 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*, *Square miles* for *Basin Areas*, and *Feet* for *Distances* 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. Select one of the “upstream” icons, (not the bottom most outlet). The three “upstream” outlets need to 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 (RM)* 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 length 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 (length/3 = travel time seconds)
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 *Compute GIS Attributes...* command from the *Calculators* menu.
2. Make sure *SCS Curve Numbers* is selected in the *Computation* list.
3. Select *Soil type coverage* for determining soil type.
4. Select *Land use coverage* for determining land use.
5. Select *OK* to compute composite curve numbers.

So that you can be satisfied that lag times and curve numbers (CN) 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 *Display* menu.
3. Turn on the *soil group*, *soil group legend*, *land use legend*, and *land use* 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 that 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. *WMS* automatically reads in the files. To view the hydrographs:

1. Select the *Show Hydrograph Window* from the *Display* menu.

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, representing the hydrograph before and after channel routing, will be displayed for some of the outlets. 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.

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.

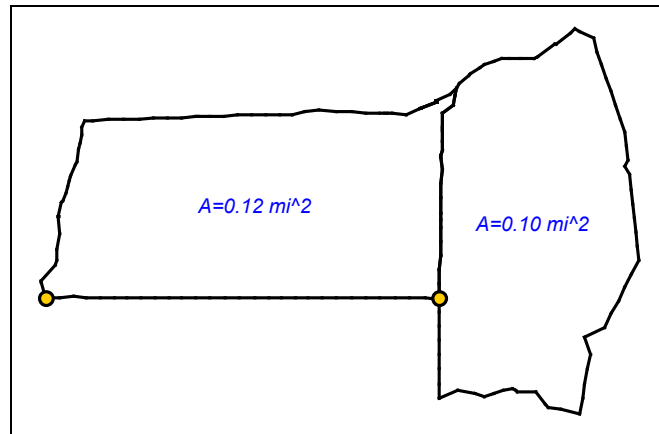



Figure 10-1 American Fork Watershed

10.1 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. Select *OK*.
7. 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.

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

10.2 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.

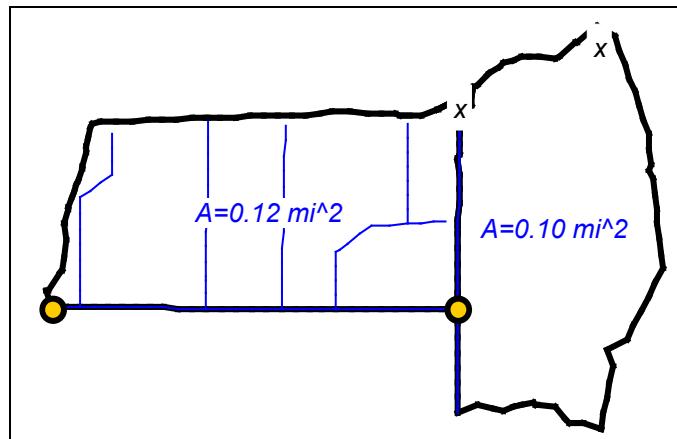




Figure 10-2 Starting Locations for Time of Concentration Arcs.

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 Arcs* 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) uses 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).

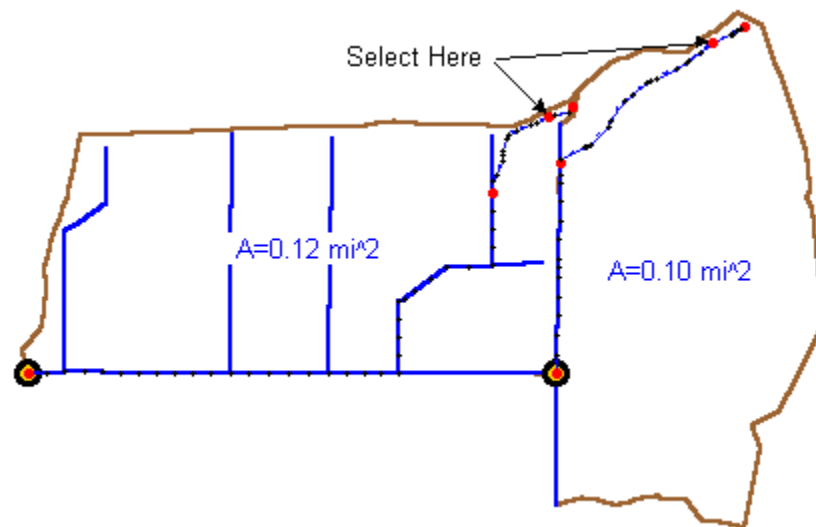



Figure 10-3 Locations to Break Sheet Flow Segments.

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.

10.3 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.

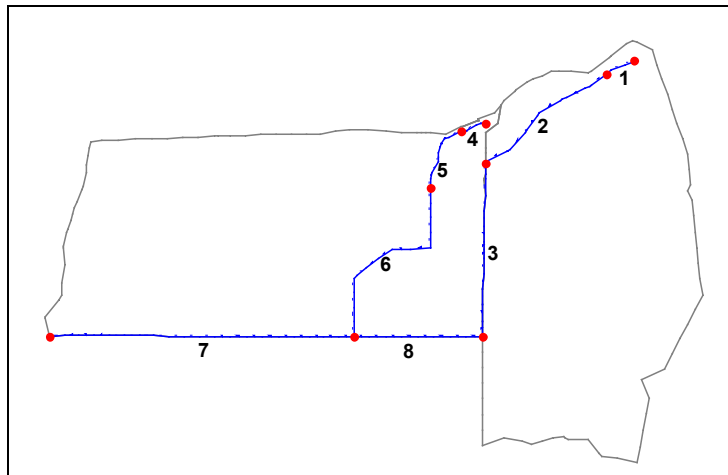



Figure 10-4 Time Computation Arcs.

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.
3. Select *OK*.

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 equation 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 0.0015.
10. Enter a *Channel 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* for both dialogs.

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.

10.4 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. For this tutorial you will be running *TR-55*, but the same time computation tools you learn in this tutorial could be used for any of the supported WMS models (such as in the *TR-20* basin data dialog, the *HEC-1* Unit Hydrograph method dialog, and the *Rational Method* dialog).

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  while the *TR-55* dialog is still open.
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 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 to close the *TR-55* dialog.
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 *Compute 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* to close the *Basin Time Computation* dialog.

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 Data* 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 its 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*.

10.5 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  with the *TR-55* dialog still open.
2. Select the upstream outlet (the outlet to the upper basin).
3. Select the *Compute travel time...* button in the *TR-55* dialog.
4. Make sure that both of the downstream arcs are active (The active status in the dialog is turned on for both).
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.

10.6 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  with the *TR-55* dialog still open.
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(s)* button.
3. Select *Done* to close the *TR-55* dialog.
4. Select the *Show Hydrograph Window* command from the *Display* menu.
5. Click on the hydrograph icon that is displayed by the upper basin.

10.7 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.

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 that 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.


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.

11.1 Reading a TIN File

You will first read in a TIN that has already been processed and used to delineate a single basin.

1. Select the *TINs* module .
2. Select the *Open...* command from the *File* menu.
3. Find and *Open* the file named “*rockynff.tin*”. By default it should be in the tutorial directory.

11.2 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.

11.3 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.

11.3.1 Choosing a State

WMS is linked to a database of state-by-state regression variables that 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*.

11.3.2 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 2 and 30% in Region 3. (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 2* 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 3.

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.

11.3.3 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 % Basin Which Overlaps Regions* button.
2. Enter 70 for the *Percent area* for Region 2.
3. Enter 30 for the *Percent area* for Region 3.
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.

11.3.4 Defining Regression Variables

All three of the 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 Computed Geometric Values* button.

11.3.5 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.

11.3.6 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.

11.3.7 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 *Compute – Basin Data* 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:

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 in the NFF simulation dialog.

11.3.8 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.

11.3.9 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* dialog is not up, 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 = 20
 - 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.

11.3.10 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 2* 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 3* 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.

11.4 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.

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:

1. Select the Select Points/Nodes tool .

2. Select the point indicated in Figure 12-1.

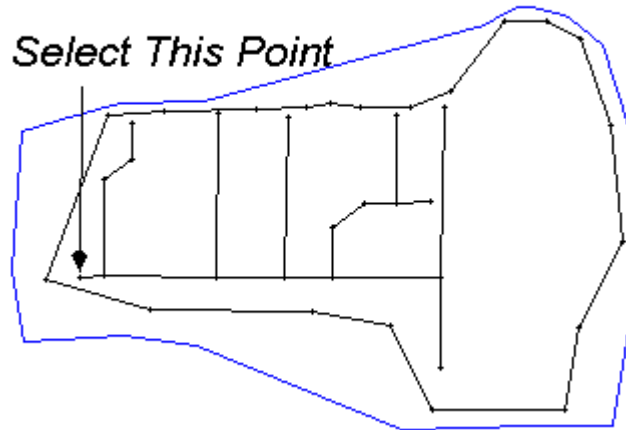



Figure 12-1 Location of Watershed Outlet Used for Stream Reordering.

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 end 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.

- 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.

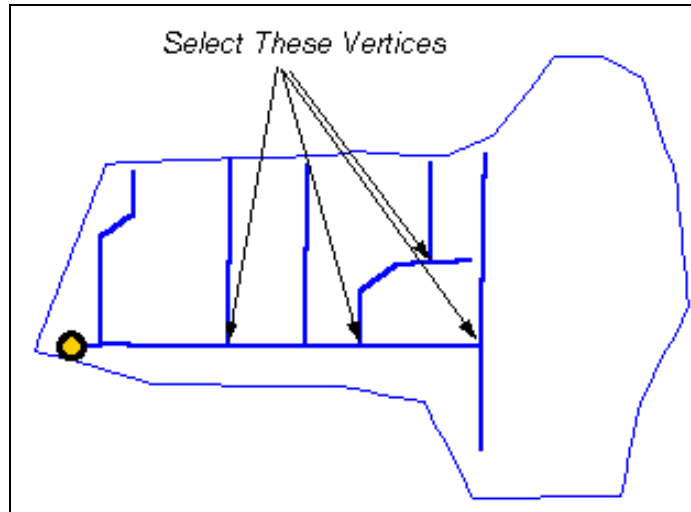


Figure 12-2 Location of Outlet Vertices.

- Select the *Add Outlets* command from the *Streams* menu.
- Select the *Define Basins* command from the *Drainage* menu.

Because of the way vertices are connected by triangle edges, there are 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.


- Select the *Correct Split Flow* command from the *Drainage* menu.
- 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 that straddle basin boundaries. After refining boundaries you will also want to eliminate triangles that do not belong to any of the catchment areas.

- Select the *Refine Boundaries* command from the *Drainage* menu.
- Select the *Delete Null Basin Triangles* command from the *Drainage* menu.
- 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. Select both basins labeled A in Figure 12-3 while holding down the SHIFT key.
7. Select the *Merge Basins* command in the *Drainage* menu.

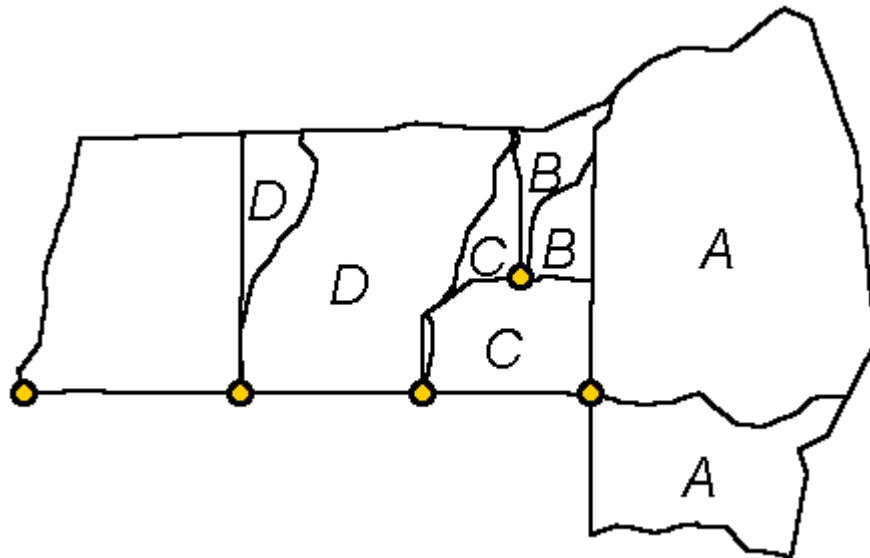


Figure 12-3 Delineated Catchment Areas Before Merging.

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.

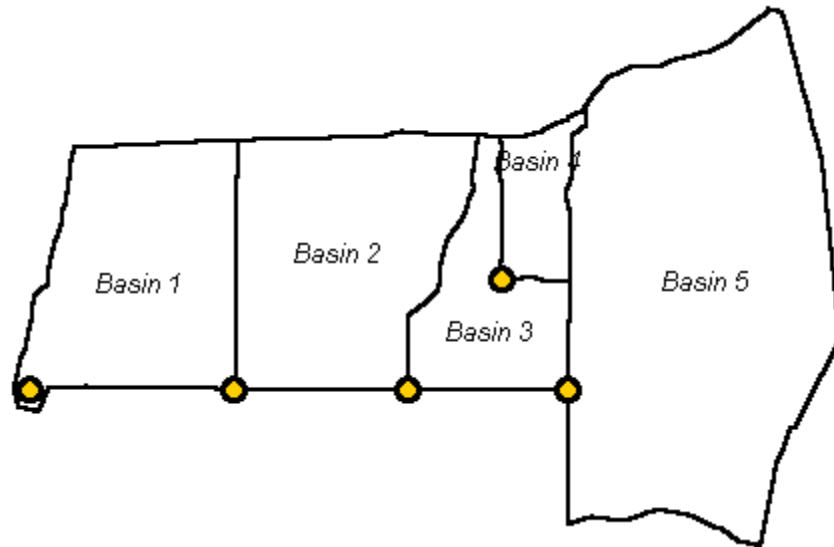


Figure 12-4 Final Delineated Basins.

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 basin icon or anywhere inside the basin.
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 “*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 Data (Eastern US)* radio group button is selected and select the *Define Data* button.
4. Enter the following values to define IDF curves using HYDRO-35

Time (min)	Depth (in)
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.
2. Select the *Refresh* icon on the lower right.

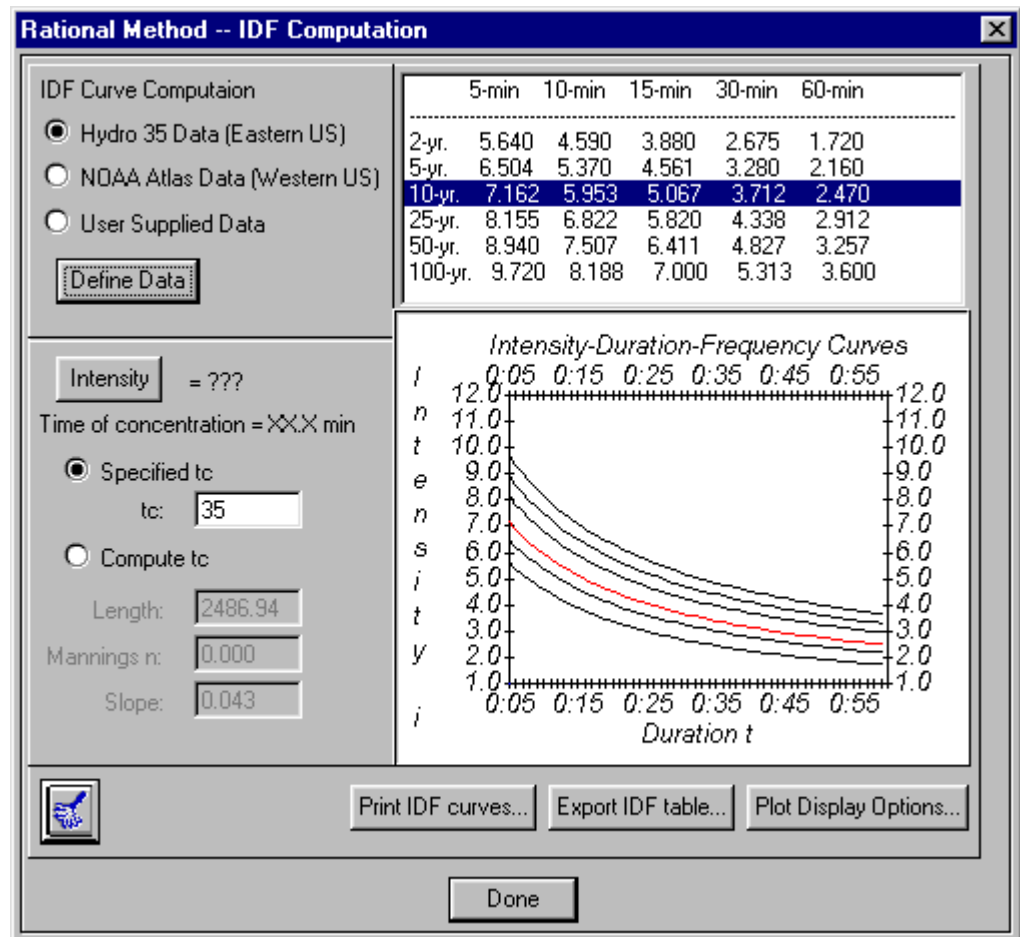


Figure 12-5 IDF Computation Dialog with the 10-yr Recurrence Interval Selected.

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 the *Select Outlet* tool .
2. Select the outlet icon of *Basin 5* as shown in Figure 12-4 (be sure to select the outlet and not the basin icon).
3. Enter a value of 5 minutes for the *Routing lag time*.
4. Select the outlet icon of *Basin 4*.

5. Enter a value of 5 minutes for the *Routing lag time*.
6. Select the next downstream outlet (the outlet icon of *Basin 3*).
7. Enter a value of 7 minutes for the Routing lag time.
8. Select the next downstream outlet.
9. Enter a value of 6 minutes for the Routing lag time.

The last or downstream-most outlet does not need to have a Routing lag time defined since the hydrograph accumulations will occur at this point.

1. Select the downstream-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 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 a 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.

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 the *Hydrologic Modeling* module icon .
2. Select *Open HEC-1 File...* from the *HEC-1* menu.
3. Find and *Open* the file named “*redri.hc1*”.
4. Select *Open...* from the *Hydrographs* menu.

5. Find and *Open* the file named “*redri.sol*”.
6. Select *Show Hydrograph Window* from the *Display* menu.
7. 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.

1. 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 type.
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. Confirm that you want to delete everything.
3. Select the *Map* module icon .
4. Select *Open...* from the *File* menu.
5. 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* field.
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.

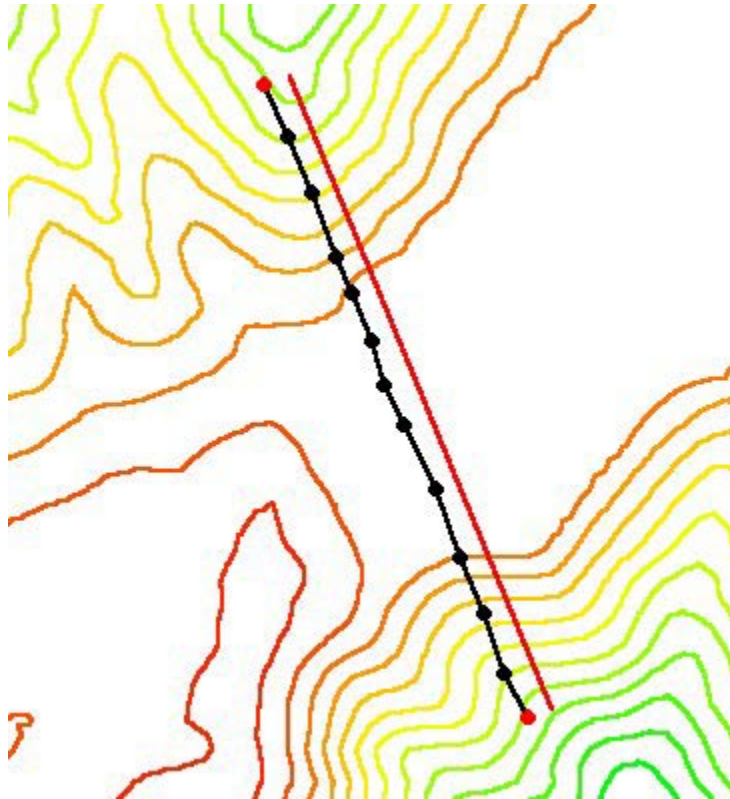



Figure 13-1 Cross section at Deer Creek.

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* dialog.

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 value 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 Calculations* 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 left 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. Confirm that you want to delete everything.
3. Select the *TINs* module icon .
4. Select *Open...* from the *File* menu.
5. 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, and 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 that you are most familiar.
8. Select *OK*.

You will see the *Detention Basin Hydrograph Routing* 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.
10. Select *Yes* to recompute basin data.
11. Select *OK* to accept the defaults in the *Units* 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, you can 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 a 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.

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:

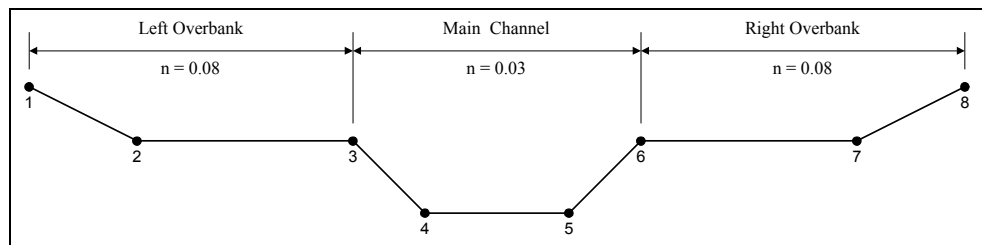


Figure 13-2 Sample cross section.

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 that 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:

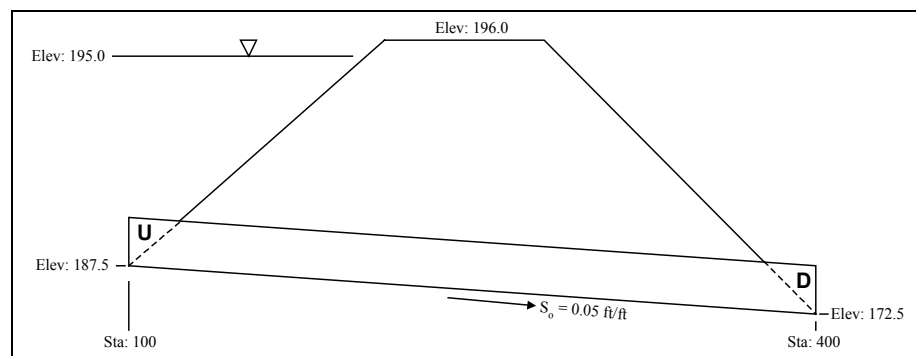


Figure 13-3 Culvert profile.

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 Bevel (45 deg. flare):Wingwall* 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* dialog.

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* dialog.
15. Click the *View rating curve data...* button to compute the rating curve and display the information.
16. Select *Done*.
17. Click the *Save tailwater file* button to bring up the file manager. Specify “*hy801.tw*” as the file name and save the file.
18. Click on the *OK* button to close the *HY8 Irregular Channel* dialog.
19. 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* dialog.
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* dialog.
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 are possible using these calculators.

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 stage*, *Fixed stage values*, *Interpolated stage*, and *Interpolated stage values* and make sure everything else is off.
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 plain for the portion of the stream near the outlet is 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.
5. Select *OK* to confirm that the drainage coverage will be read in as a generic coverage.

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.
5. Select *OK* to confirm that the drainage coverage will be read in as a generic coverage.

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 stage* 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.

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 *DEM Attribute (*.*)* from the *Files of type* field.
5. Find and *Open* “*asplugrd.asc*”.

This will open the *DEM Attributes* dialog, which allows you to assign attributes to the data that you import.

1. Select the *Land use* option for the *Attribute*.
2. Select *Arc/Info* for the *File type*, and select the *OK* button.
3. 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 *DEM Attribute (*.*)* from the *Files of type* field.
3. Find and *Open* “*aspsogrd.asc*”.
4. Select *Soil type* for the *Attribute*.
5. Select *Arc/Info* for the *File type*, and select the *OK* button.
6. 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. Select *Super Files (*.sup)* from the *Files of type* field.
3. 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 GIS Attributes...* command from the *Calculators* menu.
3. Make sure *SCS Curve Numbers* is selected in the *Computation* field.
4. Select the *Import* button.

5. 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:

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 *Land Use* and change the coverage's *Attribute set* to *Land Use*.
5. Select the *OK* button in the *Coverages* window.
6. Select the *Import...* command from the *File* menu.


7. Select *Feature object polygons Shapefile (*.shp)* from the *Files of type* field.
8. Find and *Open* “*aspenlu.shp*”.

In the polygon attribute text window of the *Import Shapefile Data* dialog, 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.

9. 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 GIS Attributes...* command from the *Calculators* menu.
5. Select the option to compute SCS Curve
6. Specify that you want to *Use a Soil type grid for determining soil type*.
7. Specify that you want to *Use a Land use coverage for determining land use*.
8. Enter a *Drainage coverage computation step* of 50.0 (Meters in this case).
9. Select the *OK* button in the *Compute GIS Attributes* 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 *Attribute set* to *Soil Type*.
5. Select the “Drainage coverage” and select the *Delete* button.
6. Select *OK* when prompted for confirmation.
7. Select the *OK* button in the *Coverages* window.
8. Select the *Import...* command from the *File* menu.
9. Select *Feature object polygons Shapefile (*.shp)*.
10. Select and *Open* “*aspenso.shp*”.


In the polygon attributes 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.

11. 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 (curve numbers)*.
4. Select *OK*.
5. Select the *Tree module* icon .
6. Select the *Compute GIS Attributes...* 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 and select *OK* to overwrite any existing data.
10. Find and *Open* “*scstut.tbl*”.
11. Select the *OK* button in the *Compute GIS Attributes* 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 *DEM Attributes (*.*)* from the *Files of type* field.
7. Select and *Open* “*aspsogrd.asc*”.
8. Select the *Soil type* option for the mask.
9. Select *Arc/Info* for the file type, and select the *OK* button.
10. Select the *Tree module* icon .
11. Select the *Compute GIS Attributes...* command from the *Calculators* menu.
12. Select the *Runoff coefficients* computation option.
13. Select the *Import* button in the *Compute GIS Attributes* dialog.

14. Find and *Open* “*soiltype.tbl*”. Notice that the table appears in the *Compute GIS Attributes* dialog after it is imported.
15. For soil type, make sure the option to *Use a soil type grid for determining soil type* is selected.
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.

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 that 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 *Interpolate 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 *2D Grid 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 Options* 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:

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 that 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/backward 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.

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 are 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. Confirm that you want to delete everything.
3. Select the *TINs* module icon .
4. Select *Open...* from the *File* menu.
5. Use the file browser to find and *Open* the file "*digitize.tin*". This file should be found in the tutorial directory.
6. Select the *Display Options...* command from the *Display* menu.

7. Make sure that *Vertices* and *Triangles* are turned on.
8. 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 *Tin 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 that 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 that 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.

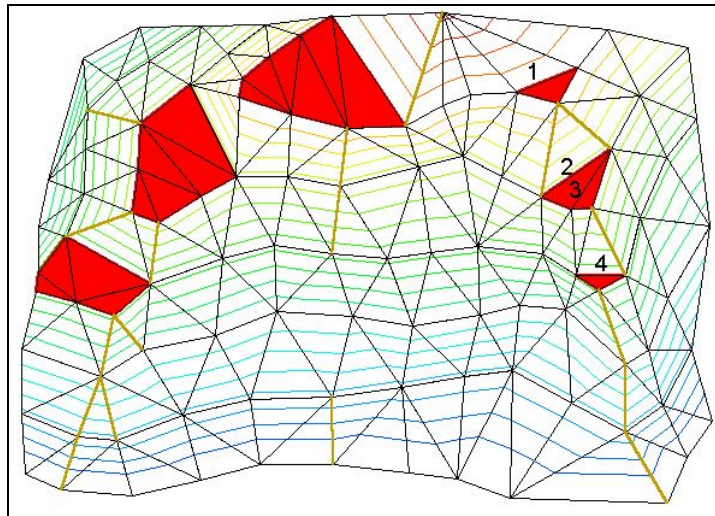


Figure 17-1 Before Swapping

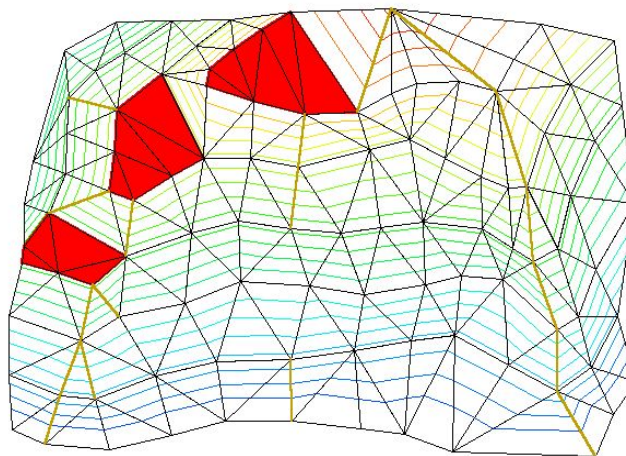


Figure 17-2 After Swapping

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.

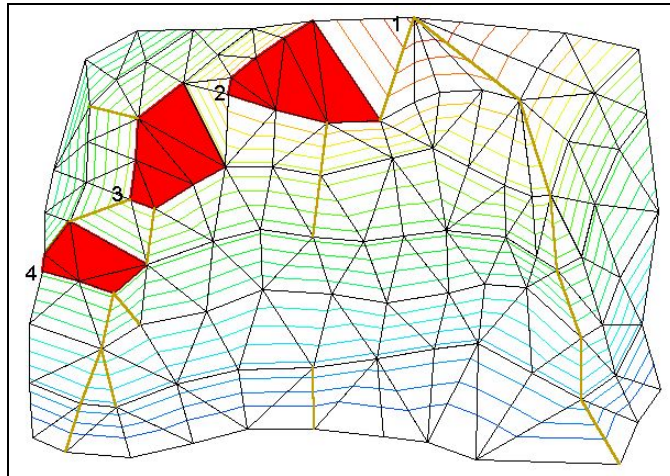


Figure 17-3. *Before Adding Breakline*

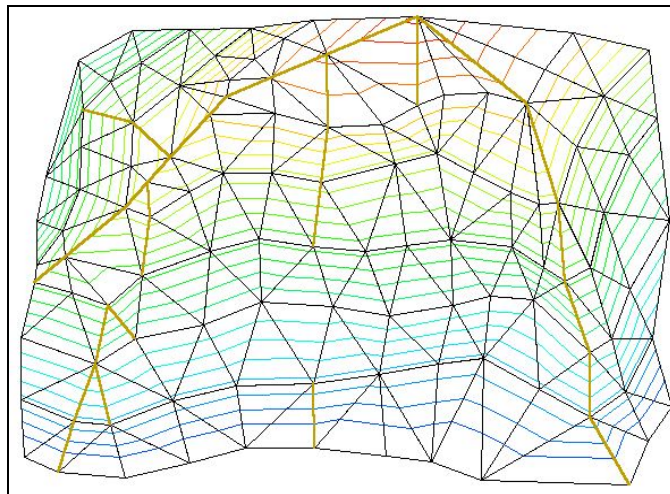


Figure 17-4 *After Adding Breakline*

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.

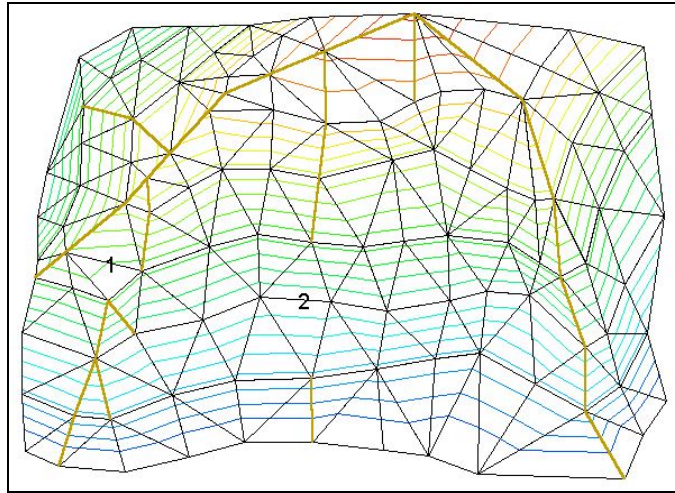



Figure 17-5 Before Removing False Dams.

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.

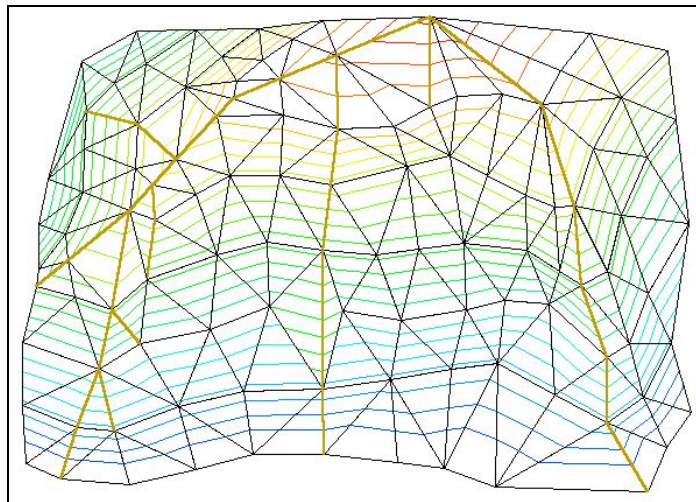



Figure 17-6 After Removing False Dams.

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 channel 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.

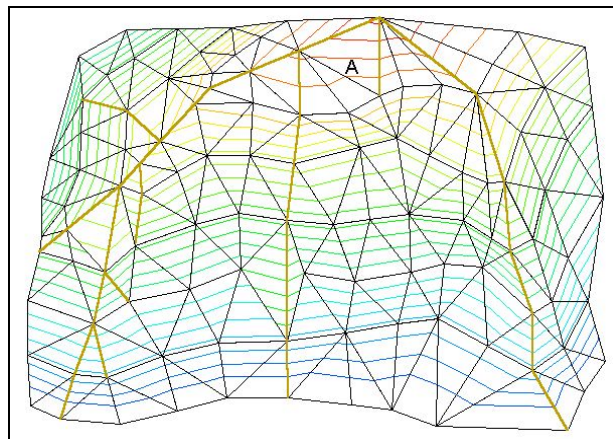


Figure 17-7 Before Adding the Point

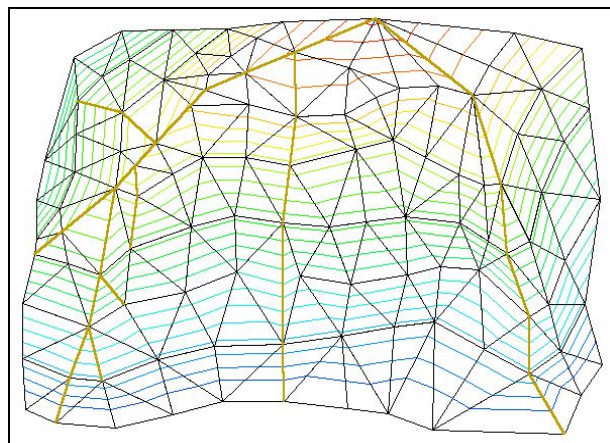



Figure 17-8 After Adding the Point.

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 *Z:* edit field 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. Confirm that you want to delete everything.
3. Select the *Open...* command from the *File* menu.
4. Find and open the file "*auto.tin*".
5. Select the *Triangulate Options...* command from the *TINs* menu.
6. Turn off the *Display triangulation process* command.
7. Click on the *OK* button.

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

8. Select the *Triangulate* command from the *TINs* menu.
9. Select the *Select Boundary Triangles* command from the *TINs* menu.
10. Select the *Select thin boundary triangles only* option.

11. Select *OK*.
12. Select the *Delete* command from the *Edit* menu.
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 Stream* 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 Stream* 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.

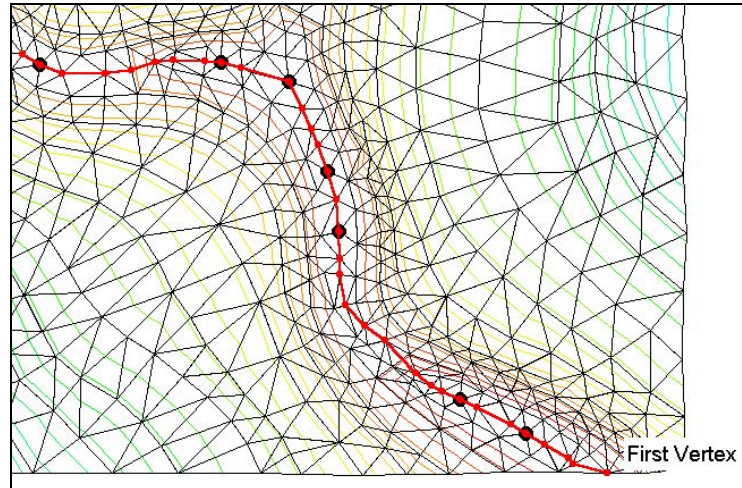


Figure 17-9 Stream Creation

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.

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.

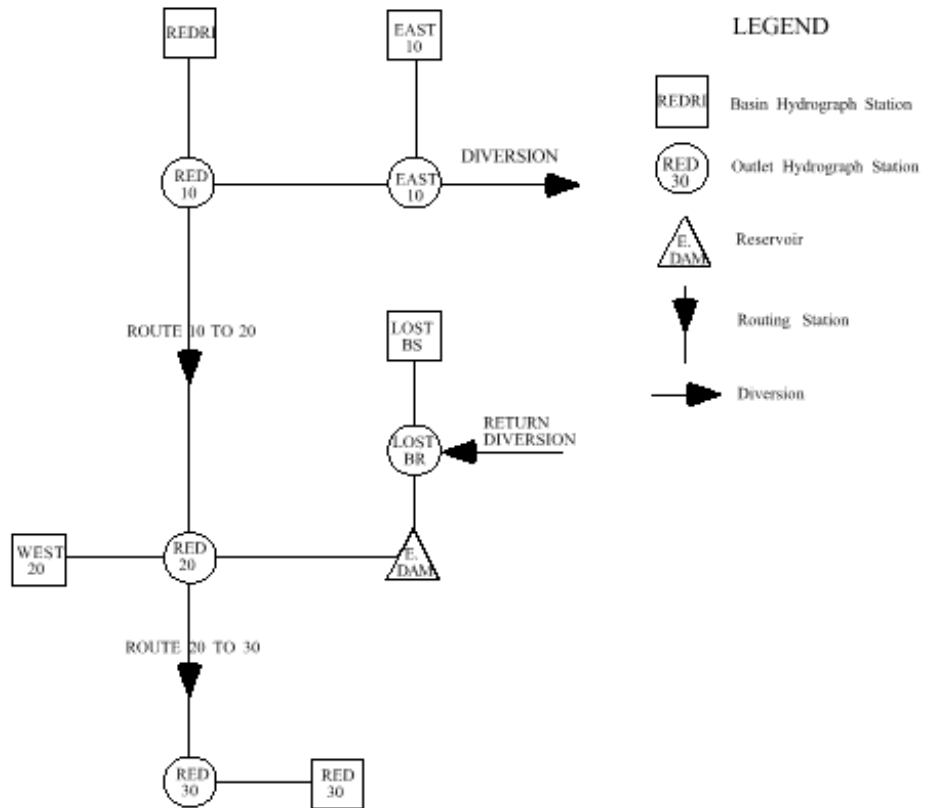


Figure 18-1 Traditional HEC-1 Tree Diagram For Red River Watershed.

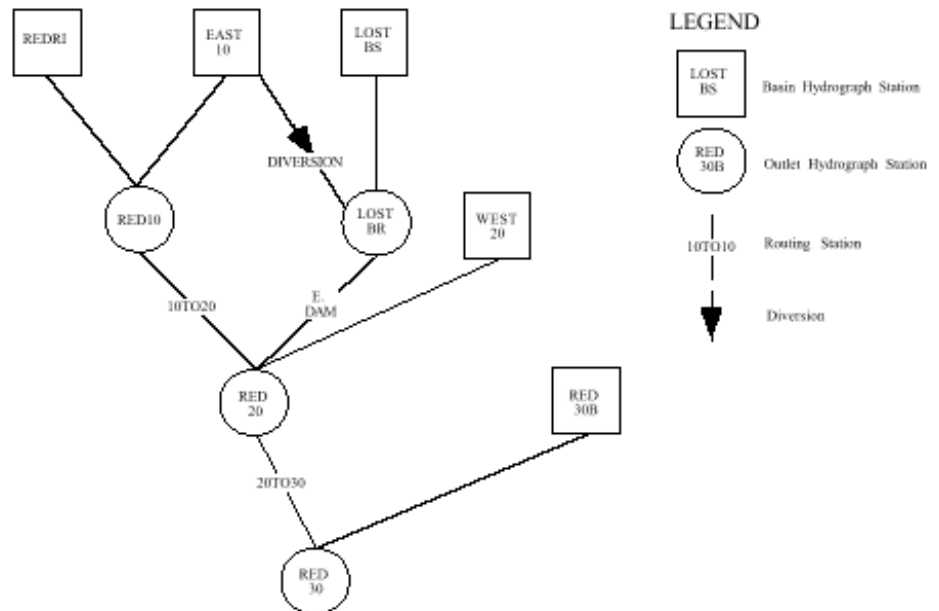


Figure 18-2 HEC-1 Tree Diagram of Red River Watershed Created by WMS.

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.

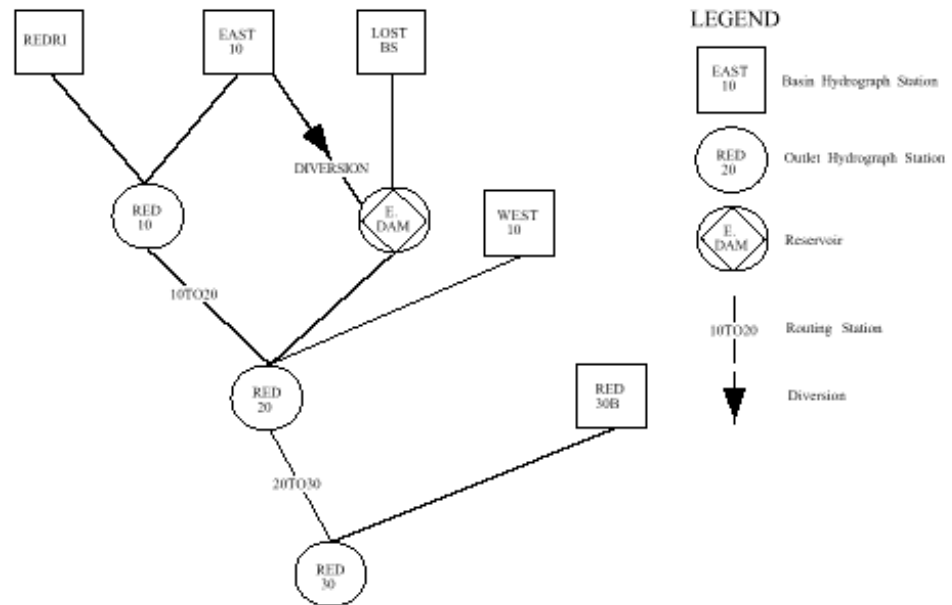


Figure 18-3 Reservoir Used in Place of Reservoir Storage Routing.

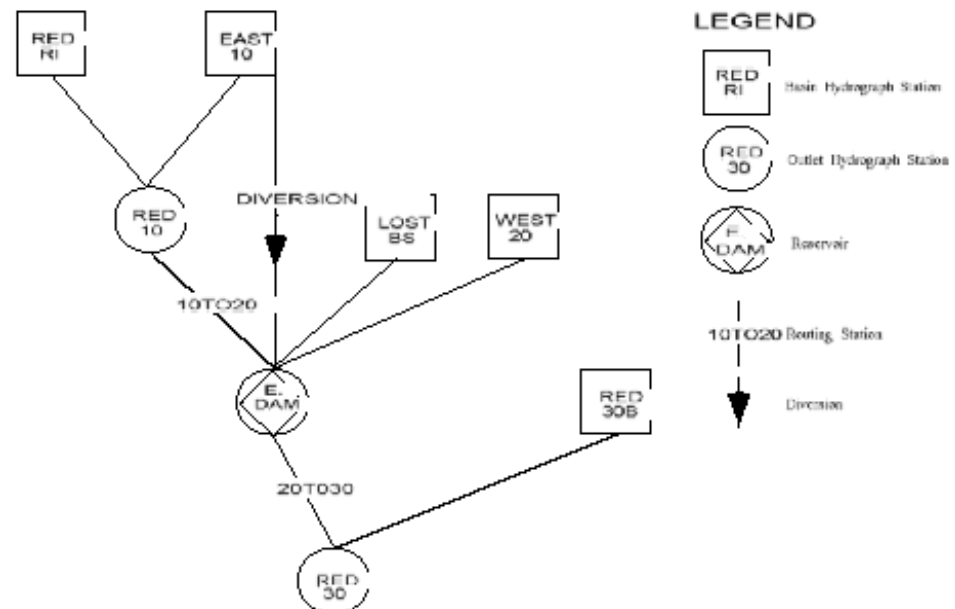



Figure 18-4 LOSTBR Outlet Eliminated with E.DAM Reservoir at RED20.

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 cannot be longer than 6 characters.
- Names must be unique.

To change the outlet names do the following:

1. Click on the outlet point 1C
2. 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 that can be used to define/edit data for hydrograph stations. Only the buttons that 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.
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.
5. Change the combining name from 2C to RED20.
6. Change the routing name from 2R to 20TO30.
7. Repeat the last three steps, changing 3C to LOSTBR, 3R to E.DAM, 4C to RED10, and 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.

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.
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.

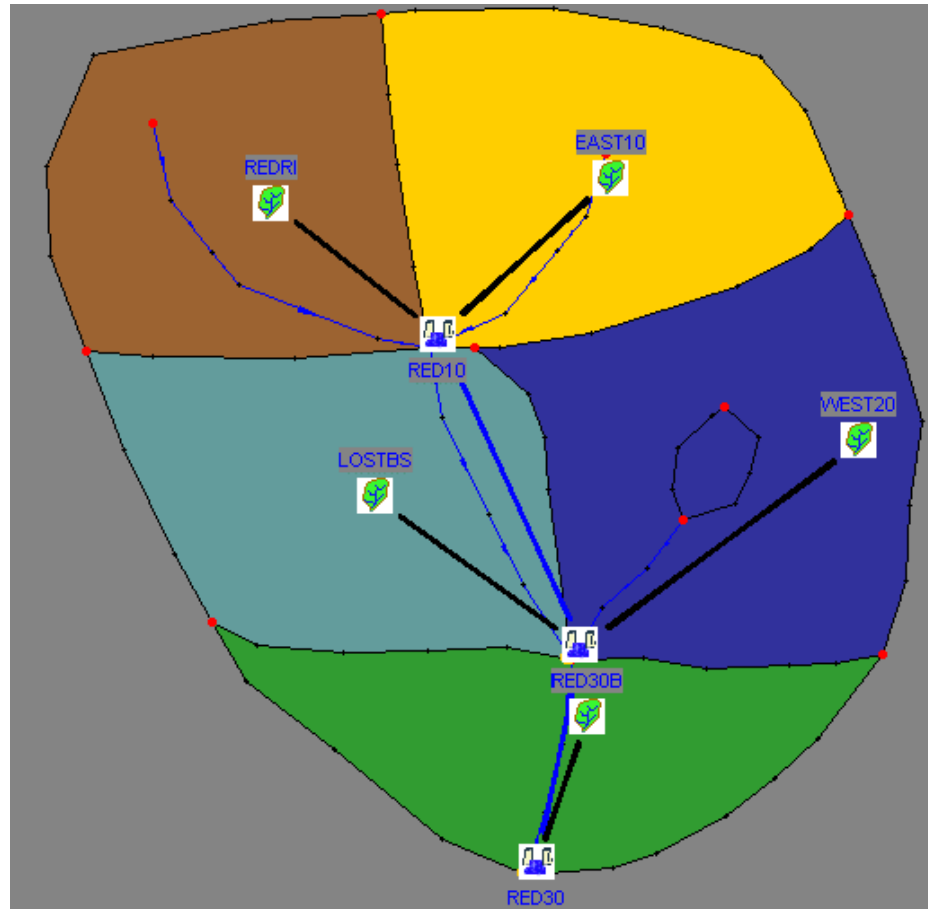




Figure 18-5 Red River Watershed Tree Defined from Feature Objects.

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).
6. Click on the OK button.
7. Click *Done*.

18.5 Job Control Data

Job control data includes the starting time, time step interval, general output specification and other custom features of HEC-1. To modify these 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 $58 \times 15\text{min}$ 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).
6. Repeat the last four 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:

1. Select the *New* button.
2. Change the Name to 400.
3. Toggle off the *Storm Total Station* button, and toggle on the *Temporal distribution station* button.
4. 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.

1. Select the *XY Opts* button.
2. Toggle on the *Unit Time* box.

3. Change the *beginning time* to 435 minutes (This corresponds to 7 hours and 15 minutes and will be converted appropriately when exiting the dialog).
4. Change the *increment* to 60.
5. Select the *Delta button* in the *Distribution* box.
6. Select the *OK* button.
7. Enter the following ten incremental values, starting with time 7:15 in edit field 1: .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 *STRTO* 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.

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		
LOSTBS	.36	62	.5	LU	BEXP=1.4	UC	TC=.8
		63	.5		STRTL=.3		R=1.2
		400	1.0		CNSTL=.04		
EAST10	.66	61	.6	LE	STRKR=.6	US	TP=1.3
		62	.3		DLTKR=1.0		CP=.8
		63	.1		RTIOL=1.0		
REDRI	.82	60	.75	LS	ERAIN=0.0	UD	1.47
		61	.25		CRVNBR=80		
		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. Select the outlet E.DAM if the *Edit HEC-1 Parameters* is still active. 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. Enter the following values, starting with the first table field: 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. Enter the following values, starting with the first table field: 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 (RS)* 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. Enter the following values, starting with the first table field: 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. Enter the following values, starting with the first table field: 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. Select the diversion arrow named DIVERT if the *Edit HEC-1 Parameters* is still active. 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. Enter the following values, starting with the first table field: 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. Enter the following values, starting with the first table field: 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. Select the outlet E.DAM if the *Edit HEC-1 Parameters* is still active. 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.

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* option and click on the *Define* button.
6. Select the *Known volume* option.
7. Select the *SV* toggle to turn it on 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. Enter the following values, starting with the first table field: 21, 100, 205, 325, 955.
11. Click on the *OK* button.
12. Select the *SE* toggle to turn it on 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. Enter the following values, starting with the first table field: 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: ELEV_L=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.

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.

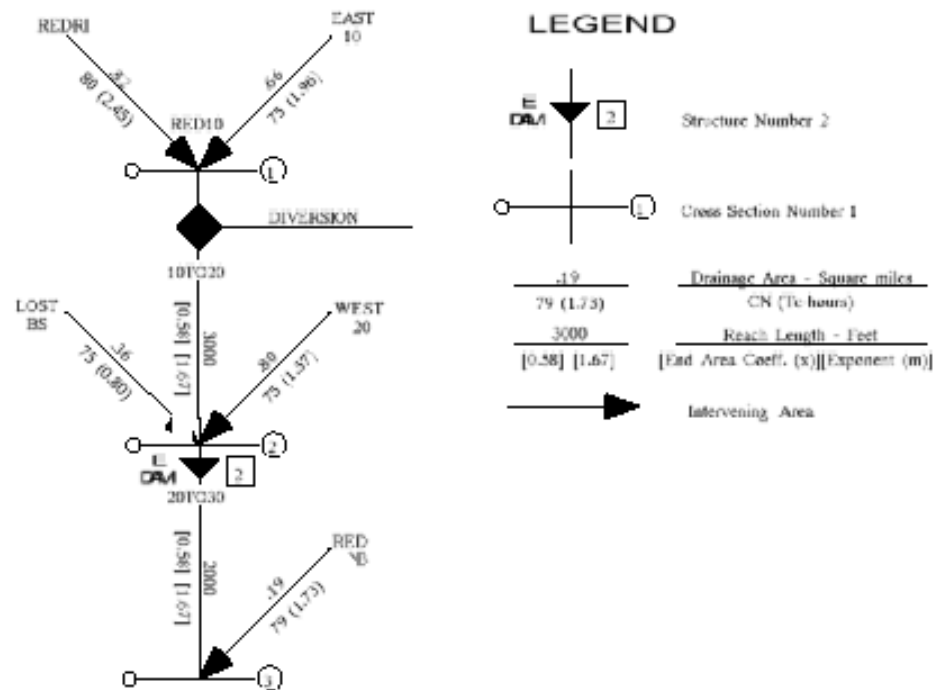


Figure 19-1 Traditional TR-20 Topological Tree Diagram For Red River Watershed.

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.

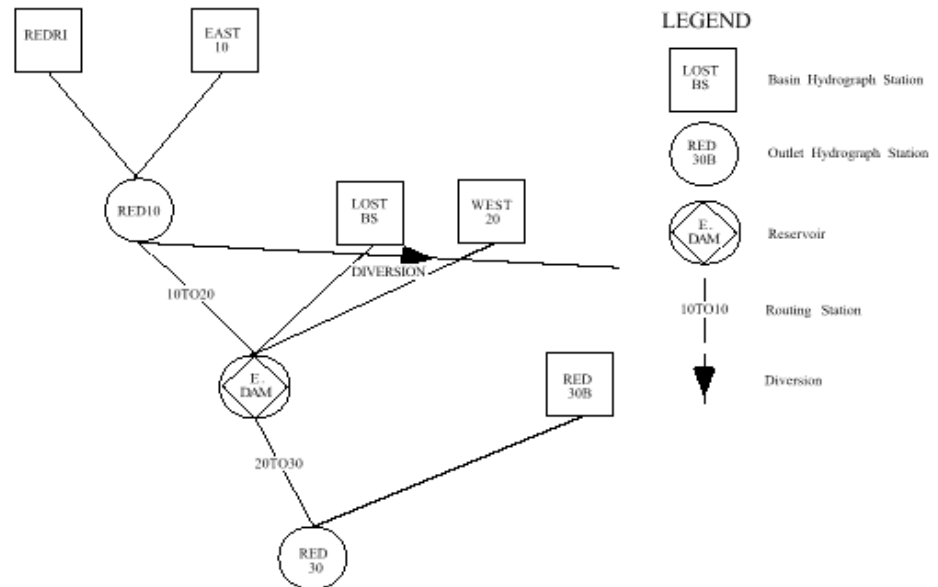



Figure 19-2 LOSTBR Outlet Eliminated with E.DAM Reservoir at RED20.

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 topological 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 cannot 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 that can be used to define/edit data for hydrograph stations. Only the buttons that 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 close 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.
6. Toggle the *Define reach routing* toggle box because there is routing from this outlet.
7. Change the routing name from 2R to 20TO30.
8. Click on the *OK* button.
9. 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 and confirm that you want delete all data.
2. Select the *Open...* command from the *File* menu.
3. Find and open the file named "redritr.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.

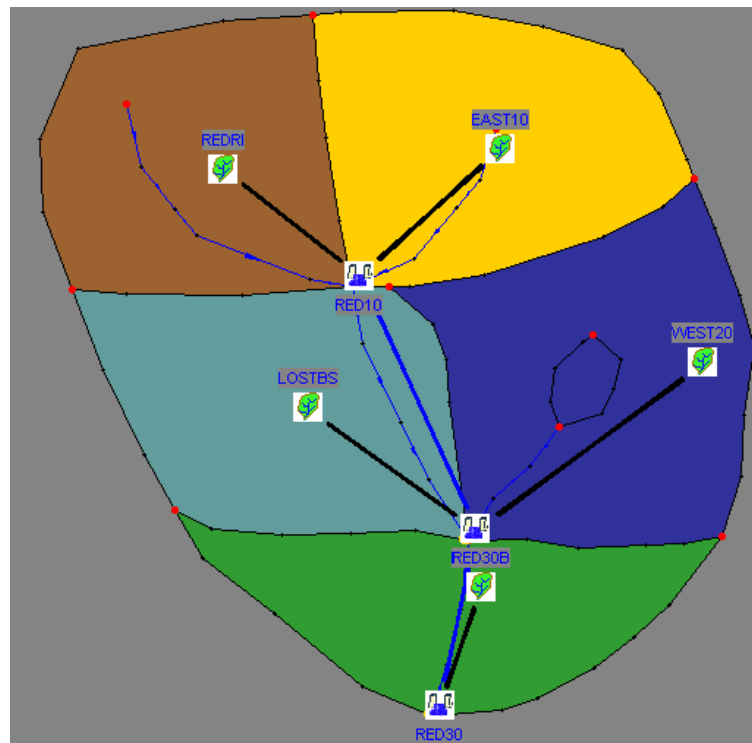


Figure 19-3 Red River Watershed Defined from Feature Objects.

19.5 Job Control Data

Job control data includes the starting time, time step interval and general output specification and other custom features of TR-20. To modify these 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

Table 19.1. Basin Parameters for the Red River Watershed.

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. Select the outlet RED10 if the *Edit TR-20 Parameters* is still active. 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. Select the diversion icon named D1 if the *Edit TR-20 Parameters* is still active. 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. Select the outlet E.DAM if the *Edit TR-20 Parameters* is still active. 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:

Table 19.2 Reservoir Data

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 ensure 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 that 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 “*redriv.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.

HSPF Interface

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


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

20.1 Opening the Watershed and Initializing the HSPF Model

You will open a watershed that has been delineated from a USGS 7.5' DEM and converted to a TIN.

1. Select the *Open* command from the *File* menu.
2. Locate and choose the file named: *littlecotton.wpr*
3. Select *OK*.

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

1. Change to the *Hydrologic Modeling* module .
2. Go to the *HSPF* menu and choose *HSPF Global Options*.
3. Click the *Initialize HSPF Data* button – all fields will become active, indicating that the HSPF model is ready for input data.

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

1. Enter Little Cottonwood Canyon Model in the Title field.
2. Click on the *Start Time* button; enter *Year: 1996, Month: 1, Day: 1, Hour: 0, Minute: 0*.
3. Click *OK*; the start date will appear to the right of the *Start Time* button.
4. Click on the *End Time* button; enter *Year: 1999, Month: 7, Day: 31, Hour: 0, Minute: 0*.
5. Click *OK*; the end date will appear to the right of the *End Time* button.
6. Enter 24 in the *hours* field under the *Time step* heading (leave minutes at 0).
7. Ensure that the *Units flag* is set to English.
8. Change the *Run Flag* to 1; this indicates that HSPF will interpret the data and run the simulation (0 indicates that HSPF will only interpret – error check- the data).
9. Set the *OUTLEV* to 10 (under *Output Levels*); this indicates maximum output to the Error and Warning files. Leave the *SPOUT* flag at 0 (this is only for Special Actions).
10. In the *Files* section, enter *littlecotton* in the *Prefix* field, then click the *Update Filenames* button. This names all the HSPF input/output files associated with this model to the same name. **Note:** you have just indicated that you will be using a file named “littlecotton.wdm” for time series input and output – this file must be created and named appropriately outside of WMS (this has been done for you in this case).

11. Select *OK*.

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

20.2 Importing Land Use and Segmenting the Watershed


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

1. Go to the *File* menu and choose *Import*.
2. Select the file named *littleclanduse.shp* and click *Open*.
3. Click *OK* on the *Import File* dialog to indicate that you want to import a *Feature Object Polygon Shape File*.
4. When the *Import Shapefile Data* dialog appears, click on the *Coverage Options* button. This will allow you to create a new coverage for the land-use data.
5. Click *New* in the *Coverages* dialog, enter *Land Use Data* in the *Name* field, and choose *Land Use* in the *Attribute Set* box.
6. Click *OK* to return to the *Import Shapefile Data* dialog; note that the new coverage you just created is now active and listed by the *Coverage Options* button.
7. Click on the *Attribute Mapping* button under *Polygons*. This will allow you to choose the database fields that correspond to WMS attributes.
8. Choose *LUCODE* in the *Database fields* window and *Land Use* in the *Coverage attributes* window, then click on the *Map* button. You will see a new line appear in the lower window.
9. Choose *PERIMETER=> Basin perimeter* line in the lower window and click *Unmap*. This attribute recognized automatically by WMS, but is not needed in this case.
10. Choose *Done*.
11. Choose *OK* in the *Import Shapefile Data* dialog to read the file.

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

1. Go to the Calculators menu and choose Compute GIS Attributes.
2. Choose *HSPF Segments* from the *Computation* drop-down list.
3. In the central section of the dialog, ensure that *Land use coverage* appears in the data type box and that *Land Use Data* appears as the name of the coverage to be used for the computation.
4. A text file that correlates the land use id to land use attributes (name, perviousness, etc.) must be read in. Click the *Import* button to browse for this file.
5. Choose the file named *littleclanduse.tbl*; Click *Open* to read it into WMS.
6. Choose *OK* in the *Compute GIS Attributes* dialog to compute the HSPF segments.
7. Click *OK* when warned that Continuing will delete all HSPF segment data.


You should now see the segments computed for the watershed displayed. Each colored area represents a segment to be modeled by HSPF. To better view the segments, you will need to change some display options.

1. Switch to the *Map* module .
2. Go to the *Feature Objects* menu and choose *Coverages*.
3. Make sure that the *Land Use Data* coverage is selected, then toggle OFF the *Visible* checkbox.
4. Click *OK*; the land-use coverage is no longer visible.
5. Go to the *Display* menu and choose *Display Options*.
6. Choose the tab entitled *TIN*
7. Toggle ON the *Land use legend*.
8. Click *OK*.



You should now see the land use legend which shows the color and title of each segment in the watershed. You may want to Pan an/or Zoom to avoid having the legend appear over the watershed.

20.3 Aggregating Segments

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

1. Switch to the *Hydrologic Modeling* module .
2. Select the basin icon by clicking on it.
3. Go to the *HSPF* menu and choose *Edit HSPF Attributes*.
4. Note in the *Edit HSPF Attributes* dialog the list of segments in the *Basin Data* window. There are 8 separate segments (or land use classifications) in this watershed..

There are two ways to reduce the number of segments computed by WMS. The first is to delete one segment, then add the area of the deleted segment to another similar segment (aggregate segments). The second is to just delete the segment, then let WMS redistribute the deleted area to the remaining segments (an area-weighted distribution is used).

1. Choose segment entitled *Other Urban Or Build Up* from the *Basin Data* window. Note that the area is about 35 acres – insignificant in this watershed.
2. Click the *Delete* Button.
3. Repeat steps 1 and 2 for the segment entitled *Bare Ground*.
4. Click *Done*.
5. Switch to the *TIN* module .
6. Go to the *Drainage* menu and Choose *Compute Basin Data*; click *OK* on the *Compute Basin Data* dialog to proceed. This will recompute basin data and distribute the deleted segments' areas to the remaining segments.
7. Now you will aggregate some segments. Switch back to the *Hydrologic Modeling* module .
8. Select the basin icon by clicking on it.
9. Go to the *HSPF* menu and choose *Edit HSPF Attributes*. Note that there are only 6 segments listed now.

10. Choose the segment entitled *Mixed Forest*. Note the area of 445.775 acres; this is the area that you will add to the *Evergreen Forest* segment to aggregate the two segments.
11. Click the *Delete* Button to delete segment 6.
12. Choose *Evergreen Forest* segment. Note the area is 12207.985 acres.
13. Enter the new *Area* = 12653. 76 acres (12207.985 + 445.775). You have successfully aggregated two segments.
14. Aggregate segments by deleting the *Bare Exposed Rock* segment and adding its area to the segment *Mixed Tundra*. The new total area will be 1385.752 acres.
15. Choose *Done* to exit the *Edit HSPF Attributes* dialog.

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

20.4 Defining Land Segment Parameters

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

1. Select the basin icon by clicking on it.
2. Go to the *HSPF* menu and choose *Edit HSPF Attributes*.
3. Choose the *Evergreen Forest* segment from the *Basin Data* window.
4. Click the *Define Activities* button.
5. Click the boxes next to *Snow (SNOW)* and *Water (PWAT)* to checkmark them. Note that the buttons become active upon checking the corresponding box.

You have just activated the Snow and Water modules for this pervious land segment. You must now enter the parameters for each active module to allow HPSF to simulate the segment correctly.

1. Click on the *SNOW* button in the *Pervious Land Activity* dialog.

- In the *SNOW* dialog that appears, enter the following values in the appropriate fields (leave other fields at the default values; these values are either generally recommended values or HSPF defaults):

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

Note: these values were extracted from meteorological datasets.

- Now that all values are entered for this segment, click the *Apply Parameters to Segments* button. This will allow you to assign these same parameters to other segments in the model.
- Choose *Shrub & Brush Rangeland* in the *Available Segments* window; move it to the *Selected Segments* window by clicking the → button.
- Repeat step 2 for *Residential* and *Mixed Tundra*.
- Click *OK*. The selected segments will be assigned the same parameters input here.

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

- Click on the *External Sources* button in the *SNOW* dialog. The *Assign External Sources* dialog will appear with a list of the datasets available in the WDM file specified for this model (*littlecotton.wdm*).
- Assign the following datasets as sources by setting the fields to the appropriate values, then clicking the *Assign* button for each line. If you make a mistake, choose the incorrect line in the lower window and click *Delete*.

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

- Since you will add other External Sources to this segment with the *PWAT* module input, you will not assign the External Sources to other segments yet. Click *Done* to return to the *SNOW* dialog.
- Click *OK* to return to the *Pervious Land Activities* dialog.

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

1. Click on the *PWAT* button in the *Pervious Land Activity* dialog.
2. In the *PWATER* dialog that appears, enter the following values in the appropriate fields (leave other fields at the default values):

PWAT-PARM1	PWAT-PARM2	PWAT-PARM3	PWAT-PARM4	PWAT-PARM5	PWAT-STATE1
CSNOFG = On RTOFG = On UZFG = On	LZSN = 5.0 INFILT = 0.48 LSUR = 2258.0 SLSUR = 0.25 KVARY = 0.7 AGWRC = 0.997	DEEPPFR = 0.1 PETMAX = 35.0 PETMIN = 30.0	CEPSC = 0.1 UZSN = 1.0 NSUR = 0.25 INTFW = 3.0 IRC = 0.7 LZETP = 0.8		CEPS = 0.2 UZS = 1.0 LZS = 6.0 AGWS = 1.6 GWVS = 1.9

1. Now that all values are entered for this segment, click the *Apply Parameters to Segments* button. This will allow you to assign these same parameters to other segments in the model.
2. Choose *Shrub & Brush Rangeland* in the *Available Segments* window; move it to the *Selected Segments* window by clicking the → button.
3. Repeat step 2 for *Residential* and *Mixed Tundra*.
4. Click *OK*. The selected segments will be assigned the same parameters input here.

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

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

Member Name	Member subscript	Units	Missing Data	Transformation	Quality flag	Multiplication Factor	Dataset
PETINP		ENGL	UNDF	SAME	0	1.0	23

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

1. Click the *Apply Sources to Segments* button. This will allow you to assign these same External Sources to other segments in the model.
2. Choose *Shrub & Brush Rangeland* in the *Available Segments* window; move it to the *Selected Segments* window by clicking the → button.
3. Repeat step 2 for *Residential* and *Mixed Tundra*.
4. Click *OK*.
5. Click *Done* in the *Assign External Sources* dialog.
6. Click *OK* in the *PWATER* dialog.
7. Click *OK* in the *Pervious Land Activities* dialog.

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

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

20.5 Defining Reach Segment Parameters

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

1. Click on the watershed outlet in the *Graphics Window* (this can be done while the *Edit HSPF Attributes* dialog is still present; double-click the outlet if you have already closed the *Edit HSPF Attributes* dialog). The *Reach/Reservoir Data* section of the *Edit HSPF Attributes* dialog will become active and display the name of the outlet chosen (*Little*).
2. Click on the *Define Activities* button.
3. Check the box to activate the *Hydraulics (HYDR)* module of HSPF for this reach.

4. Click the *HYDR* button in the *Reach/Reservoir Activities* dialog.
5. In the *HYDR* dialog that appears, enter the following values in the appropriate fields (leave other fields at the default values):

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

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

1. Click on the *Define FTABLE* button the *HYDR* dialog.
2. In the *FTABLE* dialog that appears, enter the following values (it is possible to change the number of rows and columns in the *FTABLE*, but the default values will be sufficient in this case):

Depth (ft)	Area (ac)	Volume (ac-ft)	Outflow (cfs)
0.0	0.0	0.0	0.0
0.25	25.99	6.393	13.876
0.5	26.828	12.995	43.954
0.75	27.666	19.807	86.269
1.0	25.50	26.828	139.241
1.25	29.340	34.059	201.950
1.5	30.181	41.499	273.794
1.75	31.019	49.150	354.355
2.0	31.858	57.009	443.334
2.25	35.211	90.540	879.902

1. Click *OK* to save the *FTABLE*.

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

1. Click on the *External Targets* button in the *HYDR* dialog.
2. Set the following fields to the specified values:

Member Name	Member subscript	Units	Access	Aggregation	Multiplication Factor	Quality flag	Transformation Function
RO		ENGL	REPL	Aggregate	1.0	0	AVER

1. Select the Use existing dataset button.
2. Click on the *Select Dataset* button

3. In the *Select Dataset* dialog, scroll down and select dataset 20 (Flow), then click the *OK* button.
4. The Dataset name, type, and number should appear in the *Assign External Targets* dialog.
5. Click *Assign* to add the External Target line to the lower window.
6. Click *Done*.

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

1. Click *OK* in the *HYDR* dialog.
2. Click *OK* in the *Reach/Reservoir Activity* dialog.
3. Click *Done* in the *Edit HSPF Attributes* dialog.

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

20.6 Creating Mass Links

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

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

Volume Name	Volume Group	Volume Member Name	Multiplication Factor	Target Name	Target Group	Target Member Name
PERLND	PWATER	PERO	0.08333333	RCHRES	INFLOW	IVOL

Note: 0.0833333 is the conversion for inches/acre per day → cfs.

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

Volume Name	Volume Group	Volume Member Name	Multiplication Factor	Target Name	Target Group	Target Member Name
IMPLND	IWATER	SURO	0.083333333	RCHRES	INFLOW	IVOL

1. Click Add Link.
2. Change the *Segment Type* (upper left) to RCHRES.
3. Set the fields to the values below:

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

1. Click Add Link.

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

1. Click *OK* to exit the *Mass Link Editor*.

20.7 Saving and Running HSPF Simulation

The last step is to save and run the HSPF simulation. To do this, select the *Run HSPF* command from the HSPF menu. HSPF should run to completion. You will then be able to view the solution.

Coordinate Conversions

Making sure that your data is in the right coordinate system, and specifying the appropriate units is an important part of any modeling project. In this brief tutorial you will learn how you can define the current coordinate system of your input data (DEMs, TINs, feature objects, images, etc.), and how you can transform individual data files so that units among data sources will be consistent. You will also learn how you can transform the coordinate system and/or units of all of the data into another coordinate system.

There are three occasions when transforming coordinates becomes important:

1. You may have a watershed that spans zones within a given coordinate system.
2. You may have data for your watershed model, but the data is in different coordinate systems. For example, you may have elevation data in one coordinate system and land use data in another coordinate system.
3. Your data may be in one coordinate system, but perhaps you want to work in another coordinate system. For example, all the data in your county's GIS database may be in one coordinate system, but your data is in another coordinate system.

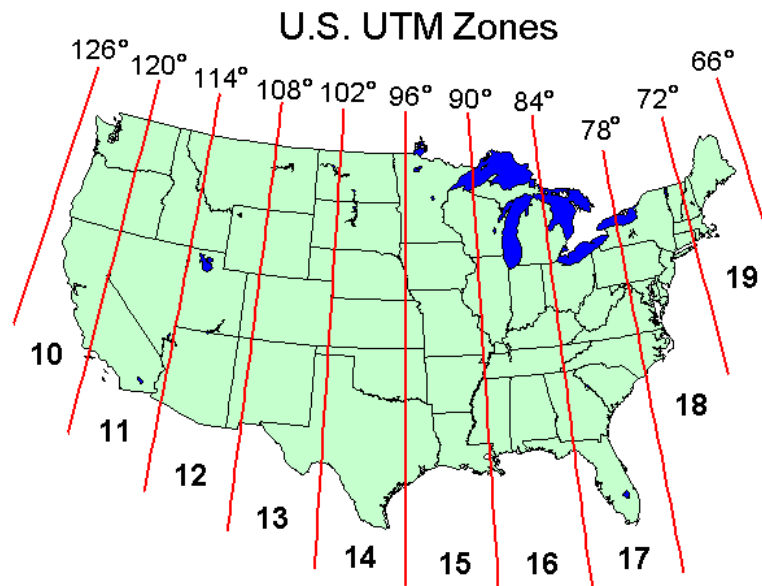
21.1 Background Information on Coordinate Systems

Most data (in the U.S.) comes in either the Universal Transverse Mercator (UTM) or State Plane Coordinate Systems. Because the UTM system provides

a more uniform definition, much of the data distributed on a National or international level comes in this format. However, many of the state and local regulating agencies (for which hydrologic projects are done) require the use of State Plane coordinates. Because of this it is quite often that you will want to convert your data from UTM (or some other coordinate system) to State Plane. An example of doing this is included as part of this tutorial.

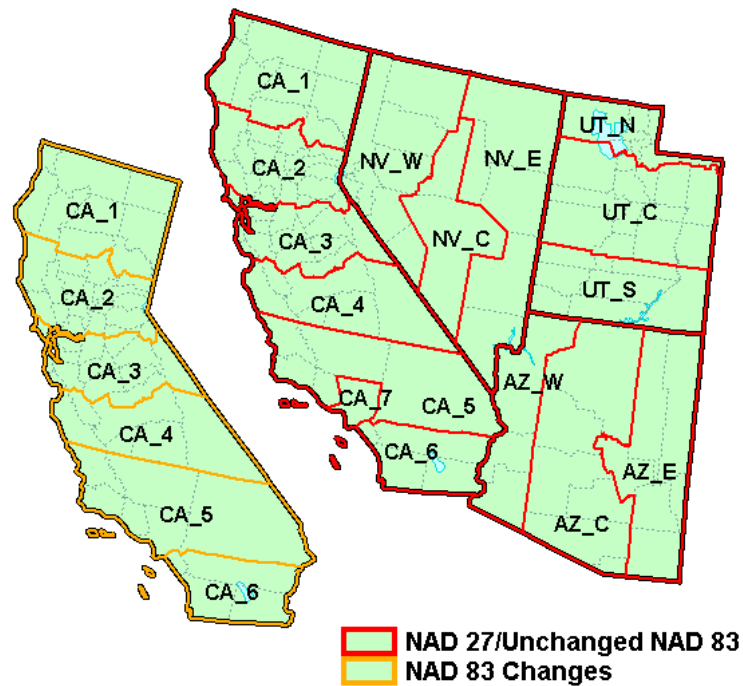
21.1.1 UTM Coordinate System

A UTM (Universal Transverse Mercator) system is a world-wide system defined in meters. The world is divided into 60 zones, 6 degrees of longitude, running from 84°N to 80°S latitude. While Northings (y coordinates) are generally the same from zone to zone, Eastings (x coordinates) start over within each zone. This means that if you have a watershed or other data that happens to straddle a UTM zone boundary, the coordinates from the two data sets (for example two adjacent DEMs) will not be adjacent! Fixing this particular problem is the topic of one of the tutorial sections. The zones that cover the conterminous US are shown in the figure below.



21.1.2 State Plane Coordinate System

The definition used for State Plane coordinates vary from state to state and within states. The figure below illustrates how states in the west (this is similar for eastern states, although many smaller states may have only a single coordinate zone) are divided into separate zones within a state. As with UTM coordinates, XY values (eastings and northings) are defined uniquely only within a given zone and therefore you may be required to convert data from one zone to another to have a consistent coordinate definition within a project area.



21.1.3 Typical Problems Requiring Coordinate Transformation

There are three primary reasons why you might need to transform coordinates from one system to another in a WMS project:

1. If you have a watershed that spans zones within a given coordinate system. For example if you have two DEMs that are needed and one is in UTM zone 12 and the other UTM zone 13 then you will need to transform one of the DEMs into the others coordinate system.
2. You have some data in one coordinate system and other data in another. For example, you may have your DEM data in UTM coordinates and your land use and soils data in State Plane. If you want to calculate a CN value then you will need to have all your data in the same coordinate system or the data will not overlay properly. As with the first case, you can either convert the DEM data to State Plane or you could convert the land use and soil data to UTM.
3. You may just want to convert all of your data from one coordinate system to another. For example, you may have all of the data you need in the UTM coordinate system, but you may be required to provide results in State Plane coordinates. You may also just want to convert data from metric to English, or English to metric units.

There are obviously a lot of different scenarios for which data might need to be transformed from one coordinate system to another, but the three examples

above should give you an idea of the issues that might encounter, and should help you plan your projects accordingly. The examples in this tutorial demonstrate how to convert data from one coordinate system to another in WMS.

21.1.4 The Current Coordinate System

When you read data into WMS it is a good idea to immediately set the current coordinate system using the option from the Edit menu. If you don't then the default of a *local* coordinate system using *feet* units is assumed. As long as your data are in English units (feet) calculations by WMS will be displayed correctly. If your data are in meters then as a minimum you need to change the units to reflect this, particularly at the point where you compute basin data. It is not required to know the actual coordinate system your data is in (i.e. UTM or State Plane, Datum, Zone number, etc.), but it is essential to have the correct units. If you do want to transform coordinate systems then you do need to know the full specifications of the current coordinate system and the one you want to transform to. Because of the way WMS manages coordinate systems there is a certain order that you should follow for properly converting data. The examples in this tutorial illustrate the following three scenarios. These scenarios should give you an understanding of how to approach transformations in WMS:


1. You have data in one coordinate system and want to convert to another.
2. You have data in the same coordinate system, but different zones.
3. You have data in three different coordinate systems and want to get them all into a consistent coordinate system.

21.1.5 Proper Order of Reading Data

Because the coordinate conversion tools operate on all data simultaneously (DEMs, TINs, Imaged, Feature Objects, etc.) it is not possible to read in data that are in two or more coordinate systems and convert one dataset or the other only, or to convert both to the same coordinate system. For this reason, you will often want to read each data set in separately, convert to the proper coordinate system, and save again. Once all data have been converted and saved, all the data may be read into together and the current coordinate system may be defined. One exception to this rule is if you have data in only two coordinate systems, of which one is the desired coordinate system for your project. In this case you would first read in the data you want to convert. You would then convert the data to the desired coordinate system. Finally, you would read in the remaining data that is already in the proper coordinate system.

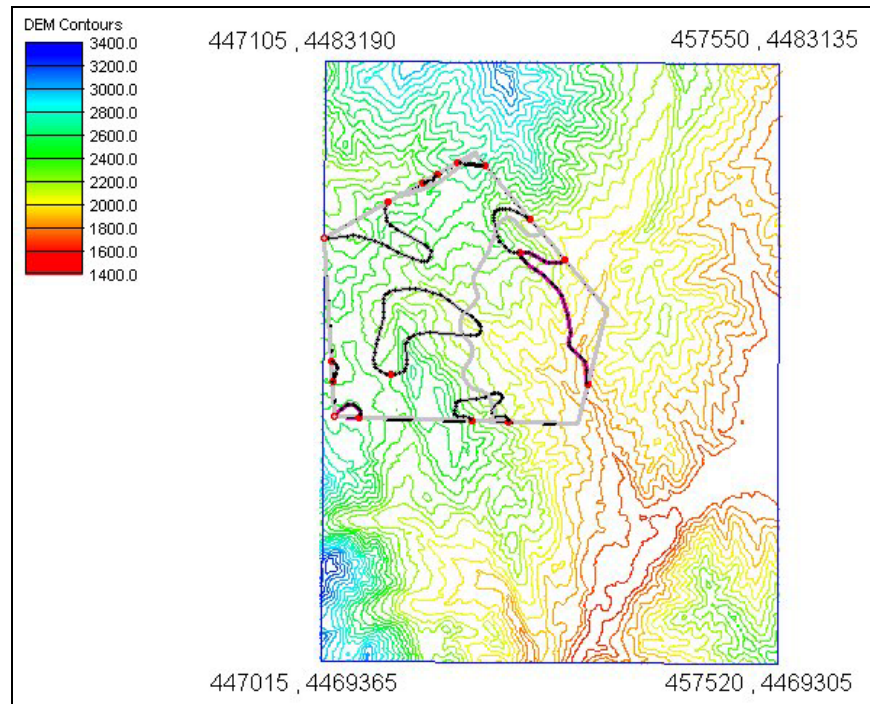
21.2 Converting Data From one Coordinate System to Another

In this first example, you have data in UTM Coordinates. But you want to work in State Plane coordinates.

1. Select the *Open* command from the *File* menu.
2. Find and *Open* the file named *utm.map*.
3. Select the *Import* command from the *File* menu.
4. Find and *Open* the file named *aspen.dem*.
5. Set the *Import* file type to *USGS DEM* and select *OK*.
6. Select *OK* to import the DEM.
7. Select the *DEM* module .
8. Select the *Contour Options* from the *Display* menu.
9. Toggle on the option for the *Display Legend*.
10. Select *OK*.

21.2.1 Setting the Current Coordinate System

If you move your cursor over the data you should notice that the coordinates correspond to the limits as shown in the following figure. Also note the range of elevation values from the contour legend.



1. Select the *Current Coordinates* option from the *Edit* menu.
2. Set the *Horizontal System* to *UTM NAD 27 (US)*.
3. Set the *UTM Zone* to *12 114W to 108W*.
4. Set the horizontal *Units* to *Meters*.
5. Leave the *Vertical System* as *Local*.
6. Set the vertical *Units* to *Meters*.
7. Select *OK*.

21.2.2 Converting from UTM to State Plane

In this case all of our data are in UTM. We could continue to work on our project in this coordinate system. However, perhaps we want to be able to overlay our results with data in another coordinate system (such as State Plane). Or maybe a state or local agency for which we are doing the project requires the data in a different coordinate system—then we would need to convert.

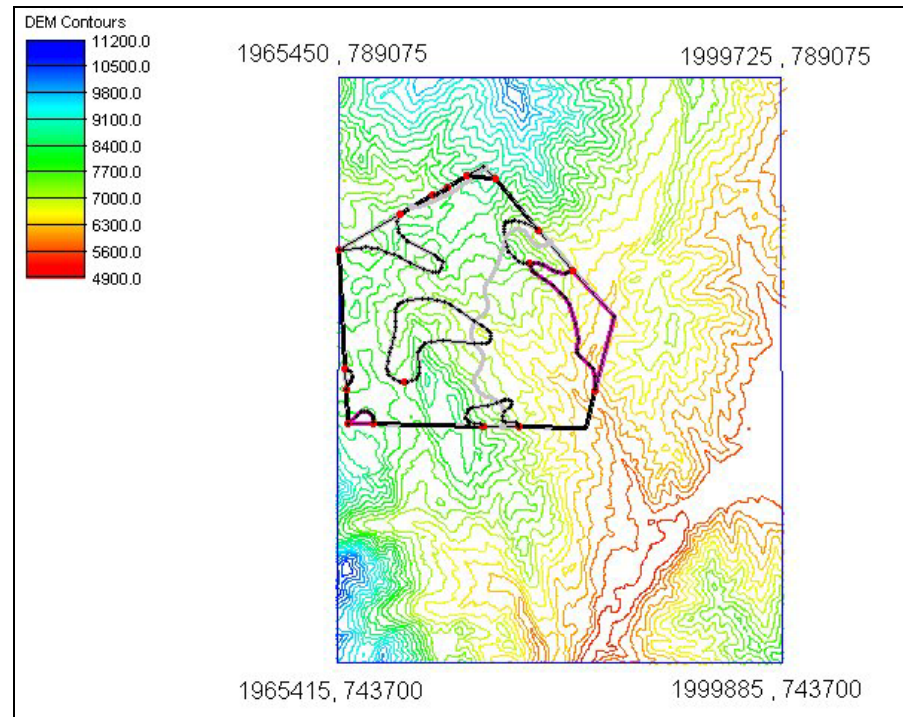
1. Select the *Map* module .

2. Delete the drawing objects on the screen by selecting the *Delete* command from the *Drawing Objects* menu.
3. Select *OK* to delete all the drawing objects.
4. Select the *Coordinate Conversion* command from the *Edit* menu.

You will note that current coordinate system defined in the previous section is already set, but you could (and would have to) define from this dialog if you wanted to.

5. Set the *Horizontal System* to *State Plane NAD 27 (US)*.
6. Set the *St. Plane Zone* to *Utah Central – 4302*.
7. Set the *Horizontal Units* to *U.S. Survey Feet*.
8. Leave the *Vertical System* as *Local*.
9. Set the *Vertical Units* to *U.S. Survey Feet*.
10. Select the *Convert* button.

At first glance it will not appear that anything has changed. However, notice the range of elevations in the legend now correspond to feet or English units rather than metric. Also, if you move your cursor over the model, you will see that the limits now correspond to the limits shown in figure below. Compare this to the previous figure.



21.3 Converting Data to Consistent UTM Zones

In this part of the tutorial you will learn how to convert sets of data straddling a coordinate zone boundary to a consistent coordinate zone so the data can be used together for a project.

The figure below identifies two 7.5-minute quadrangles in Colorado that straddle UTM Zones. Longitude 108 West is divisible by 6—making it a boundary between coordinate zones (zones 12 and 13 in this case – see the UTM zones figure at the beginning of this chapter). This means that even though the Parachute and Rulison quadrangles should be adjacent to each other when we read them into WMS, the X values of Parachute will be much larger than those of Rulison because they “start over” again as we move from zone 12 (west of 108) to Zone 13 (east of 108). So...to use these together in a project we have to either convert Rulison to equivalent coordinates for Zone 12 or Parachute to 13. You should know that “starting over” again doesn’t mean starting with 0 in this case since UTM X coordinates have a “false” easting. This “false” easting is a good thing too...otherwise, converting Parachute to Zone 13 would mean giving it negative X coordinates. This could prove problematic for some calculations.

W108°15'		W108°0'		W107°45'	
LL ,CO	CUTOFF GULCH,CO	MCCARTHY GULCH,CO	RIO BLANCO,CO	HORSE MOUNTAIN CO	R FAL
NT E,CO	CIRCLE DOT GULCH,CO	FORKED GULCH,CO	ANVIL POINTS,CO	RIFLE,CO	SI
NG F,CO	RED PINNACLE CO	PARACHUTE CO	RULISON,CO	NORTH MAMM PEAK,CO	HU ME
ON CK E,CO	DE BEQUE,CO	HOUSETOP MOUNTAIN CO	HAWKHURST CREEK,CO	SOUTH MAMM PEAK,CO	HIG MOU
W108°15'		W108°0'		W107°45'	

1. Select the *New* command from the *File* menu and confirm that you wish to delete all data.
2. Select the *Import* command from the *File* menu.
3. Find and *Open* the file named *1143CEL0.DDF*. (actually you could pick any of the 1143***.DDF files since this is an SDTS formatted USGS DEM).
4. Specify the *Import File* option as *USGS DEM*.
5. Select the *Add* button.
6. Find and *Open* the file named *1144CEL0.DDF*.

At this point you should notice the problem that occurs when two DEMs straddle a UTM boundary – the Rulison quadrangle appears to be to the west (in fact, far to the west) since the X coordinates are starting over as we move into Zone 13. Therefore, you do not want to import these DEMs together. To be able to merge them we will need to do the following:

- Import Rulison (1143***.DDF) and export this file as an Arc/Info ASCII Grid file. After converting Parachute, we will have to export the Parachute DEM to an Arc/Info ASCII Grid file (since to merge two

DEMs they must have the same format). For this reason, we have to convert the Rulison quadrangle to the Arc/Info ASCII grid format.


- Import Parachute, convert the Parachute DEM to Zone 13, and export the Parachute DEM as an Arc/Info ASCII Grid file.
- Import both quadrangles together.
 1. Select the *Parachute, Co – 2400* quadrangle from the *Files* text list box.
 2. Select the *Delete* Button.
 3. Select *OK*.
 4. Select the *Export* command from the *File* menu.
 5. For the *Save as type*: select *Arc/Info Grid (*.asc)*.
 6. Change the filename to *rulison.asc*.
 7. Select the *Save* button.

We have now converted the Rulison DEM to the Arc/Info ASCII grid format. Then, when we are finished converting the other DEM, we can import the two DEMs together. We must convert the format of the Rulison DEM since it is not possible in WMS to save a DEM in the USGS SDTS format; nor is it possible to read in two DEMs of different formats.


1. Select the *New* command from the *File* menu and confirm that you wish to delete all data.
2. Select the *Import* command from the *File* menu.
3. Find and *Open* the file *1144CELO.DDF*.
4. Select the *Import File Type* to be *USGS DEM* and select *OK*.
5. Select the *OK* button to import the DEM.
6. Select the *Coordinate Conversion* command from the *Edit* menu.
7. Select the *Current Coordinates* button and define the DEM coordinates to be *UTM NAD 27 (US), Zone 12*, with units of *Meters* for both horizontal and vertical (leave the vertical system as *Local*).
8. Select *OK*.

9. Now set the conversion coordinate system to *UTM NAD 27 (US), Zone 13*, and units of *Meters* for both horizontal and vertical (again leave the vertical system to *Local*).
10. Select the *Convert* button.
11. Select the *Export* command from the *File* menu.
12. For the *Save as type*: select *Arc/Info Grid (*.asc)*.
13. Change the filename to *parachute.asc*.
14. Select the *Save* button.

We now have the Parachute DEM saved in UTM Zone 13 coordinates so it will be placed geographically next to the Rulison DEM. With both DEMs now saved in the Arc/Info ASCII grid file format, we can read them in together.

1. Select the *New* command from the *File* menu and confirm that you wish to delete all data.
2. Select the *Import* command from the *File* menu.
3. Find and *Open* the file named *rulison.asc*.
4. Select the *Import file type* to be *Arc/Info Grid File* and select *OK*.
5. Select the *Add* button.
6. Find and *Open* the file named *parachute.asc*.
7. Select the *OK* button to import both DEMs.
8. Select the *DEM* module .
9. Select the *Display Options* command from the *Display* menu.
10. Toggle on the display of *No data cells*.

You will see a small gap of undefined elevations near the bottom of the two DEMs. This is a common occurrence when transforming grids and so you must now interpolate across the boundary to fill in the gap.


1. Select the *Select DEM cells* tool .
2. Drag a box around the no data cells in the gap between the two DEMs.
3. Select the *Fill NODATA Cells* command from the *DEMs* menu.

You have now completed the transformation of the Parachute quadrangle from UTM Zone 12 coordinates to UTM Zone 13 coordinates so that it could be used together with the Rulison quadrangle to delineate a watershed. This same process would apply no matter what type of data you have (DEMs, TINs, Feature Objects, etc.). Remember though in the case of the DEMs we had to first read in the Rulison quadrangle and export it as an Arc/Info ASCII grid. Then, after converting the Parachute quadrangle, we were able to read the two in together.

21.4 Converting Data from Multiple Coordinate Systems

In this example, the coordinate system we want to work in is the UTM coordinate system. We have a DEM already in the correct UTM coordinates, but our land use file is in State Plane while our soils file is in Geographic (latitude – longitude) coordinates.

We will first read the soils file in, convert the file, and then save the file. Second, we will read the land use file in and convert it. At this point, all three data sets are in the UTM coordinate system. Then, we can read the soil file back in and then import the DEM.

1. Select the *Map* module .
2. Select the *Import* command from the *File* menu.
3. Find and *Open* the file named *aspensogeographic.shp*.
4. Make sure the *Import file type* is *Feature Object Polygon Shape File* and then select *OK*.
5. Select the *Coverage Options* button.
6. Set the *Attribute Set* to *Soil Type*.
7. Confirm that you want to change the coverage type.
8. Select *OK*.
9. Select *OK* to import the soils.

You have now imported the soils file. If you move your cursor over the soils data and notice the x-y-z coordinate tracking window you should see that these data are in Geographic or latitude longitude coordinates. Negative x values are indicative of longitudes in the western hemisphere.

1. Select the *Coordinate Conversion* command from the *Edit* menu.

2. Select the *Current Options* button.
3. Change the *Horizontal System* to *Geographic NAD 27 (US)* and the vertical *Units* to be *Meters*, leaving all other settings the same.
4. Select *OK*.
5. For the *Convert to* options set the *Horizontal Units* to *UTM NAD 27 (US)* and *Zone 12 114W to 108W*. Set both the *Horizontal Units* and the vertical *Units* to *Meters*.
6. Select the *Convert* button.

Now we will save the soils data to a .map file so that we can read it back in after converting the land use data.

1. Select the *Save* command from the *File* menu.
2. Set the *Save file type* to be *Feature Object Files (*.map)*.
3. Enter *aspensoutm.map* for the file name and select the *Save* button.
4. Select *New* from the *File* menu and confirm that you wish to delete everything.
5. Select *Import* from the *File* menu.
6. Find and *Open* the file *aspenlustplane.shp*.
7. Confirm that the input file type is a shapefile and select *OK*.
8. Select the *Coverage Options* button.
9. Set the *Attribute type* to *Land Use*.
10. Select *OK* to confirm that you wish to change the attribute set.
11. Select *OK* for the *Coverages* dialog.
12. Select *OK* to import the land use file.
13. Select *Coordinate Conversion* from the *Edit* menu.
14. Select the *Current Options* button.
15. Set the *Horizontal System* to *State Plane NAD 27 (US)*, the *St. Plane Zone* to *Utah Central – 4302*, the *Horizontal Units* to *U.S. Survey Feet*, and the vertical *Units* to *U.S. Survey Feet*.
16. Select *OK*.

17. For the *Convert to* options set the *Horizontal Units* to *UTM NAD 27 (US)* and *Zone 12 114W to 108W*. Set the *Horizontal Units* to *Meters* and the vertical *Units* to *Meters*.
18. Select the *Convert* button.

Now that the land use is in UTM coordinates read in the soil map file and the DEM.

11. Select the *Open* command from the *File* menu.
12. Find and *Open* the file named *aspensoutm.map* (the file you saved for the soils above).
13. Select the *Append* button so that it adds this coverage to the existing feature object data.
14. Select the *Import* command from the *File* menu.
15. Find and *Open* the file named *aspen.dem*.
16. Set the *Import* file type to *USGS DEM* and select *OK*.
17. Select *OK* to import the DEM.

21.5 Conclusion

Through these three examples, you should now have a better idea of how the coordinate conversion tools can be used in WMS. These examples represent typical problems you may encounter while collecting geographic data for use in WMS.