WMS User Manual (v9.1)

The Watershed Modeling System

PDF generated using the open source mwlib toolkit. See http://code.pediapress.com/ for more information.
PDF generated at: Fri, 22 Mar 2013 17:35:52 UTC

Contents

Articles

References

1. Introduction

Introduction to WMS

WMS is a comprehensive environment for hydrologic analysis. It was developed by the Enironmental Modeling Research Laboratory [1] of Brigham Young University [2] in cooperation with the U.S. Army Corps of Engineers Waterways Experiment Station and is currently being developed by Aquaveo LLC [3].

WMS offers state of the art tools to perform automated basin delineation and to compute important basin parameters such as area, slope and runoff distances. It also serves as a graphical user interface for several hydraulic and hydrologic models. With its management of coordinate systems, WMS is capable of displaying and overlaying data in real world coordinates. The program also provides many display tools for viewing terrain surfaces and exporting images for reports and presentations.

All of the former models (HEC-1, TR-55, TR-20, Rational, NFF, HMS, OC Hydrology, MODRAT) are still supported with version 8.1, with the additions of a complete SWMM interface and a much improved and fully functional interface to a spatially distributed model, GSSHA (formerly CASC2D). The WMS hydraulic interface was also updated in WMS 8.1 making it compatible with HEC-RAS 4.0 beta. The RAS model can be run as steady state or unsteady and results used to delineate floodplain extents and animations of flood waves for complete flood plain analysis. WMS 8.1 has also integrated an improved hydraulic toolbox and incorporated the latest release of the widely used Federal Highways culvert design model, HY-8.

Related Topics

- WMS Basic Modeling Concepts
- DEM Guidelines
- TIN Guidelines
- Feature Object Guidelines
- ArcView Data Guidelines

References

[1] http://www.emrl.byu.edu/
[2] http://www.byu.edu/
[3] http://www.aquaveo.com/

WMS Tutorials

A rich set of step-by-step tutorials has been developed to aid in learning how to use WMS.

The tutorials are in PDF format. They are installed in the "docs" directory in the folder where WMS is installed. They can also be accessed by clicking on the links below.

The tutorials are listed below by subject and are **not** necessarily listed in the suggested order of completion. Some tutorials assume a basic knowledge of WMS and some build on other tutorials. When this is the case the tutorial itself will state which tutorials you should complete beforehand.

Many of the tutorials have files that are needed to run the tutorial. These files can be found in the "tutorial" folder where WMS was installed, or can be downloaded here.

Printing: To print the tables below, click here and select "Printable version" from the wiki toolbox on the left.

General WMS

Category	Tutorial (PDF)	Required Files
Introduction	• Introduction to WMS [1] • Images [2] • Basic Feature Objects [3] • Advanced Feature Objects [4]	• [5] • [6] • [7] • [8]
Editing Elevations	• DEM Basics [9] • Using TINs [10]	• [11] • [12]
Watershed Modeling	• DEM Delineation [13] • Advanced DEM Delineation Techniques [14] • Time of Concentration Calculations and Computing a Composite CN [15]	• [16] • [5] • [17], [18]

Hydrologic Models

Category	Tutorial (PDF)	Required Files
HEC-1	• HEC-1 Interface [19]	• [20]
HEC-HMS	• HEC-HMS Interface [21]	
Rational Method	• Rational Method Interface [22]	• [23]
NSS (National Streamflow Statistics)	• National Streamflow Statistics Program (NSS) Interface [24]	• [17]
Maricopa County (Phoenix, AZ, USA) Hydrology	• Maricopa County: NSS and HEC-1 [25] • Maricopa County: Master Plan – Creating a Predictive HEC-1 Model [26]	• [27]
MODRAT	• MODRAT Interface – Schematic [28] • MODRAT Interface (GIS-based) [29] • MODRAT Interface (Map-based) [30]	• [31]

Orange County Rational and Unit Hydrograph Methods	• Orange County Rational Method [32] • Orange County Unit Hydrograph [33] • Orange County Small Area Hydrograph [34] • Orange County Hydrology – Using GIS Data [35] • Orange County Rational Method – GIS [36] • Orange County Unit Hydrograph – GIS [37]	• [38]
Spatial Hydrology-HMS ModClark and GSSHA	• HEC-HMS Distributed Parameter Modeling with the MODClark Transform [39] • Developing a GSSHA Model using the Hydrologic Modeling Wizard [40] • Using NEXRAD Rainfall Data in an HEC-HMS (MODClark) Model [41] • Using NEXRAD Rainfall Data in GSSHA [42]	• [43]

Distributed Hydrologic Modeling using GSSHA

Category	Tutorial (PDF)	Required Files
WMS Basics	• Loading DEMs, Contour Options, Images, and Coordinate Systems [44] • Watershed Delineation using DEMs and 2D Grid Generation [45] • Creating Feature Objects and Mapping their Attributes to the 2D Grid [46]	• Personal [47], GSSHA Images [48] • Personal [47], Raw Data [49] • Personal [47], GSSHA Tables [50], Raw Data [49], Watershed Delineation Files [51]
GSSHA Modeling	• GSSHA Initial Overland Flow Model Setup [52] • Correcting Overland Flow Problems [53] • Infiltration [54] • Stream Flow [55] • Developing a GSSHA Model Using the Hydrologic Modeling Wizard in WMS [56] • Post-Processing and Visualization of GSSHA Model Results [57]	• Personal [47], Raw Data [49], GSSHA Basic Model [58] • Personal [47], GSSHA Tables [50], GSSHA Digital Dams [59] • Personal [47], Raw Data [49], GSSHA Tables [50], GSSHA Digital Dams [59], GSSHA Infiltration [60] • Personal [47], GSSHA Tables [50], GSSHA Infiltration [60] • Personal [47], GSSHA Tables [50], Raw Data [49] • Personal [47], Raw Data [49], GSSHA Visualization [61]
Applications	• Precipitation Methods in GSSHA [62] • Analyzing the Effects of Land Use Change (Part - I) [63] • Analyzing the Effects of Land Use Change (Part - II) [64] • Long Term Simulations in GSSHA [65] • Simulating Sediment Transport [66] • Simulating Constituent Transport [67] • Overland Flow Boundary Conditions in GSSHA [68] • Snowmelt Modeling in GSSHA [69]	• Personal [47], GSSHA Tables [50], Raw Data [49], GSSHA Precipitation [70] • Personal [47], GSSHA Tables [50], GSSHA Scenarios [71] • Personal [47], GSSHA Tables [50], GSSHA Scenarios [71] • Personal [47], Raw Data [49], GSSHA Long Term [72] • Personal [47], GSSHA Sediment [73] • Personal [47], GSSHA Contaminants [74] • Personal [47], GSSHA Boundary Conditions [75] • Personal [47], GSSHA Long Term [72]
Calibration	• Manual Calibration of GSSHA models [76] • Stochastic Simulations of GSSHA models [77] • Automated Calibration of GSSHA models [78]	• Personal [47], GSSHA Calibration [79] • Personal [47], GSSHA Calibration [79] • Personal [47], GSSHA Calibration [79]
Groundwater Modeling	• Groundwater Modeling in GSSHA [80] • Advanced Groundwater Modeling in GSSHA [81] • Subsurface Tile and Storm Drains [82]	• Personal [47], Raw Data [49], GSSHA Groundwater [83] • Personal [47], GSSHA Groundwater [83] • Personal [47], GSSHA Subsurface [84]

Hydraulics and Floodplain Modeling

Category	Tutorial (PDF)	Required Files
HEC-RAS	• HEC-RAS Analysis [85] • HEC-RAS – Managing Cross Sections [86]	• [87] • [88]
Floodplain Delineation	• Floodplain Delineation [89] • Stochastic Modeling Using HEC-1 and HEC-RAS [90]	• [91] • [92]
SMPDBK (Simplified Dam Break)	• Simplified Dam Break [93]	• [94]
HY-8 Culvert Modeling Wizard	• HY-8 Modeling Wizard [95]	• [96]
Modeling with FHWA's Hydraulic Toolbox	• Modeling with the Hydraulic Toolbox [97]	• [98]

Storm Drain Modeling

Category	Tutorial (PDF)	Required Files
EPA-SWMM and xpswmm	• SWMM Modeling [99]	• [100]
FHWA Storm Drain	• Storm Drain: Rational Design [101] • Storm Drain: Hydrographic Design [102]	
FHWA HY-12	• HY-12: Rational Design [103]	• [104]

Water Quality Modeling

Category	Tutorial (PDF)	Required Files
HSPF	• HSPF Interface [105]	• [106]
CE-QUAL-W2	• CE-QUAL-W2 Interface [107]	• [108]

Related Topics

- Model Documentation
- References

References

[1] http://wmstutorials-9.1.aquaveo.com/1%20Introduction-Intro.pdf
[2] http://wmstutorials-9.1.aquaveo.com/2%20Introduction-Images.pdf
[3] http://wmstutorials-9.1.aquaveo.com/3%20Introduction-BasicFeatureObjects.pdf
[4] http://wmstutorials-9.1.aquaveo.com/6%20Introduction-AdvancedFeatureObjects.pdf
[5] http://wmstutorials-9.1.aquaveo.com/demedit.zip
[6] http://wmstutorials-9.1.aquaveo.com/images.zip
[7] http://wmstutorials-9.1.aquaveo.com/feature.zip
[8] http://wmstutorials-9.1.aquaveo.com/featureadv.zip
[9] http://wmstutorials-9.1.aquaveo.com/4%20EditingElevations-DEMBasics.pdf
[10] http://wmstutorials-9.1.aquaveo.com/5%20EditingElevations-UsingTINs.pdf
[11] http://wmstutorials-9.1.aquaveo.com/dembasics.zip
[12] http://wmstutorials-9.1.aquaveo.com/tins.zip
[13] http://wmstutorials-9.1.aquaveo.com/7%20WatershedModeling-DEMDelineation.pdf

[14] http://wmstutorials-9.1.aquaveo.com/8%20WatershedModeling-AdvancedDEMDelineationTechniques.pdf
[15] http://wmstutorials-9.1.aquaveo.com/9%20WatershedModeling-TimeConcAndCN.pdf
[16] http://wmstutorials-9.1.aquaveo.com/demdelin.zip
[17] http://wmstutorials-9.1.aquaveo.com/nss.zip
[18] http://wmstutorials-9.1.aquaveo.com/tr-55.zip
[19] http://wmstutorials-9.1.aquaveo.com/10%20WatershedModeling-HEC1Interface.pdf
[20] http://wmstutorials-9.1.aquaveo.com/hec-1.zip
[21] http://wmstutorials-9.1.aquaveo.com/11%20WatershedModeling-HECHMSInterface.pdf
[22] http://wmstutorials-9.1.aquaveo.com/12%20WatershedModeling-RationalMethodInterface.pdf
[23] http://wmstutorials-9.1.aquaveo.com/rational.zip
[24] http://wmstutorials-9.1.aquaveo.com/13%20WatershedModeling-NSSInterface.pdf
[25] http://wmstutorials-9.1.aquaveo.com/14%20WatershedModeling-MaricopaNSS.pdf
[26] http://wmstutorials-9.1.aquaveo.com/15%20WatershedModeling-MaricopaHEC1.pdf
[27] http://wmstutorials-9.1.aquaveo.com/MARICOPA.zip
[28] http://wmstutorials-9.1.aquaveo.com/16%20WatershedModeling-ModratSchematic.pdf
[29] http://wmstutorials-9.1.aquaveo.com/17%20WatershedModeling-ModratGIS.pdf
[30] http://wmstutorials-9.1.aquaveo.com/18%20WatershedModeling-ModratMap.pdf
[31] http://wmstutorials-9.1.aquaveo.com/MODRAT.zip
[32] http://wmstutorials-9.1.aquaveo.com/19%20WatershedModeling-OCRational.pdf
[33] http://wmstutorials-9.1.aquaveo.com/20%20WatershedModeling-OCHydrograph.pdf
[34] http://wmstutorials-9.1.aquaveo.com/21%20WatershedModeling-OCSmallHydrograph.pdf
[35] http://wmstutorials-9.1.aquaveo.com/22%20WatershedModeling-OCUsingGIS.pdf
[36] http://wmstutorials-9.1.aquaveo.com/23%20WatershedModeling-OCRationalGIS.pdf
[37] http://wmstutorials-9.1.aquaveo.com/24%20WatershedModeling-OCHydrographGIS.pdf
[38] http://wmstutorials-9.1.aquaveo.com/OrangeCounty.zip
[39] http://wmstutorials-9.1.aquaveo.com/36%20Spatial-HMSModClark.pdf
[40] http://wmstutorials-9.1.aquaveo.com/37%20Spatial-GSSHAWizard.pdf
[41] http://wmstutorials-9.1.aquaveo.com/38%20Spatial-NexradModClark.pdf
[42] http://wmstutorials-9.1.aquaveo.com/39%20Spatial-NexradGSSHA.pdf
[43] http://wmstutorials-9.1.aquaveo.com/spatial.zip
[44] http://wmstutorials-9.1.aquaveo.com/40%20Gssha-WMSBasics-DEMImageCoords.pdf
[45] http://wmstutorials-9.1.aquaveo.com/41%20Gssha-WMSBasics-DEMDelineation.pdf
[46] http://wmstutorials-9.1.aquaveo.com/42%20Gssha-WMSBasics-MappingFeatures.pdf
[47] http://wmstutorials-9.1.aquaveo.com/Personal.zip
[48] http://wmstutorials-9.1.aquaveo.com/GsshaImages.zip
[49] http://wmstutorials-9.1.aquaveo.com/RawData.zip
[50] http://wmstutorials-9.1.aquaveo.com/tables.zip
[51] http://wmstutorials-9.1.aquaveo.com/WatershedDel.zip
[52] http://wmstutorials-9.1.aquaveo.com/43%20Gssha-Modeling-InitialSetup.pdf
[53] http://wmstutorials-9.1.aquaveo.com/44%20Gssha-Modeling-CorrectingOverland.pdf
[54] http://wmstutorials-9.1.aquaveo.com/45%20Gssha-Modeling-Infiltration.pdf
[55] http://wmstutorials-9.1.aquaveo.com/46%20Gssha-Modeling-StreamFlow.pdf
[56] http://wmstutorials-9.1.aquaveo.com/47%20Gssha-Modeling-Wizard.pdf
[57] http://wmstutorials-9.1.aquaveo.com/48%20Gssha-Modeling-PostProcessing.pdf
[58] http://wmstutorials-9.1.aquaveo.com/BasicGSSHA.zip
[59] http://wmstutorials-9.1.aquaveo.com/DigitalDam.zip
[60] http://wmstutorials-9.1.aquaveo.com/Infiltration.zip
[61] http://wmstutorials-9.1.aquaveo.com/Visualization.zip
[62] http://wmstutorials-9.1.aquaveo.com/49%20Gssha-Applications-Precipitation.pdf
[63] http://wmstutorials-9.1.aquaveo.com/50%20Gssha-Applications-LandUseChange1.pdf
[64] http://wmstutorials-9.1.aquaveo.com/51%20Gssha-Applications-LandUseChange2.pdf
[65] http://wmstutorials-9.1.aquaveo.com/52%20Gssha-Applications-LongTerm.pdf
[66] http://wmstutorials-9.1.aquaveo.com/53%20Gssha-Applications-SedimentTransport.pdf
[67] http://wmstutorials-9.1.aquaveo.com/54%20Gssha-Applications-ConstituentTransport.pdf
[68] http://wmstutorials-9.1.aquaveo.com/55%20Gssha-Applications-OverlandBoundaryConditions.pdf
[69] http://wmstutorials-9.1.aquaveo.com/64%20Gssha-Applications-Snowmelt.pdf
[70] http://wmstutorials-9.1.aquaveo.com/Precipitation.zip
[71] http://wmstutorials-9.1.aquaveo.com/Scenarios.zip
[72] http://wmstutorials-9.1.aquaveo.com/LongTerm.zip

[73] http://wmstutorials-9.1.aquaveo.com/Sediment.zip
[74] http://wmstutorials-9.1.aquaveo.com/Contaminants.zip
[75] http://wmstutorials-9.1.aquaveo.com/BoundaryCondition.zip
[76] http://wmstutorials-9.1.aquaveo.com/56%20Gssha-Calibration-ManualCalibration.pdf
[77] http://wmstutorials-9.1.aquaveo.com/57%20Gssha-Calibration-Stochastic.pdf
[78] http://wmstutorials-9.1.aquaveo.com/58%20Gssha-Calibration-AutomatedCalibration.pdf
[79] http://wmstutorials-9.1.aquaveo.com/Calibration.zip
[80] http://wmstutorials-9.1.aquaveo.com/59%20Gssha-Groundwater-GroundwaterModeling.pdf
[81] http://wmstutorials-9.1.aquaveo.com/60%20Gssha-Groundwater-AdvancedGroundwaterModeling.pdf
[82] http://wmstutorials-9.1.aquaveo.com/61%20Gssha-Groundwater-TileDrains.pdf
[83] http://wmstutorials-9.1.aquaveo.com/Groundwater.zip
[84] http://wmstutorials-9.1.aquaveo.com/SubSurface.zip
[85] http://wmstutorials-9.1.aquaveo.com/25%20HydraulicsAndFloodplainModeling-HECRASAnalysis.pdf
[86] http://wmstutorials-9.1.aquaveo.com/26%20HydraulicsAndFloodplainModeling-HECRASManagingCrossSections.pdf
[87] http://wmstutorials-9.1.aquaveo.com/hecras.zip
[88] http://wmstutorials-9.1.aquaveo.com/xsecs.zip
[89] http://wmstutorials-9.1.aquaveo.com/27%20HydraulicsAndFloodplainModeling-FloodplainDelineation.pdf
[90] http://wmstutorials-9.1.aquaveo.com/28%20HydraulicsAndFloodplainModeling-StochasticModeling.pdf
[91] http://wmstutorials-9.1.aquaveo.com/flood.zip
[92] http://wmstutorials-9.1.aquaveo.com/stochastic.zip
[93] http://wmstutorials-9.1.aquaveo.com/29%20HydraulicsAndFloodplainModeling-SimplifiedDamBreak.pdf
[94] http://wmstutorials-9.1.aquaveo.com/smpdbk.zip
[95] http://wmstutorials-9.1.aquaveo.com/30%20HydraulicsAndFloodplainModeling-HY8.pdf
[96] http://wmstutorials-9.1.aquaveo.com/HY8WizardTutorialFiles.zip
[97] http://wmstutorials-9.1.aquaveo.com/62%20HydraulicsAndFloodplainModeling-HydraulicToolbox.pdf
[98] http://wmstutorials-9.1.aquaveo.com/HydraulicToolbox.zip
[99] http://wmstutorials-9.1.aquaveo.com/31%20StormDrainModeling-SWMMModeling.pdf
[100] http://wmstutorials-9.1.aquaveo.com/stormrat.zip
[101] http://wmstutorials-9.1.aquaveo.com/32%20StormDrainModeling-StormDrainRationalDesign.pdf
[102] http://wmstutorials-9.1.aquaveo.com/33%20StormDrainModeling-StormDrainHydrographicDesign.pdf
[103] http://wmstutorials-9.1.aquaveo.com.s3.amazonaws.com/63%20StormDrainModeling-HY12RationalDesign.pdf
[104] http://wmstutorials-9.1.aquaveo.com.s3.amazonaws.com/HY12Rational.zip
[105] http://wmstutorials-9.1.aquaveo.com/34%20WaterQualityModeling-HSPF.pdf
[106] http://wmstutorials-9.1.aquaveo.com/hspf.zip
[107] http://wmstutorials-9.1.aquaveo.com/35%20WaterQualityModeling-CEQUALW2.pdf
[108] http://wmstutorials-9.1.aquaveo.com/cequal.zip

What's new in WMS version 9.1

The WMS software development team is excited about the release of WMS 9.1! This page lists the exciting new features that have been added to WMS 9.1.

What's new in WMS 9.1

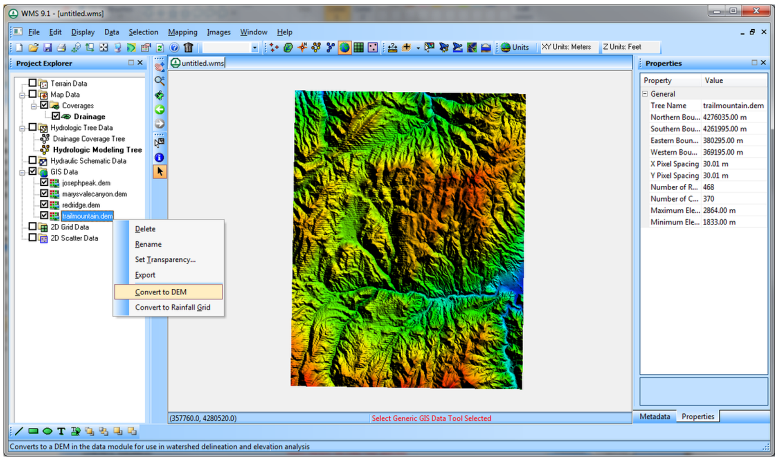
Support of new GIS and CAD File Formats for Import and Export

Add GIS Data Command

The Add GIS Data command in the Get Data toolbar allows you to read many formats of vector and raster GIS data. After reading the data, it can be viewed or converted to a format that can be used for hydrologic modeling in WMS.

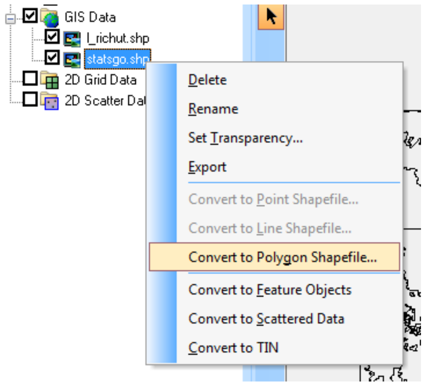
Raster Display Options

If you have read raster elevation data using the Add GIS Data command or if you have online data that contains raster elevations, there are various options for displaying hill shading on the raster data. You can also convert any raster elevation data to a DEM.



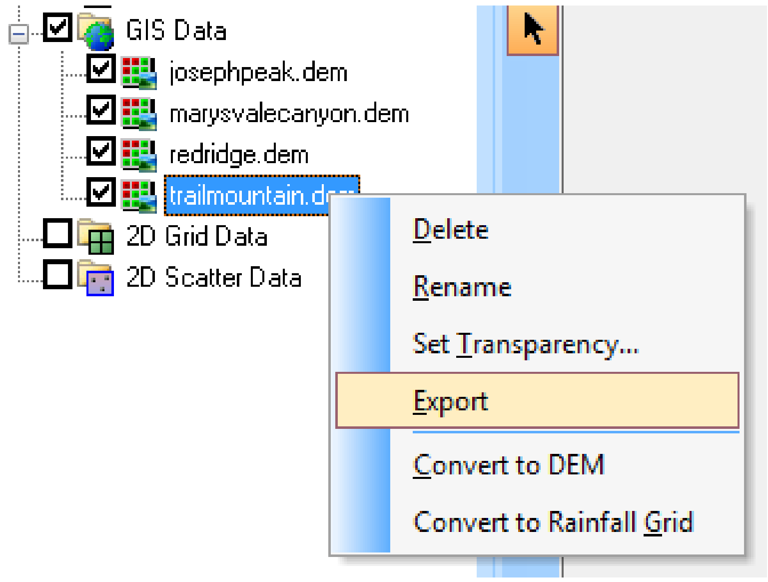
There are various options to view and convert raster elevation data grids to data that can be used in WMS for hydrologic modeling. There are also options for other types of raster data.

GIS Vector Data Conversions



GIS Vector Data can be converted to a shapefile. Once your GIS data are converted to a shapefile, they can be used in any of the hydrologic modeling computations for tasks like computing curve numbers and infiltration coefficients.

Raster Data Conversions



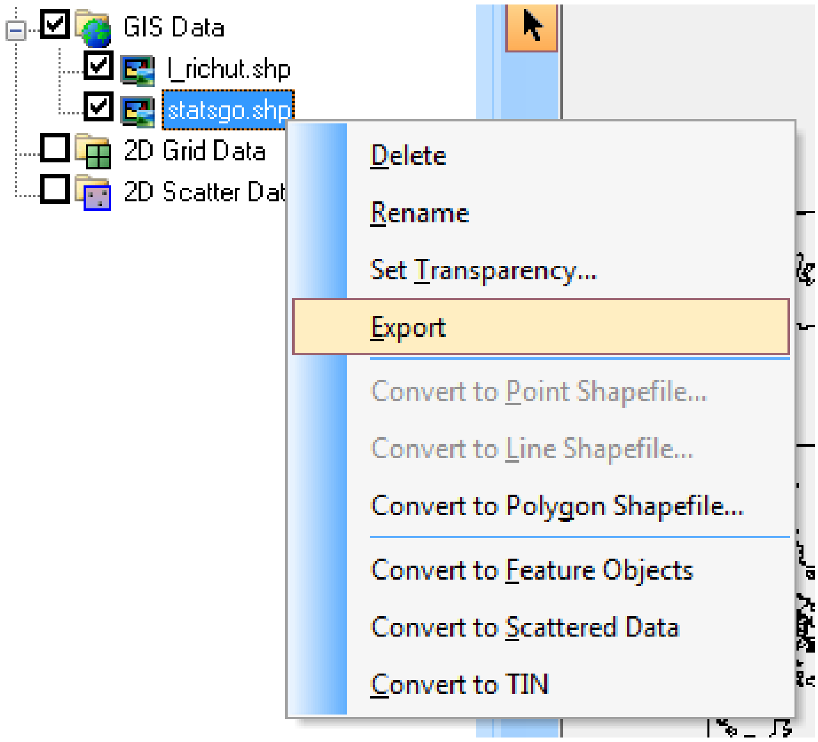
Raster data with information about land use or soil type can be converted to a land use or soil type grid by right-clicking on the layer. USGS NLCD and European CORINE Land use data can also be downloaded for anywhere in the United States and Europe using the Get Data tool. More information about the new data sources available in WMS 9.1 can be viewed here.

WMS DEMs can be exported to several digital elevation file formats, including the following:

- GeoTiff
- BIL
- ERDAS Imaging IMG
- Surfer ASCII Grid
- Surfer Binary Grid
- USGS ASCII DEM
- XYZ ASCII Grid
- ArcInfo ASCII Grid
- DXF 3D Point
- Float/Grid
- DTED
- MapInfo Grid
- GlobalMapper Grid
- Windsim GWS

All these files as well as many other formats can also be imported using the Add GIS Data button.

Vector/Shapefile Data Conversions



Vector GIS data that is read into WMS can be converted to a shapefile and used in WMS or saved to one of the many other supported formats. The supported formats include:

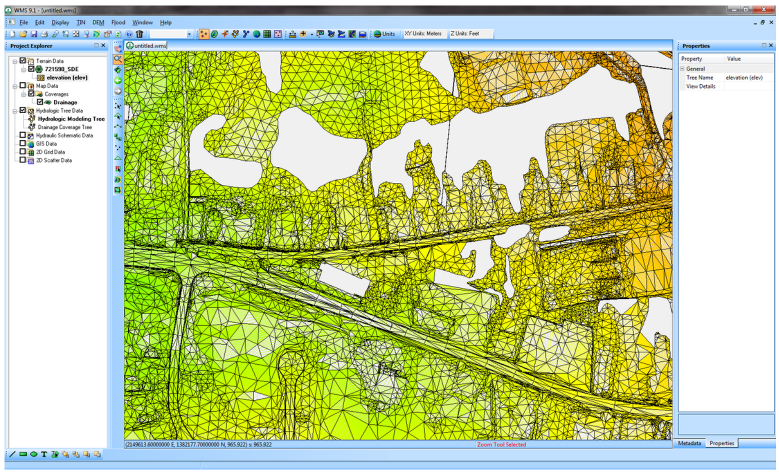
- DXF Files (*.dxf)
- Area Shapefiles (*.shp)
- Line Shapefiles (*.shp)
- Point Shapefiles (*.shp)
- Google Earth KMZ Files (*.kmz)
- MapInfo MIF/MID Files (*.mif)
- MapInfo TAB/MAP Files (*.map)
- Simple ASCII Text Files (*.txt)
- CSV (Comma-separated value) Files (*.csv)
- SVG Files (*.svg) [1]

GIS Module

All images, raster GIS data, and vector GIS data are now stored in the GIS module. In the GIS module, you can import, export, and convert GIS data to different formats. You can also map data in the GIS module to data in WMS that can be used to build your watershed models.

TIN Elevations from LandXML Files

WMS 9.1 allows you to read LandXML files as a TIN. This command to import LandXML files preserves your points and the triangle connections that were built in the CAD program that was used to generate the LandXML file.

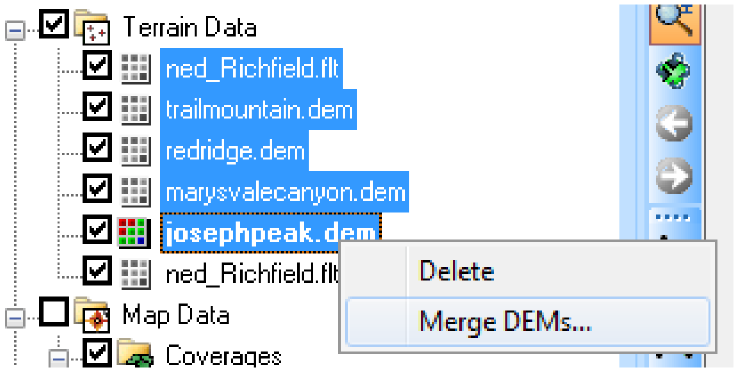


LandXML TIN imported into WMS.

DGN, DWG, and other CAD File Support

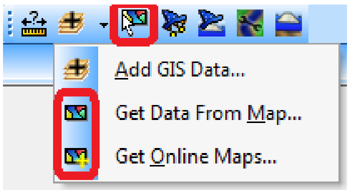
WMS 9.1 supports most of the latest DGN, DWG, and DXF file formats through the Teigha library [2]. Currently, WMS 9.1 uses version 3.05.01 of the Teigha library.

Multiple DEMs



WMS 9.1 allows you to read and manage multiple DEMs in the project explorer. Each of these DEMs can have flow directions and accumulations for watershed delineation. You can clip and modify single DEMs or merge multiple DEMs from different sources into a single DEM. You can convert raster elevation data in the GIS module to DEMs. These DEMs can be merged and exported to any of the raster elevation formats supported by WMS.

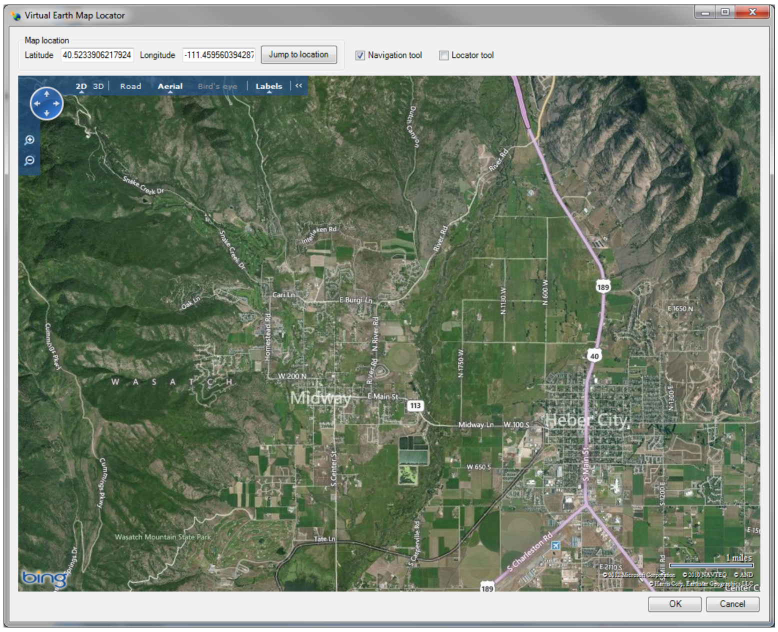
Improved Web Service (Online Data) Tools



WMS 9.1 has three tools for obtaining online data: The Get Data tool, the Get Data From Map command, and the Get Online Maps command. All of these tools can be used to obtain various types of raster data: Images, elevation data, land use data, vegetation data, and more.

Get Data

The Get Data tool and the Get Data From Map command work in much the same way. You need to set your current projection before using either of these tools. The difference is that with the Get Data tool, you select the area where data is desired from your WMS window. With the Get Data From Map command, you select the area from the Virtual Earth Map Locator window. This Virtual Earth window pops up after selecting the Get Data From Map command. In either case, you go to the area you are interested in and WMS will bring up another window that allows you to select the type of data you would like to download. With all the raster data sources, you specify a resolution of the data you want to download and WMS will download the data.

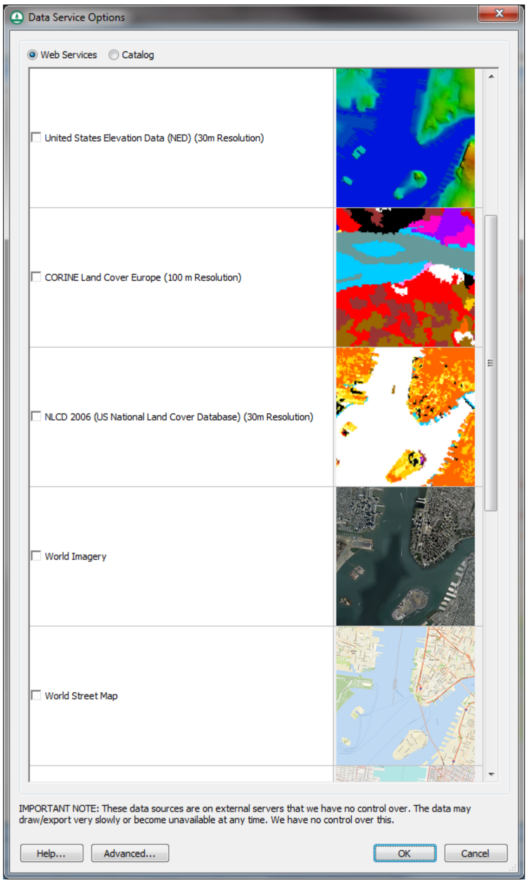


The Virtual Earth Map Locator window.

Several new data sources have been added to the Get Data tools in WMS, including various sources for high-quality imagery. Samples of many of the data sources are shown in the new Get Data dialog.

Get Online Maps

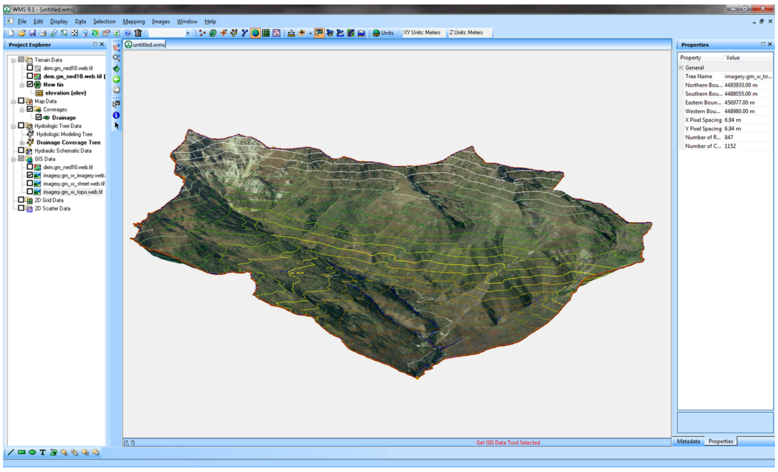
The Get Online Maps command allows you to define online data sources you want added to your display. This command was introduced in WMS 9.0, and has been improved in WMS 9.1. WMS 9.1 launches a separate process to download the online data when it is downloading so you no longer need to wait for your data to download before working. WMS 9.1 also has tools to convert your online data to static data and to convert it to various formats that can be used for watershed modeling.



The Get Data dialog.

Texture Mapping on TINs

The capability to texture map images to TINs has been added. This capability allows you to read or download an image using the Get Data tools and then texture map the image on a TIN. This gives you the option to create a nice image of your watershed model and to visualize the locations and terrain surrounding hydraulic structures in your watershed. You can also use the texture mapping capabilities to view flood locations and contours from a HEC-RAS analysis.

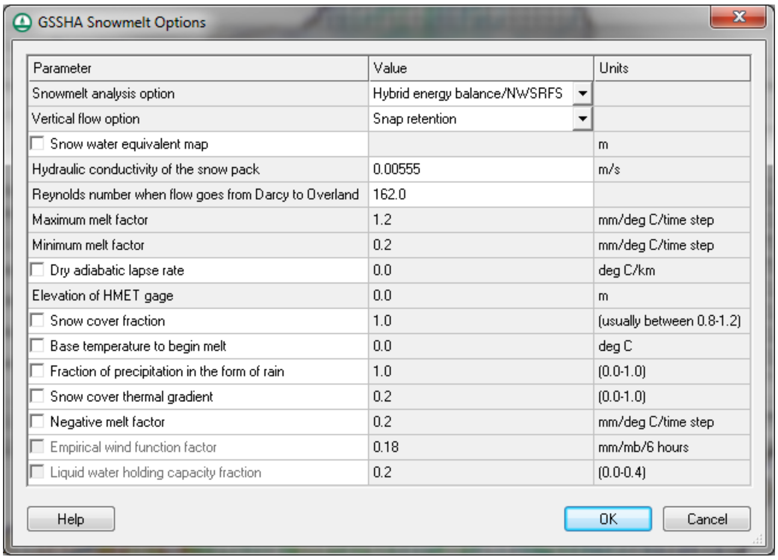


A delineated watershed with a texture mapped image.

Model Improvements

GSSHA Snowmelt

Several options to support the GSSHA snowmelt models have been added to the WMS interface. When running long-term GSSHA simulations, the GSSHA snowmelt parameters can be modified in the easy-to-use WMS interface. WMS 9.1 also has options to define raster HMET files for use in GSSHA long term simulations.



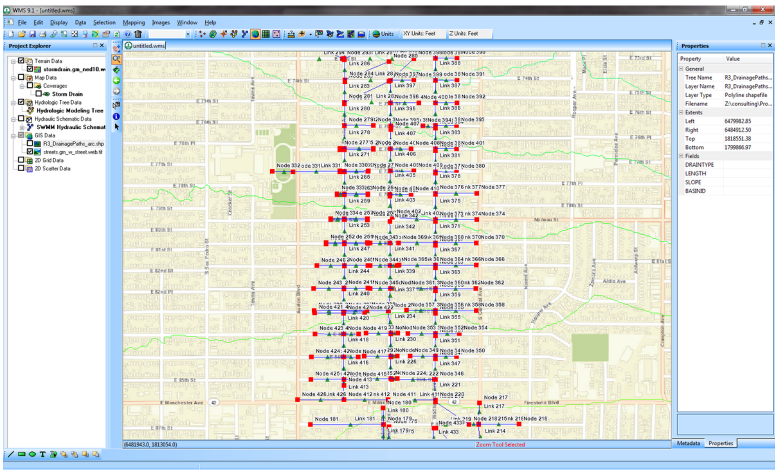
GSSHA Snowmelt Options Dialog.

HEC-RAS Water Surface Elevations

If you have an existing HEC-RAS model and want to bring it into WMS, there has not previously been a way to read the water surface elevations from the HEC-RAS solution. Now if you export your water surface elevations with the GIS file that's exported from HEC-RAS, WMS reads these water elevations.

SWMM Attributes

If the correct attributes have been assigned to arcs in a shapefile, WMS now imports storm drain attributes such as names, shapes, diameters, lengths, and upstream and downstream invert elevations to your SWMM model in WMS. This makes your SWMM modeling easier because you don't have to re-enter the pipe attributes that are already defined in a shapefile.



You can map attributes from shapefiles to SWMM models in WMS 9.1.

Bug fixes

To view the list of bugs fixed in WMS 9.1, visit the WMS bugfix page.

References

[1] http://en.wikipedia.org/wiki/Scalable_Vector_Graphics
[2] http://www.opendesign.com/the_oda_platform

FAQ

WMS Frequently Asked Questions.

Installation and Licensing Issues

Hardware Locks

Visit the Hardware Locks page for Hardware lock troubleshooting

GSSHA

The font size in the model wrapper window is too small! Can I change the font size?

No, there isn't any way to change the font in the model wrapper itself, but you can view the same information in a different file.

In order to view and edit the text:
1)Open the *.sum file that GSSHA creates into Wordpad or a similar program
2)Change the font.

Information included in the *.sum file:

GSSHA start-up information
GSSHA EVENT RUNOFF SIMULATION SUMMARY
THE FOLLOWING WARNINGS ARE GIVEN:
THE FOLLOWING PROCESSES WERE SIMULATED:
etc.

References

Publications Involving WMS

Smemoe, C.M., Nelson, E.J., Zundel, A.K., and Miller, A.W. (2007). “Demonstrating Floodplain Uncertainty Using Flood Probability Maps,” Journal of the American Water Resources Association, 43(2), 359-371.

Smemoe, C.M., Nelson, E.J., Zundel, A.K., and Miller, A.W. (2006). “Floodplain Risk Analysis Using Annual Exceedance Probability Maps,” Hydrological Science and Technology, 22(1-4), 185-196.

Smemoe, C.M., Nelson, E.J., and Zhao, B.I. (2004). “Spatial averaging of land use and soil properties to develop the physically-based green and ampt parameters for HEC-1,” Environmental Modelling and Software, 19(6), 525-535.

Smemoe, C.M., “Floodplain Risk Analysis Using Flood Probability and Annual Exceedance Probability Maps,” Ph.D. Dissertation, Brigham Young University, April 2004.

Green, Jonathan I., and E. James Nelson, “Calculation of Time of Concentration for Hydrologic Design and Analysis Using Geographic Information System Vector Objects,” Journal of Hydroinformatics, Vol. 4, No. 2, pp. 75-81, April 2002.

Jones, Norman L., Stephen G. Wright, and David R. Maidment, 1990, "Watershed delineation with triangle based terrain models," ASCE Journal of Hydraulic Engineering, Vol. 116, No. 10, Oct. 1990, pp 1232-1251.

Nelson, E. James, Norman L. Jones, and A. Woodruff Miller, "An algorithm for precise drainage basin delineation," ASCE Journal of Hydraulic Engineering, March, 1994, pp. 298-312.

Nelson, E. James, and Norman L. Jones, "Reducing Elevation Roundoff Errors in Digital Elevation Models," Journal of Hydrology, Vol. 169, pp. 37-49, 1995.

Nelson, E. James, Norman L. Jones, and Russell J. Berrett, "Adaptive Tessellation Method for Creating TINs from GIS Data," ASCE Journal of Hydrologic Engineering, Vol. 4, No. 1, 1999.

Nelson, E. James, Glenn E. Moglen, and Larry Arneson, "Building a GIS Database to Support Hydrologic Modeling at the Maryland State Highway Administration," Proceedings of the 78th Annual Meeting of the Transportation Research Board, January 10-14, 1999, published on CD ROM.

Nelson, E. James., A. Woodruff Miller, and Eric Dixon, "Mud Canyon Fire: A Hydrologic Evaluation of Rainfall Runoff from a Burned Watershed," International Journal of Wildland Fire, Vol. 9, No. 1, 2000.

Noman, Nawajish S., E. James Nelson, and Alan K. Zundel, “A Review of Automated Flood Plain Delineation from Digital Terrain Models,” ASCE Journal of Water Resources Planning and Management, Nov/Dec 2001, Vol. 127, No. 6, pp. 394-402.

Noman, Nawajish S., E. James Nelson, and Alan K. Zundel, “An Improved Process for Floodplain Delineation from Digital Terrain Models,” ASCE Journal of Water Resources Planning and Management, accepted for publication, estimated publication in September 2003.

Noman, Nawajish S., and E. James Nelson, “ArcGIS Hydro Data Model – Chapter 5 River Channels,” ESRI Press, 2002, 31 pp.

Omer, Creighton R., E. James Nelson, and Alan K. Zundel, “Impact of Varied Data Resolution on Hydraulic Modeling and Flood Plain Delineation,” Journal of the American Water Resources Association, Vol. 39, No. 2, April 2003.

Model Documentation Publications

Model documentation files are included as .pdf files in the "docs" subdirectory where WMS is installed.

Secondary Publications (methodologies used in WMS)

Band, L.E., 1986, "Topographic partition of watersheds with digital elevation models", Water Resources Research, Vol. 22, No. 1, pp. 15-24.

Clough, R.W., and J.L. Tocher, 1965, "Finite element stiffness matrices for analysis of plates in bending," Proc. Conf. Matrix Methods in Structural Mechanics, Wright-Patterson A.F.B., pp. 515-545.

Franke, R., 1982, "Scattered data interpolation: tests of some methods," Mathematics of Computation, Vol. 38, Nol 157, pp. 181-200.

Garbrecht, J., and L.M. Martz, 1993, "Case application of the automated extraction of drainage network and subwatershed characteristics from digital elevation models by DEDNM," AWRA Proceedings of the Geographic Information Systems and Water Resources, March 1993, pp. 221-229.

Grayman, W. M., R. M. Males, and J. J. Harris, 1982, "Use of integrated spatial data and modeling capabilities for urban runoff analyses," International Symposium on Urban Hydrology, Hydraulics and Sediment Control, University of Kentucky, Lexington, Kentucky, July 27-29.

Jenson, S.K. and J.O.Domingue, 1988, "Extracting topographic structure from digital elevation data for geographic system analysis", Photogrammetric Engineering and Remote Sensing, Vol. 54, No. 11, pp 1593-1600.

Lawson, C. L., 1977, "Software for C1 surface interpolation," Mathematical Software III, J. R. Rice, ed., Academic Press, New York, pp.161-194.

Lee, D. T., and B. J. Schacter, 1980, "Two algorithms for constructing a Delauney triangulation," International Journal of Computer and Information Sciences, Vol. 9, No. 3 pp. 219-242.

Maidment, D. R., ed., 2002, "Arc Hydro: GIS for Water Resources," ESRI Press, Redlands, CA.

Maidment, D. R., ed., 1993, "Handbook of Hydrology," McGraw-Hill.

Martz, L.M., and J. Garbrecht, 1993, "DEDNM: A software system for the automated extraction of channel network and watershed data from raster digital elevation models," AWRA Proceedings of the Geographic Information Systems and Water Resources, March 1993, pp. 211-220.

Peucker, T.K., and D.H. Douglas, 1975, "Detection of surface-specific points by local parallel processing of discrete terrain elevation data," Computer Graphkcs and Image Processing, Vol. 4, pp. 375-387.

Shepard, D., 1968, "A two dimensional interpolation function for irregularly spaced data," Proceedings of the 23rd National Conference ACM, pp. 517-523.

Smith, R. E. and Goodrich, D. C. (2000). “Model for rainfall excess patterns on randomly heterogeneous areas.” Journal of Hydrologic Engineering, 5(4), 355-362.

Southard, D.A., 1990, "Piecewise planar surface models from sampled data", Scientific Visualization of Physical Phenomena, Springer-Verlag, New York, NY, pp. 667-680.

Watson, D.F., 1981, "Computing the n-dimensional Delauney tesselation with application to Voronoi polytopes," The Computer Journal, Vol. 8, No. 2, pp. 167-172.

Readme

GETTING STARTED

INSTALLATION

Installing from a CD:

When the CD is inserted, a screen should pop-up automatically. Click on the WMS icon to begin the setup program that will guide you through the WMS installation. If the pop-up screen does not appear when you insert the CD, run the program "setup.exe" in the \Wms\Pc directory of the CD.

Installing from the web:

WMS and its supporting files (tutorial files, models) can be downloaded from the following URL:

http://www.aquaveo.com/downloads

REGISTERING WMS

When you first install WMS, you will need to register it. You can choose to register with a password, a hardware lock, or with an evaluation password, which lets you evaluate the program for free for 30-60 days.

To Obtain A Password:

Select Enable when WMS starts up. This brings up the Register dialog, which steps you through the registration process. If not using a hardware lock, a password will be sent to your email to allow you to register the product.

Hardware Lock:

Follow the instructions you received with the hardware lock to install the hardware lock and accompanying drivers. If you did not receive hardware lock instructions, or they have been misplaced, they can be found in the \Utils\Hwlock\Instructions directory on the CD. There are separate files for single user and network hardware locks. These files can be read using your web browser. If you would like to purchase or have questions about hardware locks, please call us at: 801-691-5530.

Documentation

The complete WMS Reference Manual is included on the CD you received with your purchase. The WMS Tutorials are also included, as well as the available manuals for the models for which WMS has interfaces. WMS documentation in .pdf format can be found in the WMS Docs directory and model documentation can be found in the Model Docs directory. If you do not have the Adode Acrobat Reader you can install for free from the Adobe website. If you would like hardcopies of any documentation related to WMS, please contact us here at Aquaveo. Pricing for our documentation can be found on our website.

Subscribing to the WMS Mailing List

The WMS mailing list keeps you informed of the latest product news, webinars, training courses, announcements, and special offers. You can subscribe to the mailing list by sending an email to "emailupdate@aquaveo.com" with the word "SUBSCRIBE wms" in the body of the message.

Subscribing to the User Forum

The WMS user forum allows users to post questions and view responses from other WMS users worldwide. The forum is also monitored by Aquaveo staff, including WMS developers. Registering and subscribing to the user forum is the best way to be notified of software updates and bug fixes. Forum registration is free. Subscriptions are controlled by the user on a per-thread basis. You can register on the user forum at:

http://forum.aquaveo.com/

Technical Support

Technical support is available for all commercial WMS users from the Aquaveo technical support staff. Our staff is available from 8:00 AM to 5:00 PM (Mountain Time). Contact technical support at:

- Email: support@aquaveo.com
- Phone: 801-691-5530
- Web: http://www.aquaveo.com/technical-support

All bug reports should specify:

- How to reproduce the problem
- The version and build date (found by selecting Help|About from the WMS menu) of WMS used
- System configuration (OS, CPU, RAM, disk space, network)
- Attach your files if possible

Non-commercial Government Users

Certain government agencies have participated in the devlopment of WMS, and are entitled to free licenses of the software. Those eligble are any offices of the Department of Defense (DoD), Department of Energy (DoE), Environmental Protection Agency (EPA), and Federal or State Highway Departments. Certain on-site contractors for these entities also are eligible.

For more information on obtaining a government license, go here [1].

References

[1] http://www.aquaveo.com/government-support

1.1 Set Up

Setting Up WMS

Installing

On the PC, the installation wizard will guide you through the installation process. You will have the option to install different parts of the WMS program including the executable files, tutorial files, documentation files, etc. If you are missing a part of the installation then reinstall and verify that all parts are included in the installation.

WMS System Requirements

Registering WMS

Find out how to register WMS using a hardware lock or password.

Defaults File (ini)

When you start WMS there are default values set for directories, display options, etc. You can modify these settings and save your own defaults in an .ini file by choosing the Save Current Settings command from the File menu. Each time you start WMS thereafter, these settings will be used.

Each project file stores its own default settings in .ini file so that the project data will appear as they were at the time they were saved.

Command Line Arguments

Find out about launching the WMS application and the available command line arguments that can be used to customize how WMS starts up.

Related Topics

- Registering WMS
- Running WMS
- WMS System Requirements
- Running WMS for the first time
- WMS command line arguments

License Agreement

AQUAVEO SOFTWARE LICENSE AGREEMENT

We understand that neither Aquaveo LLC nor its employees makes any warranty express or implied, or assumes any legal responsibility for the accuracy, completeness, or usefulness of the computer programs and documents herein ordered. We acknowledge that Aquaveo's liability for damages associated with these programs is limited to the amounts paid to Aquaveo. We understand that we may register the executable on only one machine per license purchased. We also agree to not make the program available to more persons, at any given time, than the number of licenses purchased. We agree to not distribute this program in unmodified or modified form, outside our organization without the written permission of Aquaveo.

License agreements for software used by Aquaveo software

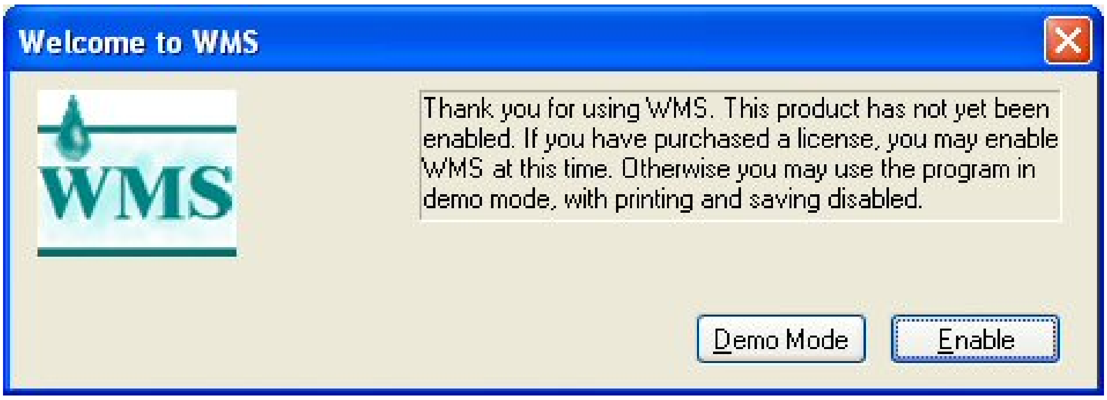
FTGL LICENSE AGREEMENT

http://ftgl.wiki.sourceforge.net/states: "FTGL is free software. You may use it, modify it and redistribute it under the terms of the MIT license or the GNU LGPL, at your option."

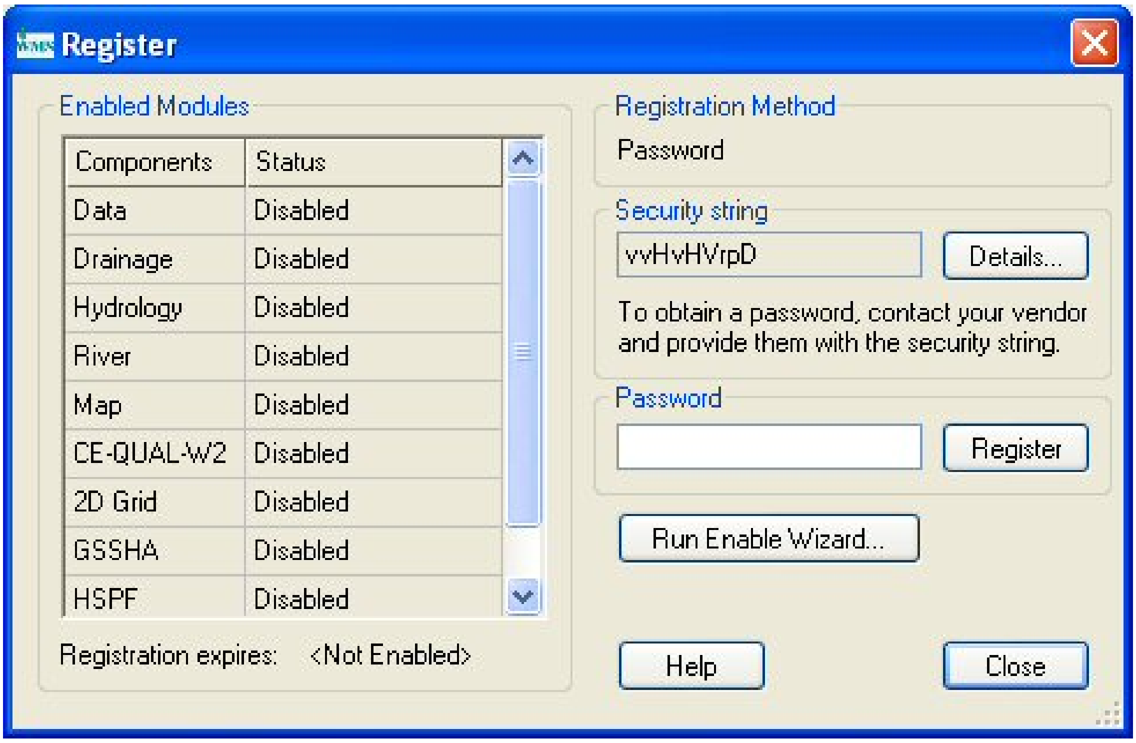
The MIT license link is to http://en.wikipedia.org/wiki/MIT_License, which states:

Registering WMS

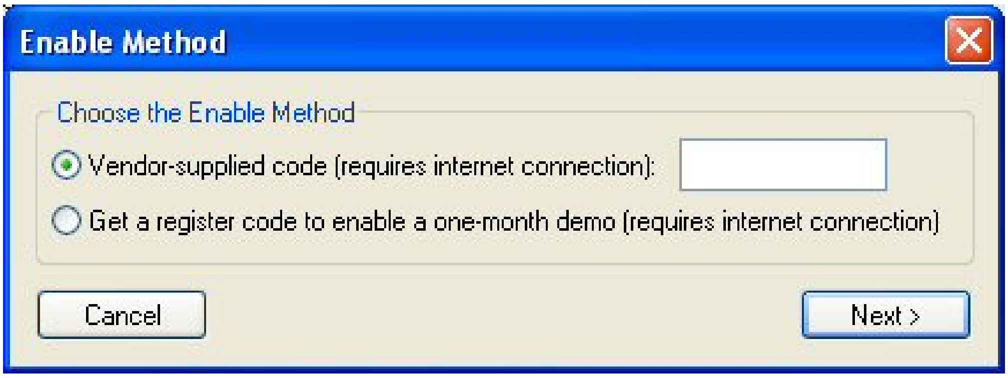
After installing WMS, it will need to be registered. You can register online whether or not you have a hardware lock. Either way, you need to obtain a string of characters called a *registration code*. Whether you're evaluating WMS, have purchased a license without a hardware lock, or have a hardware lock, you need a **registration code** to enable your version of WMS. If you are evaluating WMS for the first time, you can obtain a free 30-day registration code by registering and providing your contact information to Aquaveo. The purpose of this page is to explain how to obtain a registration code and enable WMS. When WMS is first launched, the following window appears:



Selecting the **Demo Mode** button will run WMS in Demo Mode. In demo mode, you can only read and view existing WMS files and files WMS supports. You will not be able to save, print, or output any changes made to WMS projects in any way. We recommend that you register WMS by choosing the **Enable** button. After pressing the enable button, the following window will appear:



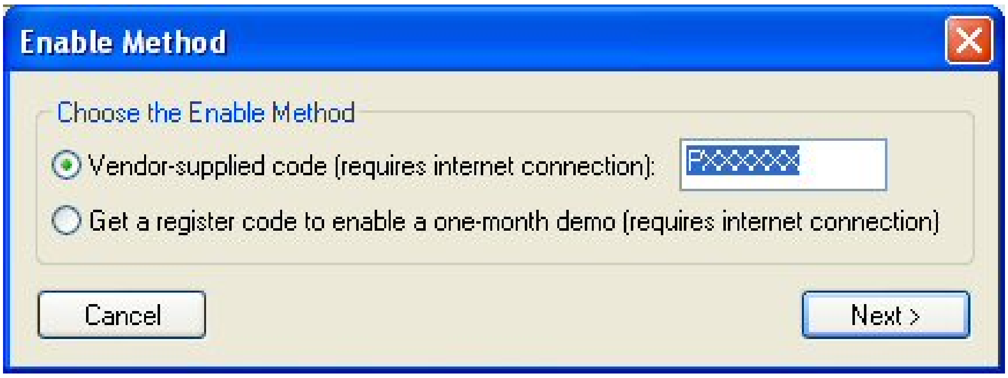
If you have just installed a new version of WMS, it is likely that your copy of WMS will not be enabled. You need to enable your copy of WMS. To enable WMS, press the **Run Enable Wizard...** button in the *Register* window. The enable wizard will appear:



In this window, you have two options. You must have an active internet connection to register your software. If you already have a vendor-supplied code (from a paid license or a temporary demo license), enter the code in the field and select *Next* to register your software (if you have a hardware lock, make sure your hardware lock drivers are installed and that your hardware lock is connected to your computer before entering your code and selecting *Next*). If you do not have a registration code, have not purchased WMS, and have not evaluated WMS before, you can obtain a registration code for a one-month demo of the software. This will give you full access to all the capabilities of WMS for one month. Select the *Get a register code...* option, select the *Next* button, and fill in your contact information on the web page that appears in your web browser. A registration code will be sent to your e-mail, and you can return to this *Enable Wizard* to enter this code after you receive it. After your evaluation period expires, you will need to purchase a license to continue using WMS.

Paid Licenses

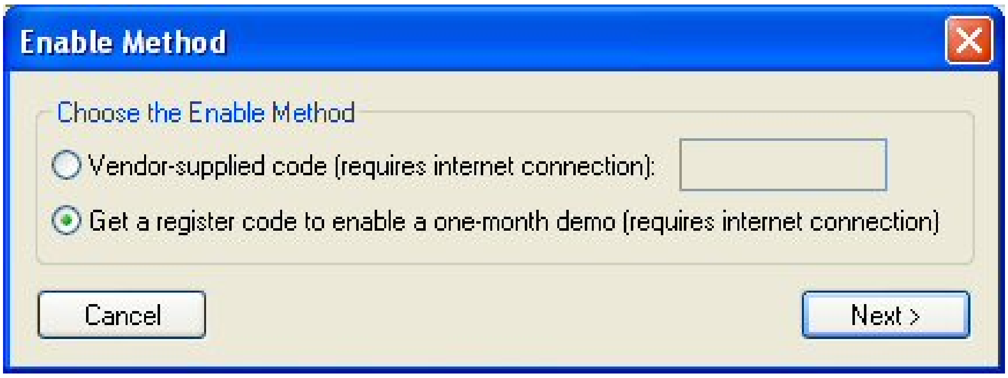
If you have purchased a license of WMS, your WMS vendor should have provided a registration code, normally sent to your e-mail address. Please enter this code in the *Enable Wizard*.



WMS will be automatically registered on-line (You must be connected to the Internet to register WMS. If you are not connected, please contact your vendor for instructions to register WMS off-line).

Evaluation Version

If you do not have a registration code, have not purchased WMS, and have not evaluated WMS before, you can obtain a registration code for a one-month demo of the software. An evaluation version that is valid for 30-60 days may be requested by selecting the **Get a register code...** button in the *Enable Wizard*.



You will be connected to a web browser. Fill out your contact information in the web browser and a valid registration code will be sent to you via email. Once you receive the registration code, enter it into the *Enable Wizard*, and select **Next**. Once you have evaluated WMS, please contact Aquaveo [3] with any questions or to purchase WMS.

Register WMS with a Password

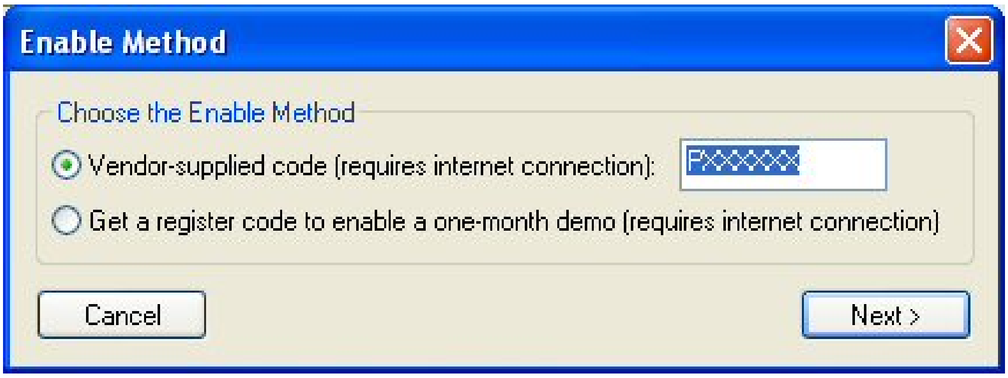
1. Start WMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in WMS.
2. Select **License code** for the *Licensing method* and enter the 7 digit alpha-numeric code that begins with the letter P. Click the **Next >** button.
3. If the registration is successful, click **Finish** to exit the *Registration Wizard*.
4. The *Register WMS* dialog displays the registered components, licensing method, and license expiration dates.

Hardware Locks

Follow the instructions you received with the hardware lock to install the hardware lock and accompanying drivers. If you did not receive hardware lock instructions, or they have been misplaced, they can be found in the \Utils\Hwlock\Instructions directory on the CD. There are separate files for single user and network hardware locks. These files can be read using your web browser.

Additional instructions on using hardware locks can be found here. We recommend going over these instructions and troubleshooting suggestions before calling technical support with hardware lock issues. In fact, reading through and following these hardware lock instructions before using your hardware lock with your software will significantly reduce the time and trouble required to get your hardware lock working.

After you have installed the hardware lock drivers on your computer and have connected your hardware lock to your computer, you can burn the lock by entering your registration code in the *Register Wizard*.



Your WMS vendor should have provided a registration code, normally sent to your e-mail address. Make sure your hardware lock is plugged into your computer, then enter this registration code in the Enable Wizard. Make sure you are connected to the internet, and your hardware lock will be automatically burned over the internet after hitting the **Next** button in the wizard.

Public Domain Version

WMS has a free public domain version. The free version provides a Widows interface to build the files for HEC-1, TR-20, and the other basic hydrologic models. If you would like the public domain version, contact a WMS vendor and a valid registration code will be sent to you via email. Once you receive the registration code, enter it into the Enable Wizard and select register, the same as for paid licenses.

Related Topics

- Running WMS
- Setting up WMS

Graphics Card Troubleshooting

XMS (WMS, GMS, or SMS) use OpenGL for rendering graphics. OpenGL is a graphics standard, but each implementation is maintained by individual graphics card companies. Different graphics cards and drivers support different versions of the OpenGL standard. XMS currently uses features up to version 1.5 of OpenGL (as of April 2009 version 3.1 was most recent version).

Some graphics cards, as well as remote desktop, do not support functionality through OpenGL version 1.5. This is mostly a problem with older integrated graphics cards, in particular those manufactured by Intel. This page will give you some ideas on troubleshooting these problems. The best solution is to get a graphics card that supports later versions of OpenGL. You will see improved performance as well as be able to access all the features of XMS.

Remote Desktop

XMS (WMS, GMS, or SMS) will have reduced capability when running remote desktop.

Since remote desktop only supports OpenGL version 1.1 not all of the features of XMS may be available.

1. One solution is to use a different remote control software that utilizes the graphics card of the computer you are controlling. www.logmein.com [1] has free and paid versions of remote desktop that behave better with XMS. RealVNC is a program that does this and can be purchased at a reasonable cost. There is a free version but it has not been tested with the XMS software. See VNC Homepage [2] for more information.
2. Another solution is to use the Mesa software rendering option available in the application's graphic preferences. See the section below on OpenGL Graphics Dialogs for discussion of this option.

Parallels Desktop for Mac

XMS has reduced capability when running in a pure virtual PC through Parallels Desktop for Mac. Although Parallels version 6.0 provides OpenGL version 2.1 support (instead of OpenGL version 1.1) when "Enable 3D acceleration" is selected in the virtual machine's hardware configuration, the Parallels virtual video card adapter does not render all XMS graphics correctly. The solution is to use the Mesa software rendering option available in XMS's graphic preferences. See the section below on OpenGL Graphics Dialogs for discussion of this option.

If you are running XMS in a virtual PC utilizing a Boot Camp partition then Parallels uses the actual graphics card installed in the Mac. See sections below regarding graphics card issues.

OpenGL Graphics Dialogs

XMS (post WMS 8.2, GMS 7.0 onward, and SMS 10.1 onward) have dialogs that allow the selection of OpenGL support. The choice is between the system default library and the Mesa software library. The system default can change based upon current conditions such as a remote login. Not all system defaults support all needed graphics functionality. Therefore Mesa is provided for better functionality at a potential reduction in speed. However, Mesa may produce poor images when printing. The user can make this tradeoff in the graphics dialog found in preferences. The dialog provides 4 options so that on subsequent runs XMS will:

1. Ask which graphics library to use if the system does not support all OpenGL functionality needed by XMS. This option is initially set and gives the following options:
   1. Autoselect the Mesa software library for this run if the system default does not support all functionality. XMS will not prompt on subsequent runs. It will just check support and select a library.
   2. Use the system default library on this run (and on future runs if the "Do not ask again box" is checked).
   3. Use the Mesa software library on this run (and on future runs if the "Do not ask again box" is checked).
2. Autoselect the Mesa software library if the system default does not support all functionality.

3. Always use the system default library.
4. Always use the Mesa software library.

Determining Graphics Card Manufacturer

Always download and install the latest drivers from your graphics card vendor. Graphics card problems are often due to using the wrong or outdated drivers. You can use a simple diagnostic program called dxdiag [3] to determine your computer's hardware, operating system, and graphics card. To use the dxdiag [3] program:

1. Select "Start"
2. Choose "Run."
3. Type "dxdiag" in the box and click "OK."
4. Click "Yes" to the prompt, and the program will begin running.
5. Select the "Display" tab and the Name listed under the "Device" section is the name of your graphics card.

You can also:

1. Right-click on the desktop and select "Properties"
2. In the Display Properties dialog, click on the "Settings" tab
3. Your video card manufacturer and chipset is shown below the "Display:" line
4. Look for the names NVIDIA, ATI, Intel, Matrox, SiS, S3, etc.

Updating Laptop Graphics Card Drivers

If you have a laptop, visit the laptop manufacturer's website (Dell [4], HP or Compaq [5], Toshiba [6], Sony [7], etc.) to get the most recent driver.

Updating Desktop Graphics Card Drivers

If you are using a desktop computer, visit the graphics card manufacturer's website to download the latest driver. Listed below are a few common graphics cards and links to download their drivers:

- 3DLabs [8]
- ATI [9]
- Diamond [10]
- Elsa [11]
- Intel [12]
- Matrox [13]
- nVidia [14]
- S3 [15] – Not all S3 card support OpenGL 1.5 which is required for all display options to be enabled.
- SIS [16] – Not all SIS card support OpenGL 1.5 which is required for all display options to be enabled.
- VIA [17] – Not all VIA card support OpenGL 1.5 which is required for all display options to be enabled.

Updating Windows Operating System

Many problems are resolved by keeping the windows operating system and hardware drivers up to date using the windows update site [18]. Hardware updates are often only installed if the "Custom" or "Optional" updates are included.

Updating XMS Software

Many problems are resolved by installing the latest version of XMS. Bugfixes and updates are released frequently. The updates can be downloaded at the Aquaveo Download Center [19].

Known Graphics Issues

- Issue: Graphic symbols are not displayed correctly and sometimes corrupt text lines located next to them.

  Hardware: Make: ATI Technologies Inc. Model: RADEON X600 PRO (0x5B62) Name: ATI Radeon X300/X550/X1050 Series

  Solution: Updating the driver will allow the symbols to display correctly, but the text corruption still remains.

Switch from Hardware to Software Rendering

THE FOLLOWING SHOULD BE ATTEMPTED ONLY IF THE OTHER SOLUTIONS PRESENTED DO NOT RESOLVE THE DISPLAY ISSUES

If you have updated your graphics driver and are still having problems, you can download this opengl32.dll ZIP file [20] and unzip the "OpenGL32.dll" and the "Glu32.dll" file to the directory where XMS is installed. Close and re-open XMS so this DLL is used for displaying XMS objects. Placing these DLL's in your XMS directory will fix most graphics-related issues, such as problems with displaying triangles on large TIN or DTM datasets and other problems with displaying large amounts of data. The following are known disadvantages to using this DLL for displaying:

- Displaying graphics using this DLL will likely be slower since software is used to display your graphics instead of your computer's graphics hardware. Panning, zooming, and rotating operations will be significantly slower.
- Some entities, such as symbols, are currently not displayed correctly when using this DLL. Only squares and circles will be displayed. Changing all symbol display options to squares or symbols will allow you to work around this problem. We are currently working on trying to fix this problem of symbols not displaying when using this DLL. (THIS PROBLEM HAS NOW BEEN FIXED IN SOME BETA VERSIONS OF XMS COMPILED AFTER March 31, 2009) In general, you will not want to use this DLL unless you are working with large datasets that have display issues where XMS closes unexpectedly.

Contacting Support

If you continue to experience problems after updating your graphics card drivers, contact support [21].

References

[1] http://www.logmein.com/
[2] http://www.realvnc.com/
[3] http://en.wikipedia.org/wiki/DxDiag
[4] http://www.dell.com/
[5] http://welcome.hp.com/country/us/en/support.html
[6] http://www.toshiba.com
[7] http://www.sony.com
[8] http://www.3dlabs.com/support/drivers/
[9] http://ati.amd.com/support/driver.html
[10] http://www.diamondmm.com/
[11] http://www.elsa.com/supports/download.asp
[12] http://support.intel.com/support/graphics
[13] http://www.matrox.com/mga/support/drivers/latest/home.cfm
[14] http://www.nvidia.com/content/drivers/drivers.asp
[15] http://www.s3graphics.com/drivers.jsp
[16] http://www.sis.com/support/support_prodid.htm
[17] http://www.viaarena.com/default.aspx?PageID=2
[18] http://update.microsoft.com
[19] http://www.aquaveo.com/downloads
[20] http://wms.aquaveo.com/OPENGL32.zip
[21] http://www.aquaveo.com/technical-support/

Running WMS for the first time

When running WMS the first time (and each time until you register), the registration wizard is the first dialog you will see. If you have licensed WMS and have a hardware lock or need to get the security string so that you can obtain a password, you should choose the Enable button.

Related Topics

- Registering WMS
- Command Line

Command Line

Several command line arguments can be used with WMS when it is launched. You can modify the properties of the shortcut that launches WMS and edit the Target item. More than one command line argument can be used at the same time.

The following command line arguments are available for WMS:

-dm <module>

The -dm command is used to specify the default module. Possible values include the following strings: tin, dem, map, tree, 2dgrid, 2dscat.

Example: C:\Program Files\WMS81\wms81.exe -dm tin

-ini <path>

The -ini command is used to specify the path to the initialization file (wms61.ini) which stores the default settings.

Example: C:\Program Files\WMS81\wms81.exe -ini C:\MyStuff

-tmp <path>

The -tmp command is used to specify the path to the temporary directory. The privileges on this directory must be such that WMS can write to it.

Example: C:\Program Files\WMS81\wms81.exe -tmp C:\Temp

-f <file file file...>

The -f command is used to specify a file or files for WMS to open at startup.

Example: C:\Program Files\WMS81\wms81.exe -f models/flood.wpr

-av <file>

The -av command is used to specify an ArcView-WMS super file for WMS to open at startup.

Example: C:\Program Files\WMS81\wms81.exe -av models/gisdata.sup

-about

The -about command is used to specify copyright and vendor information for WMS.

Example: C:\Program Files\WMS81\wms81.exe -about

Related Topics

- Setting up WMS
- Running WMS

2. General Information

Bugfixes WMS

Important Note

If you download and install the latest update and your software maintenance has expired, you will not be able to run WMS. Please make sure your WMS maintenance license has not expired before downloading. You can determine your maintenance expiration date by selecting the Help | Register command from WMS. You can renew your maintenance by visiting Aquaveo's web page [1] or by contacting a sales representative. Prices and other information about renewing your license are available on the WMS Pricing Page [2]. The latest update to WMS can be downloaded here [3].

Release 9.1/Intermediate Release 9.1.4 – February 22, 2013

WMS 9.1.4 has been posted on Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/downloads [3]. The following known bugs have been fixed in this version:

Bug ID Date Submitted–Date Updated—Summary

- 5867 1/30/2013–2/22/2013—Crash when running the OC Rational method
- 5866 1/30/2013–2/22/2013—HEC-HMS files exported from WMS freeze up after selecting sub basins
- 4117 9/12/2011–2/21/2013—Error when loading GSSHA solution in Contaminants tutorial
- 5945 2/19/2013–2/19/2013—The **Edit...** button in the Culvert and Site Data section in the HY8 Modeling Wizard is missing
- 5944 2/19/2013–2/19/2013—WMS crashes when navigating through the HY8 modeling wizard
- 5846 1/22/2013–2/19/2013—Print out error information when running the OC rational method
- 5868 1/30/2013–2/19/2013—The Crest Elevation is not saved in the *Crossing Data* dialog
- 5874 1/31/2013–2/19/2013—Save curve number and other report files to Temp directory instead of current directory
- 5888 2/1/2013–2/19/2013—WMS 9.1 reports a missing resource when renumbering a MODRAT tree
- 5941 2/18/2013–2/19/2013—Incremental Data Checkbox does not show up in Figure 4-3 of GSSHA Nexrad tutorial
- 5856 1/24/2013–2/15/2013—Add capability to compute layer widths less than 5 meters in the CE-QUAL-W2 Layer Editor
- 5901 2/6/2013–2/14/2013—Add the .sdf extension to the list of filters that can be read when importing an HEC-RAS GIS file
- 5918 2/7/2013–2/13/2013—WMS crashes when trying to acces HY-8
- 5930 2/13/2013–2/13/2013—Issue with Drainage Coverage **Tree | Find** tool
- 5903 2/6/2013–2/13/2013—Problem reading floodplain boundary into WMS
- 5904 2/6/2013–2/12/2013—Problem extracting cross sections if you have a cross section point that coincides with a TIN vertex
- 5923 2/11/2013–2/11/2013—Problem with extra pipes associated with a superlink arc when building a GSSHA storm/tile drain model

- 5905 2/6/2013–2/8/2013—Check adding storm/tile drain pipes without connecting them directly to a channel at the downstream end
- 5897 2/5/2013–2/8/2013—WMS crashes when assigning reach routing for OC Rational model
- 4477 12/12/2011–2/7/2013—Online Map issues
- 4437 12/6/2011–2/7/2013—Add options to download online vector data through Global Mapper
- 4435 12/6/2011–2/7/2013—Fix problem with online image taking a long time to load after zooming or panning
- 4410 11/30/2011–2/7/2013—Outlet issues
- 4337 11/9/2011–2/7/2013—More than one plot is selected to be generetaded when using the channel calculator
- 5909 2/6/2013–2/6/2013—Crash when reading LandXML file
- 4312 11/2/2011–2/5/2013—Right clicking to access *Display Options* should select the correct module first
- 4281 10/25/2011–2/5/2013—Crash in GSSHA Land use change #2 tutorial
- 4246 10/17/2011–2/5/2013—Problems in the HY8 modeling wizard that should be addressed (from Jim)
- 4207 10/6/2011–2/4/2013—Bug with approximate total number of points in *Import DEM* dialog
- 4202 10/5/2011–2/4/2013—Tin Triangles not Displaying when option to display triangles is checked
- 4200 10/5/2011–2/4/2013—Crop Collar not working
- 4181 9/29/2011–2/4/2013—Recompute All Stations should also delete the river hydraulic schematic
- 4138 9/15/2011–2/1/2013—WMS using Outlet IDs instead of Outlet Names in *SWMM* dialog
- 5749 12/7/2012–2/1/2013—0005734: Share the app name and registry path stuff
- 5745 12/6/2012–2/1/2013—Problem computing water surface elevations using the channel calculator
- 4077 9/2/2011–1/31/2013—Cell elevation text does not match setting
- 4021 8/22/2011–1/31/2013—Contour values do not exactly match DEM values
- 4011 8/19/2011–1/31/2013—Problems with node boundary contitions in GSSHA
- 3984 8/15/2011–1/31/2013—DEM Display Issues
- 3972 8/12/2011–1/31/2013—Memory issue continues after project is closed
- 3969 8/12/2011–1/31/2013—*GSSHA* Menu not displayed
- 3968 8/12/2011–1/31/2013—Not all DEM points display
- 3967 8/12/2011–1/31/2013—Contours displaying wrong
- 3962 8/11/2011–1/31/2013—Pipe Flow scatter data not working properly
- 3931 8/9/2011–1/31/2013—DXF/DWG files not exporting
- 3890 8/2/2011–1/31/2013—Linked outlets and nodes not saved in WMS
- 3888 8/2/2011–1/31/2013—WMS Duplicates SWMM hydraulic schematic
- 3887 8/2/2011–1/31/2013—Cannot delete Hydraulic Schematic data
- 3869 7/28/2011–1/31/2013—Adding an embankment to a DEM does not seem to impact flow directions
- 3833 7/20/2011–1/31/2013—Cannot enter reach and reservoir routing at same node in HEC-HMS
- 3822 7/19/2011–1/31/2013—Problem displaying stream height cylinders in oblique view (WMS)
- 3817 7/15/2011–1/31/2013—Redistributed stream display problem
- 3816 7/15/2011–1/31/2013—Big letters displayed on button instead of symbol
- 3815 7/15/2011–1/31/2013—Link numbers do not stay aligned with links
- 3814 7/15/2011–1/31/2013—Vertices appear as nodes
- 3806 7/14/2011–1/31/2013—2D scatter set will not duplicate
- 3801 7/13/2011–1/30/2013—HY12 Tutorial Issues
- 3673 6/15/2011–1/30/2013—WMS writing wrong rainfall card for EPA SWMM tutorial
- 3605 6/1/2011–1/30/2013—gssha crashes before it finishes running when using the green and ampt infiltration option
- 3600 6/1/2011–1/30/2013—Shared tests failing in WMS but passing in SMS.
- 5834 1/17/2013–1/30/2013—Flows not being computed for MODRAT model
- 3538 5/18/2011–1/29/2013—Limiting Number of Cross sections in Simplified Dam Break Model

- 3493 5/9/2011–1/29/2013—Cannot register MPICH2 without a password
- 3228 3/10/2011–1/29/2013—Compile error when running the shared build
- 3100 2/10/2011–1/29/2013—contours not updating when making large change in specified interval
- 2905 12/22/2010–1/29/2013—Multiple Tc arcs generated
- 2615 9/29/2010–1/29/2013—MODRAT burned simulation not incrementing soil number
- 5782 12/26/2012–1/29/2013—Add Custom1 copy protection option to WMS 9.1 and Dev
- 2594 9/21/2010–1/25/2013—Digital dams, 2D grid flow vectors, and elevations not updating when you adjust 2D grid elevations
- 2578 9/14/2010–1/25/2013—WMS crashes when converting DEM to TIN after editing TOPAZ generated streams
- 2577 9/14/2010–1/25/2013—WMS crashes when reversing directions of a stream network
- 2566 9/9/2010–1/25/2013—The *Cross Section Attributes* dialog gives bogus help strings
- 2558 9/9/2010–1/25/2013—CAD file (.dwg) saved out from TIN tutorial does not open in AutoCAD 2008
- 2555 9/9/2010–1/25/2013—Feature Request for georeferenced PDFs
- 2553 9/8/2010–1/25/2013—WMS crashes when inserting breaklines
- 2547 9/7/2010–1/25/2013—WMS displays an image in previous view, even when previous view is oblique
- 2426 7/21/2010–1/25/2013—Printing in WMS 8.3 not as clear as 8.0
- 2259 5/26/2010–1/25/2013—Error Selecting Polygons
- 2254 5/26/2010–1/25/2013—wms 8.3 BUG - opening jpeg when language is korean
- 2243 5/24/2010–1/25/2013—convert dem to filtered TIN crash
- 2131 4/23/2010–1/25/2013—Error building pyramids
- 2238 5/20/2010–1/25/2013—Problem swapping edges or inserting breaklines on TINs
- 2232 5/19/2010–1/25/2013—In MODRAT when you input a hydrograph it only uses the first 1500 minutes of the hydrograph.
- 2180 5/4/2010–1/25/2013—DEM contours to feature objects crash in WMS 8.3
- 2219 5/13/2010–1/24/2013—Crash when closing WMS after reading stochastic GSSHA solution
- 2124 4/22/2010–1/24/2013—Issue reading an image file
- 2081 4/14/2010–1/24/2013—WMS often exports duplicate points in HEC-RAS cross-sections when using bank arcs
- 2053 4/7/2010–1/24/2013—After running the SMPDBK model some of the output data is inconsistent.
- 2049 4/6/2010–1/24/2013—Hydrologic Modeling Tree display
- 2018 3/29/2010–1/24/2013—Terraserver (MSRMaps) issue
- 2017 3/29/2010–1/24/2013—Cropping collar problem with TIFFs in WMS 8.3
- 2016 3/29/2010–1/24/2013—Problems reading JPEG and TIFF Images in WMS 8.3
- 2015 3/29/2010–1/24/2013—area calculation of basins
- 1987 3/15/2010–1/24/2013—Closed bracket in WMS title bar moves one space while panning in WMS
- 1985 3/12/2010–1/24/2013—HMS Interface does not include kinematic wave reach routing for rectanglular or triangular channels
- 1943 3/3/2010–1/24/2013—SWMM link and node information not saved
- 1859 2/3/2010–1/24/2013—WMS 8.2 not able to save the 3 DEMs
- 1823 1/25/2010–1/24/2013—Unable to rename Cross section reach name
- 1779 1/15/2010–1/24/2013—Crop – Uncrop Collar not working on jpeg
- 1751 1/5/2010–1/24/2013—*Delineate Basins Wizard* shifts the TOPAZ stream network
- 1743 1/4/2010–1/24/2013—NSS not showing correct regions for Utah
- 1742 1/4/2010–1/24/2013—HEC-1 will not run while an image is loaded
- 1673 12/3/2009–1/24/2013—The SetItemData function changed from taking a DWORD to DWORD_PTR. 64-bit issue

- 1249 7/31/2009–1/23/2013—Importing multiple USGS DTED DEMs of varying resolutions creates problems
- 1239 7/30/2009–1/23/2013—NSS reports 0% error for all simulations
- 1238 7/30/2009–1/23/2013—Change tutorial files installation path to \My Documents instead of \Program Files
- 1232 7/29/2009–1/23/2013—GSSHA wiki, Tutorial 3 Fixing Digital Dams, Display Options Crash
- 1214 7/23/2009–1/23/2013—The shapefile "basins.shp" does not display properly in the Chapter 2, Maricopa County tutorial.
- 1171 7/15/2009–1/23/2013—Problem writing a KMZ animation file from WMS 8.2
- 1151 7/13/2009–1/23/2013—The run1.wpr file located on 'files' is more complete than the Volume 3, Chapter 4, Stochasic Modeling tutorials calls for.
- 1142 7/10/2009–1/23/2013—Need better way of assigning HYDGRP parameters when joining SSURGO data
- 1089 6/29/2009–1/23/2013—Process combined index map using land use and soil maps instead of coverages
- 1029 6/11/2009–1/23/2013—Deleting arcs takes a long time (because of UpdateVertToArcMap in feConvertVertexToNode)
- 982 5/29/2009–1/23/2013—NFF Error in 8.0 and previous – Charles Lutter
- 894 5/5/2009–1/22/2013—NEXRAD radar rainfall division by 0 error
- 872 4/29/2009–1/22/2013—MODRAT does not compute any ordinates for hydrographs
- 861 4/28/2009–1/22/2013—Problem displaying image in GIS module with ArcObjects enabled
- 859 4/28/2009–1/22/2013—WMS not responding when extracting cross-sections from this TIN
- 651 3/4/2009–1/22/2013—Crash when converting from drainage to a 1-D hydr. centerline coverage
- 632 2/27/2009–1/22/2013—WMS Printing Text Bug
- 618 2/26/2009–1/22/2013—SWMM hydrauilc schematic data is not saved to or read from the wms or wpr project file
- 551 2/6/2009–1/22/2013—Using the default GSSHA *.cmt file to generate initial parameters
- 550 2/6/2009–1/22/2013—Verify GSSHA smooth streams to thalweg data is working correctly
- 539 2/6/2009–1/22/2013—WMS creates duplicate feature arcs when converting this CAD file to Feature Arcs
- 427 1/13/2009–1/22/2013—Change all *.wpr files in the WMS 8.2 tutorials to *.wms files
- 275 12/3/2008–1/22/2013—One too many discharge curves shown in reservoir storage capacity window for MODRAT
- 256 11/24/2008–1/22/2013—Adding and retrieving diversions messes up tree item display when panning
- 251 11/21/2008–1/18/2013—Interpolation error message in detention basin calculator
- 212 11/12/2008–1/18/2013—There is a heap corruption (writing beyond the bounds of allocated memory) when running through the TINs tutorial-difficult fix
- 198 11/10/2008–1/18/2013—CAD→TIN command not working for Bowman Dam CAD file
- 197 11/10/2008–1/18/2013—CAD file displays in WMS 8.0 but not WMS 8.1
- 838 4/22/2009–1/18/2013—The projection dialog needs to be larger- or at least be resizable
- 117 10/16/2008–1/18/2013—pon Completion of the second step or the Volume 6, Chapter 3 Tutorial (NEXRAD), WMS Crashes.
- 285 12/8/2008–1/18/2013—Optimize drawing the soil and land use legends
- 187 11/6/2008–1/18/2013—Problem importing infiltration parameters from saved GSSHA projects
- 172 10/31/2008–1/18/2013—Vanishing WMS Bug
- 5796 1/4/2013–1/18/2013—allow more than 300 rows of data in HEC-HMS XY series windows
- 5794 1/4/2013–1/18/2013—CN doesn't update in TR-55 when a project with a computed CN is added
- 5808 1/8/2013–1/18/2013—Stream link numbers don't refresh
- 5830 1/15/2013–1/17/2013—Letter "e" is missing in the word "Select" for Select grid column
- 5742 12/5/2012–1/17/2013—Problem getting NED DEM data for Hawaiian Islands
- 5813 1/10/2013–1/14/2013—Several bugs related to the OC Rational method
- 5783 12/28/2012–1/11/2013—GIS to feature objects wizard doesn't transfer land use names

- 5784 12/28/2012–1/10/2013—rename "Level2" option in GIS to feature objects wizard
- 4467 12/9/2011–1/10/2013—The WDM file can't be read when trying to define external sources when using HSPF.
- 5579 10/12/2012–1/9/2013—It takes a long time to read WMS Map files
- 4837-3/13/2012–1/8/2013—Make sure all the tutorials (especially the first few) have the correct directories where the files are located
- 5415 8/20/2012–1/4/2013—Changed to culvert parameters not saved
- 5530 10/1/2012–1/3/2013—(Your horizontal and vertical units are inconsistent)-message for some tutorials
- 5731 11/30/2012–12/27/2012—Crash when trying to run an OC Rational simulation
- 5571 10/11/2012–12/27/2012—Possible problem reading solutions from multiple scenarios
- 5712 11/28/2012–12/26/2012—WMS only updates the CELL ID when you select cells in a GSSHA model
- 5554 10/8/2012–12/14/2012—WMS does not recognize MODRAT project file or MODRAT solution file
- 5565 10/10/2012–12/14/2012—Add an option to modify the active dataset for more than one cell on your 2D grid
- 5570 10/11/2012–12/13/2012—Problem with tutorial 55: GSSHA Overland Flow
- 5592 10/15/2012–12/13/2012—Transition to new CheckForVersion function for version checking
- 5604 10/18/2012–12/12/2012—Color fill contour method is not working for displaying GSSHA solutions
- 5608 10/19/2012–12/12/2012—Issue with computing the CN for a TR55 model
- 5610 10/19/2012–12/12/2012—Problem downloading data when using UTM coordinates with units of feet
- 5607 10/19/2012–12/11/2012—Problem downloading data when using State Plane coordinates
- 5700 11/19/2012–12/10/2012—The fields for Point #3 are not active in the Register Image dialog
- 5744 12/6/2012–12/10/2012—Problem Merging TINs
- 5750 12/7/2012–12/10/2012—The XY Series editor is causing WMS to crash
- 5706 11/20/2012–12/6/2012—The crossings calculated by HY8 are not shown in the HY8 wizard in WMS
- 5678 11/9/2012–12/3/2012—Problem with numbering WMS soil IDs
- 5656 11/6/2012–12/3/2012—Use the same shared code to determine resolution when using the get data tools or when exporting/converting raster data
- 5655 11/6/2012–11/30/2012—Create export and conversion options for all types of raster data
- 5569 10/11/2012–11/30/2012—Fix dialog tab ordering
- 5654 11/6/2012–11/28/2012—Use the same shared code when converting or exporting online or static raster data to another format
- 5653 11/6/2012–11/21/2012—Share the *Get Online Maps* dialog with the **Get Data** tools in webservices.cpp
- 4949 4/4/2012–11/16/2012—clicking the define button in the HSPF segments dialogue takes a very long time to open
- 5084 5/7/2012–11/16/2012—Implement computing GSSHA index maps using land use and soil type grids (8 hours)
- 5551 10/5/2012–11/6/2012—Can't compute flow directions and accumulations in GSSHA Tutorial
- 5550 10/5/2012–11/6/2012—Missing Information in the Landuse ID box
- 5563 10/9/2012–11/6/2012—WMS is crashing when using Hydraulic Toolbox to generate Storage Discharge Curves
- 5561 10/9/2012–11/6/2012—WMS crashes when using the *Importing USGS DEMs* dialog
- 5213 6/20/2012–11/6/2012—We need to move the tutorials to live with our code.
- 5526 9/27/2012–11/6/2012—Share GMS' options to convert online data to static data (24 hours)
- 5594 10/15/2012–11/2/2012—Show the Teigha version in the WMS **Help | About** command
- 5593 10/15/2012–11/2/2012—Add Lat/Lon coordinates at the bottom of the WMS window, similar to GMS
- 5595 10/15/2012–11/2/2012—Remove the Quick Tour from the *WMS Help* menu
- 5624 10/25/2012–10/31/2012—HY-12 is not building
- 5583 10/12/2012–10/31/2012—A separate coverage is created when converting CAD objects

- 5577 10/11/2012–10/31/2012—The vertical units are changed when reading in a DEM with *.asc extension
- 4795 3/5/2012–10/30/2012—HEC time of concentration not converted from min to hr
- 585 2/13/2009–10/30/2012—Error when trying to run HSPF
- 4937 4/2/2012–10/30/2012—"Could not open WDM file" error during HSPF tutorial
- 5615 10/22/2012–10/25/2012—When saving a file in WMS, the *.hyd file can't be created after running the hydraulic toolbox
- 5146 5/31/2012–10/23/2012—The time series data editor is not working
- 5002 4/20/2012–10/23/2012—Menu command to read a GEoTIFF DEM
- 5467 9/6/2012–10/22/2012—WMS crashes if a feature point is created where a point already exists
- 5093 5/9/2012–10/22/2012—WMS crashes when you close the program if you're using a storm drain FHWA coverage
- 5472 9/7/2012–10/19/2012—Problems with mapping attributes from storm drain shapefile to storm drain coverage (submitted by Nate Dye)
- 5474 9/7/2012–10/18/2012—TOPAZ does not run on computers with the Microsoft visual studio 2005 redistributable package
- 5496 9/18/2012–10/17/2012—WMS has a hard time downloading certain types of elevation data from Web Services
- 5221 6/21/2012–10/17/2012—WMS shows no graphics for lexington.dwg
- 5511 9/24/2012–10/17/2012—Need to frame after reading a *.dgn file (maybe *.dwg file also)
- 5512 9/24/2012–10/17/2012—Crash in **DEM | Point Attributes** for multiple cells
- 4390 11/23/2011–10/17/2012—QC Testing – HY-12 Tutorial errors
- 5033 4/25/2012–10/17/2012—The channel calculator opens up when you click **Create Stage Point**
- 5529 9/28/2012–10/2/2012—Convert to Static Image is not working

Release 9.0/Intermediate Release 9.0.8 – October 5, 2012

WMS 9.0.8 has been posted on Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/downloads [3]. The following known bugs have been fixed in this version:

Bug ID Date Submitted—Updated-Summary

- 5399 8/15/2012–9/28/2012—WMS should read in HEC-RAS water surface elevations from *.geo files
- 5492 9/17/2012–9/27/2012—WMS 9.0 freezes when attempting to load a GeoTIFF with land use data
- 5447 8/31/2012–9/26/2012—Texture Mapping Behavior
- 5389 8/13/2012–9/26/2012—Duplicate TINs option is messed up
- 5390 8/13/2012–9/25/2012—Problem adding points to a TIN
- 4828 3/9/2012–9/13/2012—Texture mapping does not work
- 5383 8/9/2012–9/13/2012—Contours lines turn themselves back on when you adjust a node of a texture mapped tin
- 5388 8/13/2012–9/13/2012—Crash when converting a 2D scatter file to a TIN
- 5411 8/17/2012–9/13/2012—crash when downloading from web
- 5439 8/30/2012–9/12/2012—Problem running GSSHA when max-min/NUM_INTERP for a break point cross section > 1.0
- 5440 8/30/2012–9/12/2012—WMS crashes when you double click on multiple basins in a row
- 5461 9/5/2012–9/12/2012—Option to Export Image
- 5462 9/5/2012–9/12/2012—WMS crashes when trying to change the attributes of an arc created outside a GSSHA grid
- 5465 9/6/2012–9/11/2012—WMS progress bar stays at zero when loading some GeoTiff files

- 5302 7/18/2012–9/10/2012—Warning message remains on screen after WMS finishes writing files for HMS
- 5199 6/13/2012–9/5/2012—WMS crashes after closing the Channel Calculations dialog
- 5201 6/14/2012–8/30/2012—WMS crashes after closing the GSSHA Contaminants dialog
- 5212 6/20/2012–8/30/2012—crash when defining a detention basin
- 5224 6/25/2012–8/30/2012—NSS crashes when the user enter a basin area of zero and presses the compute results button
- 5424 8/24/2012–8/29/2012—There is an error message when trying to export a GIS file WMS 9.0x64
- 5251 7/2/2012–8/27/2012—Handle data outside of selected bounds for the Get Data command (6 Hours)
- 5253 7/2/2012–8/23/2012—Check Defining Cross Section D and Z values (6 Hours)
- 5265 7/6/2012–8/21/2012—Problem with assigning mapping table values from a GSSHA mapping table file (again)
- 5272 7/11/2012–8/20/2012—drainage basin display options button does not resize with display options window
- 5304 7/20/2012–8/20/2012—crash when selecting a basin, exiting, and then selecting the basin again
- 5308 7/20/2012–8/20/2012—WMS crashes when trying to select a basin
- 5309 7/20/2012–8/17/2012—WMS freezes when attempting to load a project file
- 5311 7/20/2012–8/16/2012—extracted cross sections are different when using TIN and DEM
- 5340 7/27/2012–8/16/2012—WMS freezes when reading in a GSSHA solution
- 5325 7/24/2012–8/16/2012—input hydrograph file not written correctly in MODRAT project
- 5338 7/26/2012–8/15/2012—WMS crashes/freezes when trying to convert from shapefile to feature objects
- 5324 7/23/2012–8/14/2012—WMS crashes after closing the HGL and EGL profiles window
- 5195 6/13/2012–7/24/2012—Need a way to import storm drain attributes from a shapefile to WMS storm drain module
- 5143 5/31/2012–6/22/2012—contour legend not included in print out
- 5113 5/16/2012–6/22/2012—WMS partially freezes when saving a file in the Hydrologic Modeling Module
- 5114 5/16/2012–6/22/2012—Image files for bug report 5113.
- 5104 5/11/2012–6/21/2012—Crash when viewing land use attribute legend
- 5021 4/23/2012–6/21/2012—The Define... button in the Detention Basin Hydrograph Routing dialog doesn't work
- 5145 5/31/2012–6/20/2012—Problem with units when defining HEC-1 reservoir data
- 5141 5/30/2012–6/18/2012—WMS crashes when you close the Channel Analysis dialog
- 5103 5/11/2012–6/18/2012—Extra quotes added at the end of a weir (and possibly other structures) when launching the detention basin calculator from WMS
- 4816 3/7/2012–6/18/2012—WMS Crash converting DEM to 2d grid
- 5131 5/22/2012–6/12/2012—WMS cannot export Thiessen Polygons to shape file if the raingage type is set to HEC
- 4829 3/9/2012–6/11/2012—Convert all triangle functions (such as gmComputeCircumcircle) to use shared triangle functions instead of WMS functions
- 4846 3/14/2012–6/7/2012—Problem with canceling getting data using web services
- 4850 3/14/2012–6/6/2012—Problem getting DEM data for really large areas
- 4971 4/12/2012–6/6/2012—WMS partially freezes when when trying to define storage capacity data
- 4847 3/14/2012–6/6/2012—-Bogus % Complete number in progress bar
- 4843 3/13/2012–6/4/2012—Crash in Basic Feature Objects tutorial-setting soil attributes
- 4963 4/6/2012–5/30/2012—HY-8 Wizard storage-capacity issue
- 5024 4/24/2012–5/18/2012—Select/Zoom Hydrologic Tree Elements
- 4946 4/4/2012–5/16/2012—Check the segment renumbering for CE-QUAL-W2
- 4990 4/18/2012–5/16/2012—Create Feature Point option in Single Point Reprojection dialog doesn't work

- 4994 4/18/2012–5/15/2012—Cannot save HEC-RAS model without a cross-section database without getting errors
- 4997 4/18/2012–5/15/2012—Crash when saving NSS project
- 5041 4/26/2012–5/11/2012—The Clean Up Model button in the GSSHA wizard is not working
- 5046 4/27/2012–5/11/2012—An additional (0,0) point is created when creating cross sections
- 5028 4/24/2012–5/8/2012—Option to change to metric units
- 5054 5/1/2012–5/8/2012—WMS crashes when you close the Constituent Mass window
- 5026 4/24/2012–5/7/2012—DEM doesn't overlap
- 5015 4/23/2012–5/7/2012—DEM and georeferenced image don't overlap
- 4855 3/15/2012–5/7/2012—WMS crashes when color filling soils polygons
- 5039 4/26/2012–5/3/2012—WMS crashes when you try to compute basin data (2)
- 5014 4/23/2012–5/3/2012—WMS crashes when you try to compute basin data
- 5020 4/23/2012–5/3/2012—WMS crashes when trying to change the projection for the DEM
- 5036 4/25/2012–5/3/2012—WMS crashes when you try to reproject a DEM
- 4845 3/14/2012–5/2/2012—Fix link in DEM Delineation tutorial: Section 4.4
- 4849 3/14/2012–5/1/2012—Specify toggle box has changed to "Set" toggle box in the Reprojection dialog
- 4883 3/21/2012–4/24/2012—HY8UnitsConversion.cpp and possibly other files are in 2 places
- 4872 3/19/2012–4/19/2012—The coverage type and attributes are changed when opening .map files
- 4995 4/18/2012–4/18/2012—WMS is not automatically updating the Energy Slope for routing.
- 4858 3/15/2012–4/18/2012—WMS can't find cross-section database
- 4868 3/19/2012–4/18/2012—WMS crashes/freezes when trying to open a *.map file
- 4579 1/12/2012–4/17/2012—Add global mapper as an option for elevation file I/O
- 4884 3/21/2012–4/12/2012—Concentration Points are not shown in the Orange County Rational Method – Tree Mapping dialog
- 4916 3/29/2012–4/12/2012—Add option to prevent values from being overirdden when computing basin data
- 4936 4/2/2012–4/9/2012—crash when trying to convert TIN to DEM
- 4926 3/30/2012–4/9/2012—WMS crashes when trying to assign precipitation info
- 4873 3/19/2012–4/6/2012—There is a problem loading HEC-RAS data
- 4760 2/23/2012–4/5/2012—crash when using the "GIS Data → Dataset" button for a GSSHA model
- 4938 4/2/2012–4/4/2012—WMS crashes if you try to close the 'Select Orange County GIS losses output file' dialog
- 4814 3/7/2012–4/4/2012—Index maps can't be created when using Data Calculator
- 4314 11/2/2011–3/29/2012—Add option to reverse the zoom direction of the mouse wheel
- 4697 2/9/2012–3/23/2012—Crash when loading in multiple DEMs
- 4786 3/1/2012–3/23/2012—Advanced DEM delineation tutorial issue
- 4712 2/13/2012–3/23/2012—Hydraulic Toolbox and WMS link is broken
- 4674 2/6/2012–2/29/2012—Problems with TIN breaklines
- 4469 12/9/2011–2/29/2012—Converting TIN to DEM results in bad data points
- 4677 2/6/2012–2/28/2012—freeze when trying to import GSSHA precipitation gage file
- 4661 2/1/2012–2/28/2012—OC Hydrograph GIS tutorial has the wrong title
- 4536 1/4/2012–2/28/2012—WMS in not properly creating the radarrain and radargrid *.dss files for HEC-HMS
- 4561 1/9/2012–2/27/2012—WMS is not properly creating the .dss file for HEC-HMS
- 4570 1/10/2012–2/27/2012—WMS crashes when trying to change display options for index maps
- 4645 1/30/2012–2/24/2012—Improper Unit
- 4664 2/2/2012–2/24/2012—WMS crashes while clicking "Next Hydrograph Station" button
- 4667 2/2/2012–2/24/2012—WMS writing wrong Overland BC code
- 4678 2/7/2012–2/22/2012—Slow printing when using an online image

- 4474 12/9/2011–2/22/2012—WMS in not properly creating the *.dss file for HEC-HMS
- 4318 11/2/2011–2/21/2012—HEC-HMS tutorial needs to be fixed
- 4412 11/30/2011–2/17/2012—WMS crashes when trying to change the display options for TINs
- 3578 5/26/2011–2/17/2012—HMS Properties Spreadsheet loses scroll location after entering Basin Data Tc dialog
- 4723 2/15/2012–2/15/2012—Saving as a google earth file does not work correctly.
- 4082 9/2/2011–2/13/2012—A single DEM appears twice in the project explorer.
- 3651 6/10/2011–2/13/2012—SCE file not found in GSSHA automatic calibration
- 4078 9/2/2011–2/13/2012—HSPF dialog takes a long time to appear
- 4376 11/17/2011–2/10/2012—Crash when importing EPA SWMM file
- 4266 10/21/2011–2/6/2012—DEM Does not default to correct coordinate system (projection) when importing
- 4265 10/21/2011–1/26/2012—Problem finding cross section database in attached file
- 4413 11/30/2011–1/26/2012—Web data not downloading

Release 9.0 Beta – January 24, 2012

WMS 9.0 Beta has been posted on Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/downloads [3]. A list of significant new features is located here. The following known bugs have been fixed in this version:

Bug ID Date Submitted—Updated-Summary

- 4280 10/25/2011–1/24/2012—Problem with GSSHA stream arc file I/O in *.cif file
- 4279 10/25/2011–1/24/2012—Problem with GSSHA overland boundary condition in *.cmt file
- 4491 12/14/2011–1/20/2012—The check box in the All row in the properties dialog doesn't work
- 4489 12/14/2011–1/18/2012—HY-8 Shapes crash
- 4390 11/23/2011–1/18/2012—QC Testing – HY-12 Tutorial errors
- 4520 1/3/2012–1/18/2012—Crash when trying to create HY-12 structure
- 4307 11/1/2011–1/17/2012—Images tutorial needs to be fixed
- 4308 11/1/2011–1/16/2012—DEM Basics tutorial needs to be fixed
- 4311 11/2/2011–1/16/2012—DEM Delineation Tutorial needs to be fixed
- 4313 11/2/2011–1/16/2012—Advanced DEM Delination Tutorial needs to be fixed
- 4436 12/6/2011–1/13/2012—Extract elevation data from the Global Mapper online elevation data options instead of from the USGS seamless site
- 4087 9/6/2011–1/11/2012—Gridded gage coverage display issues
- 4157 9/22/2011–1/10/2012—Material not saved in "Land Poly Atts" window
- 4421 12/1/2011–1/9/2012—Date/Time Display on the Hydrograph Plot window
- 4494 12/15/2011–1/9/2012—WMS crashes while trying to read GSSHA solution
- 4501 12/19/2011–1/6/2012—Minor HY-8 tutorial issue
- 4431 12/6/2011–1/5/2012—Fix problem with WMS framing if you have an online map and you download a DEM from online data
- 4518 12/29/2011–1/5/2012—New Tool for contouring based on a selected area needs a name
- 4432 12/6/2011–1/5/2012—Add a forward and back view button to the WMS interface in the static tools, similar to the buttons in HydroDesktop
- 4528 1/3/2012–1/4/2012—Soil erosion parameters in cmt file is saved incorrectly
- 4433 12/6/2011–12/30/2011—Add an option to convert online images to a normal (static) image in the map module
- 4443 12/6/2011–12/14/2011—Add elevation data site to GSDA site
- 4434 12/6/2011–12/14/2011—Create a separate non-global-mapper interface for accessing online images

- 4451 12/6/2011–12/12/2011—Make the MODRAT renumber selection dialog modeless
- 4423 12/1/2011–12/9/2011—GSSHA Overland Boundary Condition
- 4199 10/5/2011–12/7/2011—Crop Collar and Uncrop Collar commands not active
- 4379 11/18/2011–12/7/2011—WMS crashes when tryring to crop the collar of an image
- 4446 12/6/2011–12/7/2011—Remove or fix Crop Collars in WMS (4 hours)
- 4086 9/6/2011–12/6/2011—NEXRAD MODClark Tutorial Issues
- 4139 9/16/2011–12/5/2011—Only one point shows up when loading WMS project with MODRAT model
- 4194 10/3/2011–12/2/2011—Contours for TIN-> DEM Conversion not displaying at all
- 4089 9/6/2011–12/1/2011—Hydrologic tree does not appear
- 4128 9/14/2011–12/1/2011—WMS crashes section 8.1 "Orange County Unit Hydrograph – GIS" tutorial.
- 4096 9/7/2011–11/30/2011—NSS not registered
- 4102 9/8/2011–11/30/2011—HEC-1 does not run after adding a diversion
- 4127 9/13/2011–11/30/2011—Mean basin slope not assigned for some equations in NSS
- 4071 9/1/2011–11/29/2011—WMS 8.4.21 crashes when trying to load GSSHA project file
- 4211 10/7/2011–11/29/2011—Crash when trying to load *.gdm file
- 4204 10/6/2011–11/29/2011—Crash when trying to smooth streams
- 4240 10/14/2011–11/28/2011—GSSHA crash when running long term richard's infiltration simulation
- 4225 10/12/2011–11/28/2011—Rewrite the code that displays the embankment cells in a GSSHA project
- 4198 10/5/2011–11/22/2011—Updates to the Data Calculator
- 4272 10/24/2011–11/11/2011—Time Series Editor
- 4271 10/24/2011–11/11/2011—Time Series Editor, Split time series for GSSHA
- 3638 6/7/2011–11/8/2011—Assertion when going to HY-12 Junction data
- 4057 8/29/2011–11/8/2011—Modifying soil input field in the MODRAT Job Control changes rain input field
- 3637 6/7/2011–11/8/2011—HY-12 Structure types are not sorted correctly for links
- 4161 9/23/2011–11/1/2011—MODRAT Tc values greater than 30 minutes capped at 30 minutes without a warning
- 4168 9/26/2011–10/31/2011—Problem with Use Thalweg Data command in the GSSHA Stream Profile Editor
- 4191 9/30/2011–10/31/2011—Input hydro graph loses data if has more than 300 time steps.
- 3977 8/12/2011–10/28/2011—Tc value does not appear where it should
- 3963 8/11/2011–10/27/2011—WMS crashes when I include pipe node in/out flow
- 4009 8/18/2011–10/26/2011—WMS crashes when clicking depth solution
- 4062 8/30/2011–10/26/2011—OC Rational Tutorial Won't Run
- 3697 6/22/2011–10/25/2011—WMS 8.4 does not save re-projection
- 3738 7/5/2011–10/24/2011—Window is not big enough for text
- 3978 8/12/2011–10/24/2011—Must save WMS project before calculating CN
- 4094 9/7/2011–10/21/2011—Uninstallation of WMS 8.4 does not work
- 3981 8/15/2011–10/21/2011—WMS 8.4.19 crashes in SMPDBK tutorial
- 4007 8/18/2011–10/20/2011—Crash when creating an arc attached to outlet
- 3644 6/9/2011–10/20/2011—Issue with storm/tile drains
- 3886 8/2/2011–10/14/2011—Rational solution not saved in EPA SWMM file unless certain contitions met
- 3889 8/2/2011–10/14/2011—Node and link elevations not saved in WMS
- 3912 8/8/2011–10/14/2011—WMS not saving invert elevations for SWMM interface.
- 3946 8/10/2011–10/14/2011—WMS not reading in Link and Node elevations or names from EPA SWMM file
- 3947 8/10/2011–10/14/2011—WMS not reading link and node names from xpSWMM file.
- 4205 10/6/2011–10/11/2011—Input hydro graph truncated for modrat simulation
- 4080 9/2/2011–10/3/2011—Problem with saving EPA-SWMM data
- 3648 6/9/2011–9/23/2011—WMS 8.4 crashes when you select Remove Flat Triangles

- 3680 6/16/2011–9/16/2011—include build number at end of WMS in "Help | About" dialog
- 3722 6/30/2011–9/15/2011—Updating Tutorial
- 3832 7/20/2011–9/15/2011—Map to Hydrologic Model option not working in Detention Basin Editor
- 4081 9/2/2011–9/14/2011—Changes to GSSHA .cmt file for WMS 9.0
- 3985 8/15/2011–9/12/2011—GSSHA calibration Parameter replacement issue
- 4035 8/25/2011–9/12/2011—Add the OVERLAND_BACKWATER card to the WMS GSSHA interface
- 4108 9/8/2011–9/9/2011—WMS becomes unresponsive when working with a CE-QUAL-W2 control file
- 3703 6/24/2011–9/7/2011—GSSHA modeling basics infiltration tutorial bug
- 3611 6/2/2011–9/6/2011—Add the GMS unit test debug dialog to shared code and share it with WMS
- 3403 4/13/2011–9/1/2011—Problem computing overbank flow when running GSSHA (likely a GSSHA bug or potential problem)
- 3671 6/15/2011–8/31/2011—Hydraulic Toolbox Tutorial quick fixes for Eric or Richard (2 hours)
- 3324 3/29/2011–8/31/2011—Problem with MODRAT Soil file definition
- 3601 6/1/2011–8/31/2011—Font settings lost on some display options in dev.
- 3481 5/4/2011–8/30/2011—Cropping collar using right click menu not working properly
- 2997 1/20/2011–8/30/2011—error about HEC programs during WMS 8.4 install
- 3569 5/25/2011–8/30/2011—MODRAT-GIS Tutorial Issues
- 3440 4/25/2011–8/29/2011—Review Spatial rainfall tutorials
- 3441 4/25/2011–8/29/2011—Review MODRAT tutorials
- 3378 4/6/2011–8/25/2011—Plots not showing up in OC rational method GIS tutorial
- 3414 4/18/2011–8/25/2011—Error when running OC Rational Tutorial
- 2805 11/19/2010–8/23/2011—Unable to write Area-Reduction Factor in GSSHA mapping table file
- 3402 4/12/2011–8/23/2011—Area reduction GSSHA table not included in WMS
- 3494 5/9/2011–8/23/2011—Could we add a new tool that would contour a given area?
- 3810 7/15/2011–8/22/2011—"Could not assign length and orientation" error when trying to "Get lengths and orientation of branch segments" in CE-QUAL-W2 tu
- 3249 3/15/2011–8/16/2011—Values multiplied by 3.28 when exporting a GeoTIFF
- 3895 8/3/2011–8/15/2011—CE-QUAL-W2 crashes when trying to import boundary conditions for a branch
- 3885 8/2/2011–8/15/2011—CE-QUAL-W2 crashes when trying to import a meteorological file
- 3813 7/15/2011–8/12/2011—Resizing WMS causing computer to freeze
- 3807 7/14/2011–8/12/2011—Crash when double clicking title bar
- 3803 7/13/2011–8/10/2011—Coordinate Conversion Crash
- 3796 7/12/2011–8/9/2011—See if 10-85 basin parameters are being transferred to the NSS database
- 3743 7/6/2011–8/4/2011—More HY-8 Wizard Tutorial Issues
- 3742 7/6/2011–8/4/2011—WMS crashes when doing DEM → Stream Arcs
- 3741 7/6/2011–8/4/2011—WMS freezes when delineating watershed in HY-8 wizard
- 3560 5/24/2011–8/4/2011—WMS 8.4 HSPF Bugs to fix (12 Hours)
- 3496 5/9/2011–8/3/2011—Clean up the hydraulic toolbox tutorial and put it online
- 3477 5/3/2011–8/1/2011—Extracting X-sections error if Elevation data is not selected
- 3478 5/3/2011–8/1/2011—Exporting KMZ vector file
- 3503 5/9/2011–7/28/2011—Export TIN-based filmloop to Google Earth
- 3497 5/9/2011–7/28/2011—Bugs that came up in the HY8 modeling wizard tutorial
- 3515 5/11/2011–7/27/2011—Not all soil types mapped to index map
- 3612 6/2/2011–7/26/2011—Add integration tests to WMS
- 3525 5/12/2011–7/25/2011—Error message while converting feature arcs to TIN
- 286 12/8/2008–7/20/2011—Rewrite Infiltration tutorial on www.gsshawiki.com
- 1511 10/12/2009–7/20/2011—New NSS databases not compatible with WMS 8.2

- 995 6/3/2009–7/20/2011—In the 2D Scatter Data tab of the Display Options dialog decimal Z magnifications are not currently supported.
- 3681 6/16/2011–7/18/2011—error when trying to install WMS 8.4.15 when WMS 8.4.14 is already installed
- 3675 6/15/2011–7/18/2011—SWMM tutorial not exporting flowrates
- 3678 6/16/2011–7/15/2011—SWMM Tutorial Bug: WMS not writing hydrograph information to EPA SWMM
- 3672 6/15/2011–7/14/2011—Bug in SWMM tutorial: area exported in sq. mi. even though computed in acres
- 3083 2/8/2011–7/14/2011—SWMM tutorial bugs
- 3588 5/31/2011–7/13/2011—Problems with Modified Puls xy series editor in HEC-HMS Parameters dialog
- 3366 4/5/2011–7/5/2011—Export a filmoop or KMZ from a TIN
- 1648 11/23/2009–7/5/2011—WMS runs out of memory when converting one TIN contour to feature objects.
- 2118 4/21/2010–7/5/2011—Clean→Snap Selected Nodes command gives you an hourglass instead of a select cursor
- 1708 12/16/2009–7/5/2011—WMS not Print Correctly
- 1496 10/7/2009–7/5/2011—Problem reading *.wpr or *.wms file with TIFF file
- 996 6/3/2009–7/5/2011—wmsnss.exe crash
- 865 4/28/2009–7/5/2011—Check to see if the fix for the single-basin at outlet problem has worked out in the dev version
- 3732 7/1/2011–7/5/2011—Freeze when trying to delineate a basin in WMS 8.4
- 3513 5/10/2011–6/30/2011—Crash or freeze when reading solution
- 3713 6/29/2011–6/29/2011—Problem saving HMS file
- 3531 5/13/2011–6/15/2011—WMS crash while converting DEM to TIM
- 3609 6/2/2011–6/7/2011—Issues with new combo boxes in Film Loop Setup
- 3590 5/31/2011–6/7/2011—Hydraulic Toolbox tutorial issue
- 3634 6/6/2011–6/7/2011—Problem filtering TIN
- 3492 5/9/2011–6/6/2011—Filtering angle option for TIN filtering not working properly
- 3404 4/13/2011–6/2/2011—Add a pre-link event to copy shapesdb.dat (used in HY8) to the 64-bit WMS folder (1 hour)
- 3431 4/25/2011–6/2/2011—Problem reading GSSHA project and GSSHA solution
- 3475 5/3/2011–6/1/2011—Model Check Error when doing GSSHA
- 3463 4/29/2011–5/31/2011—GSSHA card improperly written out in *.cif file
- 3396 4/11/2011–5/31/2011—Index maps do not display in GSSHA model
- 3397 4/11/2011–5/16/2011—GSSHA Film Loop
- 2550 9/7/2010–5/11/2011—Need to be able to assign land use IDs manually
- 3471 5/2/2011–5/3/2011—WMS crashes when opening hydrograph plot from palmer canyon GIS MODRAT model
- 3239 3/14/2011–5/2/2011—problem with base files in GSSHA tutorials
- 3240 3/14/2011–4/29/2011—GSSHA stops prematurely after adding a detention basin
- 3231 3/11/2011–4/21/2011—Problem with flow accumulations at depression points
- 3248 3/15/2011–4/20/2011—WMS 8.4 always says you're out of date
- 3261 3/17/2011–4/19/2011—Redo the way we handle stream and cell boundary conditions in GSSHA
- 3254 3/15/2011–4/19/2011—Error when selecting below last time step of GSSHA depth output
- 3348 3/31/2011–4/18/2011—Problem with downloading land use data (possibly other web services also)
- 3368 4/5/2011–4/14/2011—Bug in GSSHA Modeling Basics Infiltration tutorial (.cmt file problem--check)
- 3369 4/5/2011–4/13/2011—Bug in GSSHA - Modeling - Wizard tutorial-GSSHA model checker errors
- 3371 4/5/2011–4/13/2011—Problem with running GSSHA with wells and saving GSSHA well files
- 3372 4/5/2011–4/11/2011—crash when running GSSHA
- 3344 3/30/2011–4/6/2011—Minor Model check problem after opening existing project file

- 3296 3/24/2011–4/6/2011—WMS hangs when processing NEXRAD radar rainfall grids
- 3298 3/24/2011–4/6/2011—Fix all problems with importing and using radar rainfall grids
- 3365 4/5/2011–4/5/2011—Remove a the "Crop Image" page from the wiki
- 3295 3/24/2011–4/5/2011—Crash when reading GSSHA project with different size of grid cells than the .msk file in the index map
- 3338 3/30/2011–4/5/2011—Select a better default for computing gage weights
- 3337 3/30/2011–4/5/2011—Add a separate menu command to compute rainfall gage weights
- 3330 3/30/2011–4/4/2011—Basin Time Computation Method is not saved
- 3345 3/31/2011–4/4/2011—Problem with debris production estimates from MODRAT GIS Tutorial
- 3343 3/30/2011–4/1/2011—Percent Impervious not mapped correctly from shapefile data
- 3272 3/18/2011–3/22/2011—WMS basin boundaries not drawn completely in HEC-HMS (Willdan project)
- 3157 2/22/2011–3/17/2011—Collar not cropped completely
- 2992 1/20/2011–3/17/2011—WMS freezes when mapping to 1D schematic
- 2993 1/20/2011–3/17/2011—WMS 8.4 freezes when trying to load a wms project file
- 3255 3/16/2011–3/16/2011—WMS 8.4 installs HEC-RAS 4.1, but tries to launch 4.0
- 3016 1/26/2011–3/16/2011—"Error reading scalar binary file, (floatarray)" appears when loading GSSHA depth solution
- 3018 1/27/2011–3/15/2011—Second window appears after importing HEC-RAS GIS file
- 2051 4/6/2010–3/15/2011—WMS crashes when trying to open RainGageBug.wms
- 3019 1/27/2011–3/15/2011—Importing .cmt file into GSSHA Map Table Editor does not work
- 3205 3/3/2011–3/15/2011—Problem with HMS diversions in Willdan project
- 3014 1/26/2011–3/14/2011—Problem with trying to read SCS default rainfall distributions when a different version of WMS is installed then uninstalled
- 2991 1/20/2011–3/14/2011—error message each time opening WMS 8.4 64-bit
- 2903 12/22/2010–3/14/2011—Loading Projection file
- 2900 12/21/2010–3/14/2011—Crash when generating a GSSHA grid
- 3204 3/3/2011–3/11/2011—WMS not saving "description" field in HMS interface
- 3198 3/1/2011–3/11/2011—% Impervious not displaying properly while Computing GIS Attributes
- 3180 2/28/2011–3/11/2011—Basin Time Computation for HEC-HMS
- 3079 2/8/2011–3/10/2011—TauDEM not working with depression points
- 3215 3/7/2011–3/10/2011—Confusing error message when network lock has problems
- 3197 3/1/2011–3/9/2011—Missing parameter in HEC-HMS Properties dialog
- 3194 3/1/2011–3/9/2011—Problem with using the hydraulic toolbox in the WMS HY8 modeling wizard
- 3073 2/4/2011–3/8/2011—HY-8 Wizard bugs
- 3195 3/1/2011–3/8/2011—Cancel on the Hydraulic Toolbox Storage capacity input dialog does not work
- 2921 1/5/2011–2/25/2011—Creating soil type coverage manually for use with GSSHA is problematic
- 3074 2/4/2011–2/25/2011—crash when delineating inundated area in HY-8 wizard
- 2976 1/18/2011–2/8/2011—NSS not functional in WMS 8.4

Release 8.4/Intermediate Release WMS 8.4.9 – February 3, 2011

An update to WMS 8.4 has been posted on Aquaveo's web site. The final version of WMS 8.4 was also released in December 2010. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug ID-Date Submitted—Updated-Summary

- 3049 1/31/2011–2/1/2011—error when converting NEXRAD data to *.dss
- 2681 10/15/2010–1/24/2011—Problem reading GSSHA sediment files
- 2680 10/15/2010–1/24/2011—Problem with reading/saving GSSHA contaminant file from WMS
- 2707 10/25/2010–1/20/2011—printed scale bar display error
- 2682 10/15/2010–1/20/2011—Problem writing GSSHA project file with overland flow boundary condition
- 2292 6/3/2010–1/20/2011—RAS import causing Issues
- 2505 8/17/2010v1/18/2011—Problem reading HEC-1 file and writing it as an HMS file
- 2665 10/11/2010–1/10/2011—DEM→Stream Arcs command not working for certain projects
- 2922 1/5/2011–1/7/2011—WMS 8.4 crashes when loading Willdan .asc dem files as a group
- 2549 9/7/2010–1/4/2011—WMS doesn't recognize all basins created w/ feature objects
- 2511 8/20/2010–11/3/2010—WMS HANGS when deleting arc
- 2624 10/4/2010–11/2/2010—Modrat Wrapper not finishing
- 2623 10/1/2010–11/2/2010—Joining SSURGO data bug
- 2565 9/9/2010–11/2/2010—Click on GSSHA Smooth Stream Dialog causes a crash
- 2400 7/12/2010–10/28/2010—Build Polygons Crash
- 2389 7/7/2010–10/28/2010—WMS 8.3 unable to Connect to Sever when selecting from web service
- 2629 10/5/2010–10/27/2010—WMS cannot load images after converting DEM to TIN
- 2436 7/21/2010–10/27/2010—WMS crashing when loading GIS data
- 2359 6/29/2010–10/27/2010—WMS 8.3 Crashing when loading Raster
- 2627 10/5/2010–10/25/2010—Severe WMS/Global Mapper bug when converting DEM to TIN
- 2597 9/23/2010–10/25/2010—When loading the hydrograph solution to HEC-1 WMS Gives an error
- 2508 8/19/2010–10/22/2010—WMS 84 can't load existing GSSHA model.
- 2546 9/7/2010–10/20/2010—WMS Clean tool gives an hourglass instead of an arrow cursor
- 2332 6/16/2010–7/19/2010—problems numbering Branch
- 2321 6/11/2010–7/16/2010—Run GSSHA model button in the Hydrologic modeling wizard not working
- 2311 6/9/2010–7/16/2010—Contour Options button not working
- 2288 6/2/2010–7/16/2010—Hydrologic modeling wizard Define project boundary
- 2244 5/24/2010–7/16/2010—flow vectors don't read in
- 2239 5/20/2010–7/16/2010—Save File button not working in Coverage Overlay dialog
- 2366 6/30/2010–7/15/2010—Display options
- 2184 5/5/2010–7/14/2010—Tutorial change request
- 2217 5/13/2010–7/13/2010—WMS not writing all lakes to file
- 2215 5/13/2010–7/13/2010—Simple dam Break not finding Cross Section
- 2237 5/20/2010–7/12/2010—Add the START_TIME and END_TIME cards to writing the GSSHA Project file and have fields for these cards in the GSSHA Job Control
- 2108 4/19/2010–7/8/2010—Changed the elevations of a few cross sections to get rid of pooling and then tried to rerun HECRAS but it wouldn't run because
- 2089 4/16/2010–7/8/2010—Cross sections Missing
- 2236 5/20/2010–7/2/2010—Crash when deleting GSSHA model
- 2234 5/19/2010–6/10/2010—Add option to select whether to export contaminant mass and concentration to GSSHA output control

- 2156 4/28/2010–6/10/2010—Error in GSSHA Calibratiobn
- 2153 4/28/2010–6/10/2010—GSSHA calibration output files
- 2060 4/8/2010–6/10/2010—Allow editing polygon-selected index map ID's in the properties window
- 2247 5/25/2010–6/9/2010—Add GSSHA Calibration Parameters in WMS
- 2235 5/19/2010–6/9/2010—Report the name of the contaminant when reading the contaminant transport solution
- 2249 5/25/2010–6/8/2010—Check while reading parameter and calibration file in GSSHA automated calibration
- 2248 5/25/2010–6/8/2010—WMS does not read observed data file
- 2075 4/12/2010–5/18/2010—"Error message when trying to open an image: ""The application has failed to start because gmp-vc90-mt.dll was not found"""
- 2065 4/9/2010–5/18/2010—"The ""Select Shapes Tool"" in the GIS module should be an active tool but it is inactive."
- 2141 4/26/2010–5/14/2010—WMS crashes when right clicking on a TIN Tree Item for a TIN that's been deleted
- 2147 4/28/2010–5/13/2010—Error reading GSSHA Stochastic Simulation results
- 2183 5/4/2010–5/12/2010—Save GSSHA Group Dialog
- 2151 4/28/2010–5/12/2010—Issues with GSSHA Automated Calibration
- 2175 5/3/2010–5/11/2010—converting dem contours to feature objects
- 2143 4/26/2010–5/11/2010—Multi-Select index map grid cells
- 1761 1/8/2010–4/19/2010—Louisiana SSURGO files will not download from Web Services
- 1753 1/5/2010–4/19/2010—Opening this GSSHA project file causes WMS 8.3 to crash
- 1754 1/6/2010–4/16/2010—WMS crashes when opening any GSSHA project with the attached *.ini file
- 2020 3/29/2010–4/14/2010—Problem converting shapefile contour lines to feature objects (and then to elevations on a TIN)

Other fixes and updates

- Fixed MODRAT/WMS bugs
- Rewrite all recursive functions associated with basin delineation and basin data computation
- Add parameters toggle in Soil Type/Land Use attributes dialog for Initial and Constant, and add attribute for Initial Loss (in) and Constant Infiltration Rate (in/hr)
- Automatically transfer Initial and Constant values to appropriate loss fields in HMS properties dialog (may require adding Initial Loss, Constant Infiltration Rate, and Percent Impervious to list of Basin data parameters)
- Change "Compute Basin Data" function to only compute selected basin data parameters (may significantly reduce computation time for large datasets)
- Modify HMS temporal distribution dialog to use more than 300 points

Intermediate Release WMS 8.3.4 – June 7, 2010

An update to WMS 8.3 has been posted on Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted—Updated-Summary

- 2071 4/12/2010–5/18/2010—Zoom tool not working in Edit DEM Elevations plot window
- 2084 4/14/2010–5/17/2010—DEM File won't read in
- 2038 4/1/2010–4/14/2010—Placing a contour label causes WMS 8.3 to crash

Intermediate Release WMS 8.3.3 – March 19, 2010

An update to WMS 8.3 has been posted on Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted—Updated-Summary

- 1860 2/4/2010–3/11/2010—Floating point error when running SMPDBK
- 1919 2/26/2010–3/11/2010—Loading dem changes Current Coordinates.
- 1944 3/3/2010–3/10/2010—Option to manually link nodes and outlets in SWMM does not work
- 1958 3/4/2010–3/10/2010—"When selecting the basin icon in the NSS model (tutorial volume 2, ch3), WMS gives error window"
- 1824 1/25/2010–3/9/2010—Floodplain delineation freeze
- 1825 1/25/2010–3/9/2010—Opening these DEMs with Current Coordinates set to State Plane causes WMS to crash

Release 8.3.0/Intermediate Release WMS 8.3.1 – January 26, 2010

An update to WMS 8.3 has been posted on Aquaveo's web site. The final version of WMS 8.3 was also released in December. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted—Updated-Summary

- 1762 1/8/2010–1/21/2010—"Change ""Join SSURGO data"" command to ""Join NRCS data"""
- 1731 12/22/2009–1/21/2010—GSSHA cross section data not saved.
- 1755 1/6/2010–1/20/2010—WMS 8.3 does not read in GSSHA custom cross-section data
- 1609 11/6/2009–1/14/2010—WMS coverages not overlaying DEM properly in WMS 8.3
- 1650 11/23/2009–1/13/2010—Converting TIN Contour to Feature Objects not working from menu
- 1664 12/1/2009–1/12/2010—WMS crashes upon opening the .wms file included in bug #1531
- 888 5/4/2009–12/3/2009—WMS 8.2 gets hung when trying to run on a computer with low resolution
- 1642 11/19/2009–12/3/2009—Bug when displaying GIS data using ArcObjects in Albers coordinate system
- 863 4/28/2009–11/19/2009—Problem reading land use ArcGrid with ArcObjects
- 1035 6/15/2009–11/18/2009—Save files to temporary directory/filename before writing over original file
- 1614 11/9/2009–11/18/2009—Map Data option for computing lag time in HEC-HMS doesn't transfer computed value
- 1621 11/12/2009–11/18/2009—HMS Lag time for computing Clark's R doesn't update when edited manually
- 1618 11/11/2009–11/11/2009—Button for Defining Inflow-Diverted flow in HMS Interface not working
- 1615 11/9/2009–11/10/2009—WMS crashes when exporting scatter data to ASCII grid file
- 1617 11/10/2009–11/10/2009—Basin area units wrong in HMS if computing basin data in acres
- 1612 11/9/2009–11/9/2009—WMS giving the wrong Values in the ENDFILES
- 1610 11/6/2009–11/9/2009—Land use shapefile descriptions not mapped to Level2
- 1251 7/31/2009–11/9/2009—Let users change rain gage display
- 450 1/15/2009–11/9/2009—WMS deletes TIN vertices and triangles when closing a project
- 271 12/2/2008–11/9/2009—Cancel button does not work on the web services dialog
- 237 11/19/2008–11/9/2009—Display doesn't refresh while dragging vertices
- 1597 11/3/2009–11/9/2009—WMS Crash
- 640 2/27/2009–11/5/2009—"Remove toggle box in CAD Data Display Options dialog – if not the whole dialog"
- 1559 10/21/2009–10/30/2009—WMS title bar modification to facilitate testing of the 'release' version and elimination of the 'release robot' version

- 1588 10/30/2009–10/30/2009—Problem with Extract Cross Sections dialog
- 316 12/12/2008–10/30/2009—Detention basin overflowed error when running the Rational Method
- 227 11/13/2008–10/30/2009—Display CAD data command does not turn CAD display off when toggled off
- 226 11/13/2008–10/30/2009—CAD Layers coverages are not highlighted when they are set as the active coverage
- 837 4/22/2009–10/29/2009—Why is the terraserver map scale dialog so big?
- 284 12/8/2008–10/29/2009—Remove CROSS_SECTION_LIST card from GSSHA files that WMS writes
- 415 1/8/2009–10/29/2009—"Upon Clicking ""Delineate Watershed"" (Tutorials, Vol II, Ch 5, Sec 5.2.7) the file name is omitted from the title bar."
- 1577 10/28/2009–10/28/2009—Default feature arc attribute not stored for 1D Hydraulic Centerline coverage
- 1572 10/26/2009–10/28/2009—"Section 2.3.1, Volume 2, Chapter 2, Step 11"
- 332 12/17/2008–10/28/2009—Properties window display bug
- 188 11/6/2008–10/28/2009—Error reading shapefiles with associated .prj files
- 178 11/4/2008–10/28/2009—Map data is hidden behind the 2D Grid when color filled contours are turned on in plan view
- 177 11/4/2008–10/28/2009—Problem displaying 2D Grid flow vectors with blocked cells and color filled contours.
- 851 4/24/2009–10/28/2009—"Sub-basins improperly delineated. Zero area error. Orange County Tutorial 6, section 6.3.2."
- 1481 10/2/2009–10/26/2009—WMS crashes when treating generic polygons as basins.
- 1568 10/23/2009–10/26/2009—MODRAT dialog warning of duplicate names when no duplicate names exist
- 1517 10/13/2009–10/26/2009—Problem with moving GSSHA projects to a different directory
- 832 4/21/2009–10/23/2009—Problem with sub-basin areas
- 1469 9/30/2009–10/20/2009—HEC-1 Data Points
- 1438 9/19/2009–10/19/2009—"Upon completion of Step 12, Section 3.4, Volume 6, the .wms or .wpr is dropped from the file name in the title bar."
- 822 4/17/2009–10/16/2009—Generate IDs button never undims when entering Infiltration after Initial Moisture
- 1535 10/16/2009–10/16/2009—WMS doesn't recognize drainage basin
- 107 10/9/2008–10/16/2009—Incorrect prompt shown
- 1531 10/15/2009–10/15/2009—Rain gage weights don't add up to one
- 666 3/6/2009–10/15/2009—Contour Options dialog spreadsheet problems
- 1488 10/6/2009–10/15/2009—Contour intervals incorrect when using specified values and then switching to color fill
- 358 12/23/2008–10/14/2009—Save button in Project Filename page of Hydrologic Modeling Wizard overwrites existing files
- 106 10/9/2008–10/14/2009—Hourglass not shown while deleting tin verts
- 485 1/22/2009–10/13/2009—Problem when trying to indicate the location of WinHspfLt.exe
- 314 12/12/2008–10/13/2009—Legend text in hydrograph plot for Rational method hydrographs
- 219 11/12/2008–10/13/2009—Edit | Preferences window needs some tweaking
- 789 4/3/2009–10/13/2009—Can't access drainage data display options
- 480 1/21/2009–10/12/2009—Forum feature request: saving depression points
- 140 10/24/2008–10/12/2009—Ghost Outlet Bug: WMS selects an imaginary outlet when selecting an outlet in the Map Module in 8.1
- 451 1/15/2009–10/9/2009—WMS vanishes when working with a project with a deep file directory
- 1472 9/30/2009–10/8/2009—MODRAT import Issue
- 426 1/13/2009–10/7/2009—Saving image registration (world file) with xmdf project file
- 398 1/7/2009–10/7/2009—Saving image registration data with .wms (xmdf) file

- 1470 9/30/2009–10/7/2009—MODRAT Output Error
- 1250 7/31/2009–10/7/2009—WMS crashes when converting a DTED DEM to a TIN and toggling the TIN display on and off
- 836 4/22/2009–10/7/2009—State plane zone description not showing in shared projection dialog
- 862 4/28/2009–10/7/2009—Another problem displaying image with ArcObjects enabled
- 245 11/21/2008–10/7/2009—Cross section not displayed in channel calculator
- 439 1/13/2009–10/7/2009—Duplicating a coverage takes a long time
- 438 1/13/2009–10/7/2009—Changing coverage name in the Coverage Properties dialog takes a long time
- 165 10/30/2008–10/6/2009—Rain Gage coverage should have Map data display attributes
- 168 10/30/2008–10/6/2009—Legend gets stuck on WMS Screen when Selecting File | New
- 433 1/13/2009–10/6/2009—Mapping shapefile to feature obejcts crash
- 361 12/29/2008–10/6/2009—Problem with images on xmswiki.com
- 1457 9/25/2009–10/6/2009—Crash when running OC rational
- 357 12/23/2008–10/6/2009—DEM point selection tool slow selecting a DEM point
- 1471 9/30/2009–10/6/2009—Adding stream arc crashes project
- 315 12/12/2008–10/6/2009—Delete button in the Elevation Discharge Input dialog does not work
- 301 12/9/2008–10/6/2009—Selecting a segment in the CE-QUAL-W2 Polygon Segment Attributes dialog does not select a segment in the graphics window
- 1485 10/5/2009–10/6/2009—WMS vanishes when opening this GSSHA project from the wiki tutorials
- 206 11/11/2008–10/5/2009—Remove Flat triangles does not work in WMS 8.2
- 449 1/15/2009–10/5/2009—TIN and map file do not overlay properly in WMS 8.1
- 179 11/4/2008–10/5/2009—Problem displaying 2D grid cell elevations with color filled contours
- 916 5/7/2009–10/5/2009—Display issue with reading map file
- 116 10/16/2008–10/5/2009—Web Services options that don't work should be dimmed out
- 130 10/21/2008–10/5/2009—Errors when running through CE-QUAL tutorial in WMS 9.0
- 1242 7/30/2009–10/2/2009—NSS will not load without write access to Program Files directory
- 157 10/29/2008–10/2/2009—Pattern palette not working in WMS 8.1
- 101 10/7/2008–10/2/2009—2D Grid Flow Vectors drawn on same layer as Color Filled Contours
- 1473 9/30/2009–10/2/2009—NSS calculation possible error
- 1271 8/6/2009–9/30/2009—WMS crashes when creating a filmloop of a partial solution
- 1594 10/31/2009–11/4/2009—Vol 3, Ch4, Section 4.3.2, Step 2. Strange symbols appear in list box.
- 1578 10/28/2009–11/4/2009—Multiple arcs should only have one attribute if connected

Intermediate Release WMS 8.2.6 – September 22, 2009

An update to WMS 8.2 has been posted on Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted-Updated—Summary

- 170 10/31/2008–9/21/2009—Bug when opening a WMS project from spatial tutorials
- 999 6/3/2009–9/21/2009—Error message occasionally pops up while moving through sediment flux or stream sediment flux dataset.
- 898 5/5/2009–9/21/2009—WMS crashes when running Clean Up Model command on this project
- 860 4/28/2009–9/21/2009—Problem reading TIFF image using File | Open command
- 852 4/27/2009–9/21/2009—Upon completion of the Volume VI - Chapter 6 tutorial, Section 6.2, Step 13 WMS crashes.
- 1391 9/8/2009–9/21/2009—GSSHA datasets don't match up with hydrographs

- 360 12/23/2008–9/18/2009—Default DEM display point size
- 464 1/19/2009–9/18/2009—Switching to Specified Values in Contour Options does not enable Value fields
- 648 3/4/2009–9/18/2009—Remove reservoir display opts from the TIN drainage display tab
- 283 12/6/2008–9/18/2009—Error message, Vol. V, Ch. 1, Section 1.2, step 24, "An unsupported operation was attempted."
- 192 11/7/2008–9/18/2009—Bug when specifying a bias when creating a 2D grid
- 171 10/31/2008–9/18/2009—Tc value not updating in ModClark tutorial
- 105 10/9/2008–9/17/2009—DrawBuffer_enum GetDrawBuffer () Assertion
- 100 10/7/2008–9/17/2009—WMS exports the canvas window background when saving a *.kmz film loop
- 989 6/1/2009–9/17/2009—WMS crashes when running DEM->Stream Arcs command
- 1199 7/20/2009–9/17/2009—WMS does not write out projection file when exporting grids from GSSHA output datasets
- 277 12/4/2008–9/17/2009—Changes to GSSHA .cmt file requested by Chuck
- 1270 8/6/2009–9/11/2009—Embankment arcs should not extend into inactive GSSHA cells or else GSSHA won't run
- 1275 8/10/2009–9/11/2009—WMS does not read in more than 5, or 6 files into the "Convert Grids" dialog at a time.
- 1286 8/12/2009–9/11/2009—In job control dialog,contaminant name and location are lost when you add a new contaminant
- 1340 8/24/2009–9/10/2009—GSSHA sample tutorial crashes WMS
- 1341 8/24/2009–9/10/2009—GSSHA doesn't save *.dep and *.ghm file when you choose to save before running
- 1343 8/24/2009–9/10/2009—Basin data not read from file
- 1389 9/3/2009–9/9/2009—WMS crashes when conforming GSSHA embankment arc to grid if no grid is available
- 1068 6/23/2009–9/2/2009—Hydrograph output toggle in GSSHA solution results window not used, misleading
- 1204 7/21/2009–8/17/2009—WMS Crashes upon completion of Step 12, Section 4.2, in the Volume 6, Chapter 3 tutorial.
- 1170 7/15/2009–8/17/2009—Problem with either GSSHA or WMS-reading a GSSHA solution file
- 1215 7/23/2009–8/11/2009—A dialog reading "There was an error reading record 1 in the Green-Ampt Land Use Table" cannot be dismissed.
- 1051 6/17/2009–8/10/2009—DEM Contours written to KMZ file, when DEM is turned off
- 1173 7/15/2009–8/10/2009—WMS crashes upon clicking the 'Assign' buttton during step 12, section 1.4, of the Volume 5, Chapter 1 – HSPF tutorial.
- 1240 7/30/2009–8/7/2009—Save File command in coverage overlay window should show default file types
- 1241 7/30/2009–8/7/2009—WMS crashes when delineating a watershed before running TOPAZ
- 1138 7/10/2009–8/6/2009—Volume 3, Chapter 2 – Floodplain Delineation, csdb.idx file not found when saving *.wms file at the end.
- 1252 7/31/2009–8/6/2009—Bug when reading in rain gages from format precip macro
- 1188 7/16/2009–8/6/2009—The attached WMS.ini file causes WMS to crash when loading DEMs
- 1002 6/3/2009–7/16/2009—HEC-1 lag time/time of concentration data computed
- 997 6/3/2009–7/15/2009—SSURGO Shape files Bug
- 1000 6/3/2009–7/15/2009—WMS crashes during the creation of an animation using the sediment flux and stream sediment flux data sets.
- 1141 7/10/2009–7/10/2009—Bug in assigning texture using Join SSURGO command.
- 1143 7/10/2009–7/10/2009—In WMS dev, the ksat algorithm in the Join SSURGO data command occasionally returns "-1.#IND00"
- 1144 7/10/2009–7/10/2009—Bug in Joining SSURGO data KSAT values in WMS dev

- 1145 7/10/2009–7/10/2009—resistance equation in Join SSURGO dialog should be changed to "equivalent conductance equation"
- 919 5/7/2009–7/10/2009—WMS disappears when running DEM->Stream Arcs command on this DEM
- 1004 6/4/2009–7/9/2009—GSSHA simulations including erosion and sediment only read in the first sediment from the .sed file into WMS.
- 903 5/6/2009–7/9/2009—Tutorials, Volume V, Chapter 2, Section 2.6.2, Step 2.
- 1066 6/22/2009–7/8/2009—CAD layers do not disable properly in the tree view
- 1007 6/5/2009–7/6/2009—TerraServer generates an error upon completion of step 23, Section 2.7.2, of the Chapter 2, Volume 1 Tutorial.
- 899 5/5/2009–7/1/2009—GSSHA hyetograph ranamed as "Hyetograph" in XY Series Editor
- 909 5/6/2009–6/30/2009—spreadsheet entries in GSSHA Solution Analysis dialog dimmed for no good reason
- 1083 6/26/2009–6/30/2009—GSSHA calibration mode results different when using different units
- 901 5/6/2009–6/30/2009—Tutorials, Volume IV, Chapter 3, Section 3.5 - After finishing the tutorial WMS crashes while saving finished project file.
- 906 5/6/2009–6/29/2009—GSSHA Outlfow hydrograph not matching stream flow graph in solution results window

Intermediate Release WMS 8.2.2 – June 23, 2009

An update to WMS 8.2 has been posted on Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted—Updated-Summary

- 897 5/5/2009–6/15/2009—WMS crashes when deleting the GSSHA cumulative rainfall dataset from the project explorer
- 902 5/6/2009–6/15/2009—MODRAT "Use Input Hydrograph" option doesn't work
- 905 5/6/2009–6/12/2009—XY Series editor "show dates" option prompts for a reference date but doesn't use it
- 908 5/6/2009–6/12/2009—WMS does not read the attached xy series file (even though it was created in WMS)
- 924 5/11/2009–6/5/2009—WMS crashes when running model checker on the attached GSSHA model
- 988 6/1/2009–6/5/2009—WMS crashes when importing these TOPAZ files
- 952 5/18/2009–6/5/2009—WMS Crashes on Check Model command in GSSHA
- 781 4/2/2009–6/5/2009—SURRGO Soils data not transferred to GSSHA index maps
- 782 4/2/2009–6/5/2009—No units shown for joined SURRGO soils data
- 896 5/5/2009–6/4/2009—infinite loop while trying to read prj file instead group project file
- 998 6/3/2009–6/4/2009vRead project file crash
- 808 4/10/2009–5/15/2009—GSSHA stops running after running the second hour of the attached 5-hour simulation
- 849 4/24/2009–5/15/2009—WMS crashes when importing this GIS (.geo) file
- 853 4/27/2009–5/14/2009—Problem reading or writing scalar data to or from 2D scattered data
- 926 5/11/2009–5/13/2009—GSSHA crashes when trying to run the attached model
- 775 4/2/2009–5/7/2009—GSSHA crashes when using lakes
- 786 4/2/2009–5/6/2009—Prompt needed for users to switch model units from feet to meters if using GSSHA
- 586 2/14/2009–5/6/2009—Title in the WMS 8.2 title bar needs to be cleaned up.
- 309 12/11/2008–5/5/2009—GSSHA does not recognize wetlands (*.wlm) file when spaces are in the file path

Intermediate Release WMS 8.2.1 – May 4, 2009

An update to WMS 8.2 has been posted on Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted—Updated-Summary

- 659 3/5/2009–5/1/2009—WMS crashes when running MODRAT with no file prefix
- 698 3/16/2009–5/1/2009—MODRAT Error
- 783 4/2/2009–5/1/2009—No feedback for MODRAT run – Randall Greenwood
- 511 1/29/2009–5/1/2009—Deleted 1D-Hyd Cross-Section Arcs reappear when reopening WMS project
- 200 11/10/2008–4/30/2009—WMS crashes when opening CAD file with image file (*.sid) loaded
- 758 3/27/2009–4/29/2009—Better reading and processing needed for NLCD 2001 Land Use grids
- 700 3/16/2009–4/28/2009—Problem with tying stream arcs to sub-basins
- 633 2/27/2009–4/28/2009—WMS RASTER import Crash
- 757 3/27/2009–4/28/2009—WMS crashes when importing an ArcGrid with ArcObjects enabled in WMS 8.1
- 778 4/2/2009–4/27/2009—Selecting Redistribute Vertices command without a selected arc launches an unnecessary dialog
- 671 3/10/2009–4/27/2009—2-D Scatter Dataset Not Created
- 665 3/6/2009–4/27/2009—Problem with saving/reading datasets associated with TINs
- 662 3/5/2009–4/23/2009—WMS 8.2 Crashes upon completion of Section 3.5, step 6 of the Volume IV, Chapter 3 tutorial.
- 667 3/7/2009–4/23/2009—WMS 8.1 Disappears upon completion of step 9 of GSSHA Tutorials:3 Fixing Digital Dams
- 436 1/13/2009–4/23/2009—WMS loses basin data reading in a project file
- 432 1/13/2009–4/22/2009—Problems with coverage att tables when closing WMS
- 702 3/16/2009–4/21/2009—Need input value length restrictions on MODRAT input values in Edit Parameters window
- 660 3/5/2009–4/21/2009—WMS crashes when using Flood menu options with nothing loaded
- 741 3/24/2009–4/20/2009—WMS 8.2 Crashes when reading in NSSmap_FL.map, step 2, section 7.4.1 of the Volume II, Chapter 7 tutorial. NSS Interface.
- 717 3/18/2009–4/20/2009—Compute Basin Data Crash!
- 110 10/9/2008–4/20/2009—Crash when going to global mapper dialog
- 817 4/16/2009–4/16/2009—Opening this project file causes WMS to crash
- 608 2/24/2009–4/14/2009—File path to LA County soils table needs to be updated in WMS 8.2 Preferences
- 780 4/2/2009–4/14/2009—Problem when using hyetograph precip option with GSSHA
- 764 3/30/2009–4/13/2009—Can't create filmloops
- 610 2/25/2009–4/13/2009—WMS disppears when trying to open an HY-8 culvert project
- 755 3/26/2009–4/13/2009—Problem converting coordinates when you download shapefiles with defined projection from web services
- 784 4/2/2009–4/10/2009—Misleading text on button in GSSHA Maps dialog
- 295 12/9/2008–4/10/2009—GSSHA 4.0 crashes in Tutorial 6.4
- 704 3/16/2009–4/10/2009—WMS 8.2 Web services dialog does not appear if not connected to the internet
- 703 3/16/2009–4/10/2009—Web service problem when downloading Louisiana data (and possibly data from other states)
- 751 3/25/2009–4/10/2009—The option to convert two shapefiles of the same type to a single coverage is not available
- 701 3/16/2009–4/7/2009—Problem with default Z units when converting NED DEM

- 746 3/25/2009–4/7/2009—Problem converting coordinates with shapefile
- 747 3/25/2009–4/7/2009—Problem converting coordinates with land use shapefile (no projection file)
- 748 3/25/2009–4/7/2009—Problem with vertical units when reading a DEM file
- 749 3/25/2009–4/6/2009—Problem setting and converting coordinates on a shapefile with a projection file
- 750 3/25/2009–4/6/2009—Problem converting shapefiles with projections in the hydrologic modeling wizard
- 692 3/12/2009–4/3/2009—Embankment Arc Profile Editor does not seem to update the endpoint elevations in the GSSHA-7 Tutorial.
- 189 11/7/2008–4/3/2009—WMS crashes in Grid Ooptions dialog (display x-y axis)
- 697 3/16/2009–4/2/2009—DEMEdit tutorial files not included in WMS 8.1 installation
- 616 2/25/2009–4/2/2009—WMS window goes blank when deleting drainage data with Select Feature Objects tool
- 448 1/15/2009–4/2/2009—Crash when opening *.map file on a TIN in WMS 8.1 and 8.2
- 522 2/3/2009–4/1/2009—Color fill contours not displayed for flood delineation
- 538 2/6/2009–3/31/2009—Delineate Basins Wizard stream display threshold units
- 337 12/18/2008–3/31/2009—Update Delineate Basins Wizard
- 507 1/29/2009–3/30/2009—Duplicate arcs being generated from 1D-Hyd Cross-Section coverage when exporting GIS file
- 477 1/21/2009–3/27/2009—River Hydraulic Schematic should be deleted when XSection and Centerline coverages get deleted
- 499 1/28/2009–3/27/2009—WMS not responding when accessing NED Data through Web Services if WebServices.exe is missing
- 508 1/29/2009–3/26/2009—Allow the option to download and read SSURGO data files for multiple SSURGO soil shapefiles
- 510 1/29/2009–3/25/2009—Allow users to select more than one soil type and/or land use shapefile for the Define Land Use and Soil Data step in the HMW
- 611 2/25/2009–3/23/2009—Web Services reports two errors when trying to save to an unexisting directory
- 547 2/6/2009–3/23/2009—Verify GSSHA embankment arc conceptualization
- 655 3/4/2009–3/19/2009—Land use and soil shapefile options not saved
- 656 3/4/2009–3/19/2009—Selected web catalog web service options (land use and soil) are not saved in the hydrologic modeling wizard
- 657 3/4/2009–3/19/2009—Terraserver web service resolutions are not used in the hydrologic modeling wizard
- 474 1/21/2009–3/19/2009—Save prompt is supressed after saving as .dwg, .dxf, .shp, or any other file type
- 658 3/4/2009–3/18/2009—Better handling of web service file names in the hydrologic modeling wizard
- 634 2/27/2009–3/18/2009—CE-QUAL-W2 bathymetry bug
- 303 12/10/2008–3/16/2009—Problem color filling contours
- 497 1/27/2009–3/12/2009—WMS crashes when re-entering the HY-8 Culvert dialog
- 614 2/25/2009–3/10/2009—WMS crashes when deleting a MODRAT diversion from a basin in WMS 8.1 and 8.2
- 532 2/5/2009–3/10/2009—Bug when setting bank arc attribute
- 641 2/27/2009–3/10/2009—Change default output in MODRAT Edit Parameters window
- 654 3/4/2009–3/10/2009—Problem creating uniform index map in WMS 8.1
- 650 3/4/2009–3/9/2009—DEM properties dialog
- 584 2/13/2009–3/9/2009—Next Hydrograph Station-> button in HEC-1 Parameters dialog causes WMS to crash
- 615 2/25/2009–3/9/2009—Add Retrieve Diversion command to right click menu in MODRAT
- 653 3/4/2009–3/9/2009—TR-20 CN entry

Release WMS 8.2.0 – April 16, 2009

WMS 8.2 has been released. This is the final release version. Bug fixes to WMS 8.2 will be posted in future updates.

Intermediate release WMS 8.2.1 (beta) – March 9, 2009

An update to WMS 8.2 Beta has been posted to the xmswiki.com web site. The main purpose of this update was to fix a problem with the WMS 8.2 beta version not working with WMS 8.1 hardware locks. The best place to obtain the update is from the WMS 8.2 beta download page on the xmswiki.com web site. The following known bugs have been fixed in this version:

Bug Id-Date Submitted-Updated-Summary

- None 12/6/2009–12/6/2009—Fixed a problem with WMS 8.2 not working with WMS 8.1 hardware locks
- 272 12/2/2008–3/6/2009—WMS writes out MODRAT basin numbers incorrectly
- 465 1/19/2009–3/6/2009—WMS crashes when color fill contouring one contour
- 649 3/4/2009–3/5/2009—WMS crashes when converted TIN to DEM
- 548 2/6/2009–3/5/2009—Crash converting TIN to DEM without triangulating TIN
- 645 3/3/2009–3/5/2009—WMS 8.1 Bug – Saving as *.sup or *.shp file

Intermediate release WMS 8.1.6 – March 4, 2009

An update to WMS 8.1 has been posted to Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted—Updated-Summary

- 737 12/8/2008–3/4/2009—Base Flow Parameters in HEC-1 not written properly to HEC-HMS
- 341 12/19/2008–3/2/2009—TIN triangulation and display bugs in WMS 8.1 and WMS 8.2
- 435 1/13/2009–2/26/2009—SWMM hydraulic schematic data not written to *.wpr or *.wms (xmdf) project files
- 498 1/27/2009–2/26/2009—Crash when reading project file into WMS 8.2
- 431 1/13/2009–2/25/2009—SMPDBK parameters not saved to *.wms (xmdf) project file
- 282 12/5/2008–2/23/2009—Convert DEM to TIN crash
- 147 10/27/2008–2/19/2009—"DEM Contours, Color Filled Polygons, and DEM Streams share same draw space in WMS 8.1"
- 761 12/8/2008–2/19/2009—Reading the attached DEM files causes WMS to crash
- 294 12/9/2008–2/18/2009—Film Loop issues in Tutorial 6.3 and 6.4
- 762 12/8/2008–2/18/2009—WMS has problems reading the attached BIL files
- 553 2/9/2009–2/17/2009—Flood depth coverage doesn't accept user defined depth intervals
- 234 11/18/2008–2/16/2009—"WMS disappears when opening the project file at the beginning of the Volume VI, Chapter 4 tutorial."
- 261 11/26/2008–2/16/2009—"WMS 8.1 disappears upon completion of step 10, of the 5th tutorial on the gsshawiki."
- 262 11/26/2008–2/16/2009—WMS 8.1 Disappears upon completion of step 3 of Tutorials:7 Break-point Cross Sections on the GSSHA wiki.
- 308 12/11/2008–2/16/2009—Error Displaying Shapefiles
- 336 12/17/2008–2/16/2009—"WMS 8.2 Vanishes upon completion of step 3, section 2.3.6, of the Volume 2, Chapter 2 tutorial (Editing DEMs)."
- 111 10/11/2008–2/12/2009—"WMS kills itself upon completion of the step 2, section 2.6.1, of tutorial vol 3, chapter 2."

- 176 11/4/2008–2/12/2009—"On the WMS Testing machine WMS 8.1 disappears upon completion of tutorial Volume V, Chapter 2, Section 2.5.2, Step 1"
- 293 12/9/2008–2/12/2009—"WMS 8.1 crashes upon completion of step 21, section 2.4.2, in the MODRAT Chapter 2, MODRAT GIS Tutorial"
- 344 12/19/2008–2/12/2009—"XMDF (*.wms) file does not load (or possibly save) multiple datasets for 2D Grid, or single dataset for 2D Scatter"
- 874 12/8/2008–2/11/2009—"Tutorials, Volume I, Chapter 5, Section 5.8.3.2, Step 7. Crash"
- 875 12/8/2008–2/11/2009—"Tutorials, Volume I, Chapter 6, Section 6.2.4, Step 10. WMS Kills itself upon completion of this step."
- 430 1/13/2009–2/10/2009—TIN data and xmdf project file
- 437 12/8/2008–2/10/2009—Display Order andy nowak
- 659 12/8/2008–2/10/2009—Orange County testing results
- 734 12/8/2008–2/10/2009—Drawing order bug with drawing tools in 8.0 and 8.1
- 746 12/8/2008–2/10/2009—The Shuffle (display order) tools for drawing objects do not work
- 552 2/9/2009–2/9/2009—MODRAT 2.0 output (.out) problems
- 559 2/9/2009–2/9/2009—MODRAT 2.0 does not run when using input hydrograph files
- 545 2/6/2009–2/6/2009—Hydraulic structures entered at GSSHA nodes not saved
- 531 2/5/2009–2/5/2009—GSSHA retention depth option not working
- 527 2/4/2009–2/4/2009—Problems with the soil file running MODRAT 1.0
- 310 12/11/2008–1/30/2009—Change the radio buttons in the Watershed Data step of the hydrologic modeling wizard from radio buttons to toggle boxes
- 446 1/14/2009–1/30/2009—Problem saving wms.ini file from the hydrologic modeling wizard
- 340 12/18/2008–1/28/2009—WMS will not open gdm files with old coord sys
- 496 1/27/2009–1/27/2009—Crash deleting a GSSHA coverage
- 429 1/13/2009–1/26/2009—Reading in options set in the HEC-HMS Meteorologic Parameters
- 441 1/13/2009–1/26/2009—Extra default (elev) data set with ModClark model and *.wms (xmdf) project file
- 493 1/26/2009–1/26/2009—Cropping collars on *.tif images not working properly
- 296 12/9/2008–1/22/2009—WMS crashes when creating soil type coverage from the Hydrologic Modeling Wizard
- 440 1/13/2009–1/22/2009—HEC-HMS ModClark transform option not saved with *.wpr or *.wms (xmdf) project files
- 483 1/22/2009–1/22/2009—Hydrologic Modeling Wizard skips around
- 258 11/25/2008–1/19/2009—We don't read 2D grid datasets when saving/reading the grid as an XMDF file-see description
- 371 12/8/2008–1/19/2009—Help file not working for some help buttons
- 743 12/8/2008–1/19/2009—Save GSSHA Project File resets display options
- 183 11/6/2008–1/16/2009—WMS crash in Grid Options dialog
- 184 11/6/2008–1/16/2009—WMS crashes in Grid Options dialog (Displaying Grid Points)
- 359 12/23/2008–1/16/2009—Adverse slopes displayed on all coverages that support stream arcs
- 416 1/9/2009–1/16/2009—Crash when creating an index map before initializing GSSHA
- 428 1/13/2009–1/16/2009—WMS displays multiple hydrographs in the same window with the same color
- 278 12/4/2008–1/15/2009—Using radar rainfall data for long-term gridded simulation
- 241 11/20/2008–1/9/2009—WMS crashes (disappears) when using select feature line branch tool
- 228 11/14/2008–1/8/2009—WMS 8.1 crashes without warning when snapping the upstream nodes of two streams together
- 155 10/28/2008–1/6/2009—Fill below and Fill above options are backwards in WMS 8.1

- 233 11/18/2008–1/2/2009—"WMS 8.1 Disappears upon completion of the last step of section 2.13.1 of the Volume VI, Chapter 2 tutorial."
- 112 10/13/2008–12/30/2008—"Extra edit field for ""Round culverts"" in GSSHA"
- 142 10/24/2008–12/18/2008—Problem displaying filmloops
- 242 11/20/2008–12/18/2008—WMS crashes in Cross-Section Attributes dialog
- 217 11/12/2008–12/17/2008—Add tmcontours.dwg and trailmountain.dem to the Tins directory of tutorials for WMS 8.2
- 229 11/14/2008–12/17/2008—Add the lc.wpr file on Chris' computer and all its associated files to the HSPF tutorial directory and delete littlecotton.wpr
- 321 12/15/2008–12/17/2008—Displaying properties of selections in the Properties Window
- 322 12/15/2008–12/17/2008—Start time for link/node datasets in GSSHA models with precip gages not correct
- 181 11/4/2008–12/16/2008—We need a default max conveyance depth value for GSSHA cross section channels in WMS 9.0
- 317 12/12/2008–12/16/2008—Unitialized memory crash
- 233 12/8/2008–12/15/2008—Crash when updating contours
- 175 11/4/2008–12/12/2008—WMS Crashes after confirming the overwrite of a storm drain file.
- 191 11/7/2008–12/12/2008—WMS crashes when viewing wetland cells in 2D Grid Display Options
- 269 12/2/2008–12/12/2008—"Users should set horizontal and vertical units to meters at the beggining of the Volume 5, Chapter 1, HSPF Tutorial."
- 274 12/3/2008–12/12/2008—WMS crashes when trying to assign detention basin parameters
- 195 11/10/2008–12/11/2008—Help About Window needs to be updated
- 161 10/29/2008–12/10/2008—WMS crashes at the end of the CE-QUAL-W2 Tutorial in WMS 8.1
- 230 11/17/2008–12/10/2008—Bug in WMS 8.2 when right-clicking on a spreadsheet cell and selecting another cell
- 213 11/12/2008–12/9/2008—"The transform command is in the TIN menu of the data module, but is not in the right-click menu for a TIN (8.1 and 8.2)"
- 156 10/28/2008–12/8/2008—Measure tool disappears if snapped to a vertex or node in WMS 8.1
- 232 11/17/2008–12/8/2008—Measure tool cannot trace arcs
- 289 12/8/2008–12/8/2008—DEM color-filled contours with flow distance contours toggled on causes crash
- 290 12/8/2008–12/8/2008—Basin labels displayed with excess space between parameters
- 834 12/8/2008–12/8/2008—GSSHA Bug List from tutorials

Intermediate release WMS 8.1.4 – December 8, 2008

An update to WMS 8.1 has been posted to Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted—Updated-Summary

- 270 12/2/2008–12/4/2008—Delineate Crash
- 791 11/4/2008–12/4/2008—The Display Order options on the Display Options window don't work
- 134 10/23/2008–12/3/2008—Define Land Use and Soil Data step in Wizard not working with GSSHA Model Coverage in 9.0
- 273 12/2/2008–12/3/2008—File IO problem when assigning reservoirs to MODRAT diversions
- 102 10/7/2008–12/2/2008—WMS crashes when color fill contouring a 2D grid and then dragging the WMS window to my other monitor
- 118 10/17/2008–12/2/2008—Problem reading DEMs
- 115 10/16/2008–12/2/2008—Contouring issue – all contour appear the same color (red)

- 131 10/22/2008–12/2/2008—Saving shapefile *.prj files
- 136 10/23/2008–12/1/2008—WMS 9.0 crashes when clicking "Compute Index Mapping Tables" in the Hydrologic Modeling Wizard
- 4 7/9/2008–12/1/2008—Installation issues with copy protection and new demo password generation.
- 194 11/7/2008–11/25/2008—change xmdf file extension from *.wpr to *.wms
- 152 10/28/2008–11/20/2008—Computing rational method hydrographs triggers a bunch of graphics asserts in WMS 8.1
- 174 11/4/2008–11/18/2008—WMS disappears in the HSPF tutorial (Vol V, Ch 1) when reading in littlecotton.wpr
- 218 11/12/2008–11/17/2008—DEM display shifts when creating feature arcs
- 877 11/4/2008–11/17/2008—WMS kills itself upon completion of Tutorial 2-2, section 2.2.3, step 7 or 8.
- 38 8/7/2008–11/14/2008—Can't edit culvert parameters – Zack Young
- 146 10/27/2008–11/14/2008—WMS 8.1 crashes when placing interior outlets
- 135 10/23/2008–11/14/2008—When opening a GSSHA project in WMS 9.0, the wizard should set the current model to be GSSHA
- 145 10/27/2008–11/14/2008—TIN display bug in WMS 8.1 – entire tin shifts when closing a feature arc
- 150 10/28/2008–11/14/2008—WMS 8.1 crashes when adding subbasins and re-delineating after having computed a CN
- 870 11/4/2008–11/14/2008—GSSHA Rain Gage File I/O Bug
- 190 11/7/2008–11/13/2008—DEM contouring on screen is reversed (canyons are shown as ridges and ridges as canyons)
- 224 11/13/2008–11/13/2008—Polygon and DEM point selection does not update correctly in graphics window
- 182 11/6/2008–11/13/2008—Canceling out of TOPAZ Units dialog still writes out a TOPAZ elevation file
- 123 10/21/2008–11/13/2008—Embankment Arc Profile Editor has several bugs (at least 9) in 9.0
- 851 11/4/2008–11/13/2008—WMS Crashes when trying to create shapefiles from these coverages
- 857 11/4/2008–11/13/2008—Embankment dialog does not change arc elevations when creating a vertical curve
- 692 11/4/2008–11/12/2008—HMS | Compute Grid Parameters does not seem to work for computing SMA grid parameters
- 196 11/10/2008–11/10/2008—Cannot define GSSHA link break nodes with erodible cross section stream arcs

Intermediate release WMS 8.1.3 – November 4, 2008

An update to WMS 8.1 has been posted to Aquaveo's web site. The best place to obtain the update is from the download link on Aquaveo's web site at http://www.aquaveo.com/ [3]. The following known bugs have been fixed in this version:

Bug Id Date Submitted—Updated-Summary

- 173 10/31/2008–11/4/2008—WMS 8.1 Crashes upon completion of step 20, section 4.5.1 of the Volume II, Chapter 4 Tutorial. (HEC-HMS Interface)
- 144 10/24/2008–10/31/2008—WMS 8.1 soil legend is squished
- 167 10/30/2008–10/30/2008—Crash saving coordinate system metadata with DEM
- 113 10/14/2008–10/23/2008—WMS disappears upon double clicking on the flood.wpr file in the "open" file dialog.
- 133 10/23/2008–10/23/2008—When moving basin labels the heads of the arrows are quite large on my computer (The WMS testing machine).
- 114 10/14/2008–10/23/2008—Upon Opening the run1.wpr file in Volume 3, Chapter 3 Tutorial WMS Crashes.
- 637 10/31/2008–10/31/2008—Error upon saving file from the Volume 1 Chapter 3 Tutorial.
- 876 10/30/2008–10/30/2008—WMS kills itself upon completion of Tutorial 2-1, section 1.3.5, step 7.

- 345 10/30/2008–10/30/2008—Observation coverage does nothing
- 860 10/30/2008–10/30/2008—Problems editing/saving/writing vertical curve weir data from a hydraulic structure point
- 837 10/29/2008–10/29/2008—CE-QUAL-W2 issues
- 283 10/29/2008–10/29/2008—Georeferencing is wrong for images saved with collar cropped
- 816 10/28/2008–10/28/2008—WMS 8.1 Freezes in the Tc Wizard when rainfall depth is not entered.
- 817 10/28/2008–10/28/2008—Hydrographs not visible near end of MODRAT-3 GIS Tutorial.
- 829 10/28/2008–10/28/2008—WMS 8.1 gives a lot of asserts when removing flat triangles on TINs
- 315 10/28/2008–10/28/2008—Saving Basin Only Projects
- 359 10/28/2008–10/28/2008—WMS can't find HSPF LT executable
- 855 10/28/2008–10/28/2008—Problem with clearing selections when selecting an item in the project explorer
- 826 10/27/2008–10/27/2008—Bug in setting min and max contour intervals for Cumulative Rainfall Data
- 868 10/27/2008–10/27/2008—Copy and paste error within WMS spreadsheets when commas are used
- 849 10/27/2008–10/27/2008—Crash after importing TIN file (RELEASE VERSION ONLY)
- 858 10/24/2008–10/24/2008—Swap edges tool is MESSED UP
- 383 10/24/2008–10/24/2008—Cross Section computational stations remain at 0.0 when extracting Cross sections for SMPDBK
- 862 10/24/2008–10/24/2008—Interpolation method other than IDW causes crash
- 832 10/23/2008–10/23/2008—TR-55 spreadsheet labels are always dimmed
- 836 10/23/2008–10/23/2008—Can't cancel out of contour options if min countour value is greater than max value
- 838 10/23/2008–10/23/2008—Bug In CE-QUAL-W2 Tutorial block: Measure tool in segments dialog resets length to zero
- 830 10/23/2008–10/23/2008—Building polygons around streams assigns polygons the wrong type
- 137 10/23/2008–10/23/2008—Right-click option in spreadsheets
- 145 10/22/2008–10/22/2008—TR-20 Reservoir Data
- 77 10/22/2008–10/22/2008—Displaying arc after reading in TIN
- 764 10/21/2008–10/21/2008—Vista Problem: wmsnff.exe crashes in Windows Vista
- 871 10/21/2008–10/21/2008—Can we make the model wrapper resizable?
- 872 10/21/2008–10/21/2008—Outflow Sedograph Plot Window bug
- 869 10/20/2008–10/20/2008—GSSHA Soil Erosion dialog needs units
- 562 10/20/2008–10/20/2008—Error appears when saving HEC-1 file with average precip defined
- 870 10/20/2008–10/20/2008—GSSHA Rain Gage File I/O Bug
- 854 10/17/2008–10/17/2008—GSSHA Multi-gage interpolation methods remain dimmed after importing a Multi-gage file
- 807 10/17/2008–10/17/2008—Error writing curve number dataset when writing HEC-HMS file
- 821 10/16/2008–10/16/2008—Static Text String for file path in Convert Grids dialog is too short
- 840 10/16/2008–10/16/2008—Units for Wetland polygons do not match units reported on the gsshawiki
- 811 10/16/2008–10/16/2008—Map Drawing Fonts will crash if non-standard fonts used
- 740 10/15/2008–10/15/2008—Hydrographs not shown in the Volume 2 - Chapter 4 (HEC-1) Tutorial
- 172 10/15/2008–10/15/2008—Tree right click menus bug, with "Change module when tree selection changes" off
- 206 10/15/2008–10/15/2008—DEM delineation and redistributing arcs
- 822 10/15/2008–10/15/2008—Bug in Contour Options dialog
- 772 10/14/2008–10/14/2008—Film loops do not work
- 815 10/14/2008–10/14/2008—Map Attributes Dialog Crash
- 825 10/14/2008–10/14/2008—Incremental Rainfall dataset shows the wrong dates and times in the properties window
- 828 10/14/2008–10/14/2008—Bug when viewing values from Hydrograph Plots

- 814 10/13/2008–10/13/2008—WMS crashes when double clicking on GSSHA embankment arc "low points"
- 865 10/13/2008–10/13/2008—Problem reading HEC-RAS Solution, crash when reading WMS 8.1 project file into WMS 9.0
- 820 10/13/2008–10/13/2008—Bugs in Volume 2, Tutorial 7 of GSSHA Training course
- 873 10/13/2008–10/13/2008—Virtual Earth gives you errors
- 824 10/10/2008–10/10/2008—Bug in Convert Grids dialog
- 827 10/10/2008–10/10/2008—Error when Importing Grids for GSSHA : "Error Reading Scalar Binary File"
- 859 10/10/2008–10/10/2008—Crash when selecting gssha hydraulic structures
- 852 10/10/2008–10/10/2008—GSSHA bug: GW_INIT_MOISTURE card not needed, not recognized by GSSHA
- 856 10/10/2008–10/10/2008—Web Services Dialog - Tutorials Vol II, Ch 1, Section 1.3.3
- 846 10/9/2008–10/9/2008—Problem displaying adverse slopes in GSSHA coverage
- 864 10/9/2008–10/9/2008—GSSHA Stochastic run dialog crashes after reading GSSHA project file
- 351 10/6/2008–10/6/2008—Tutorial files for tutorial 3 (basic feature objects)
- 354 10/3/2008–10/3/2008—Typo in new tutorials

Intermediate release WMS 8.1.1 – October 2, 2008

Note about this update: Because NFF [4] has been replaced by NSS in WMS 8.1, you may need to uninstall NFF from your computer before installing the update to WMS 8.1. It may also be a good idea to install NSS on your computer, though this should not be necessary. After installing this update, if you have trouble running NSS in WMS 8.1, make sure your NSSv4.mdb database file in the WMS 8.1 directory exists and is not set to Read-Only. If you continue to have problems, download and install NSS from this location [5].

- The NFF (National Flood Frequency) database has been converted to the newer NSS (National Streamflow Statistics) database in WMS 8.1.
- A problem running GSSHA in "batch" mode has been fixed.
- A problem in web services with converting data to anything other than NAD 83 has been fixed.
- "File Import Wizard – Step 1 of 2" Dialog freezes upon clicking next in the Vol 1 Ch 5 tutorial has been fixed.
- Arc Selection Problem – Tutorials – Volume 1, Chapter 6, Section 6.2.3, step4 has been fixed.
- Tutorials, Volume II, Chapter 1, Section 1.3, step 4 – mixed units in "Units" dialog has been fixed.
- Terra Server creates an error in the Volume III, Chapter 1, Tutorial has been fixed.
- Tutorials, Volume III, Chapter 3, Section 3.4.2, Step 6 – Z Scale Combo Box inactive has been fixed.
- Penman Evapotranspiration method parameters not displaying in Mapping Tables (GSSHA) has been fixed.

Related Topics

- What's new in WMS 9.0
- What's new in WMS 8.4
- What's new in WMS 8.3
- What's new in WMS 8.2
- What's New in WMS 8.1
- System Requirements
- Graphics Card Troubleshooting

References

[1] http://www.aquaveo.com
[2] http://www.aquaveo.com/wms-pricing
[3] http://www.aquaveo.com/downloads?tab=3#TabbedPanels
[4] http://water.usgs.gov/software/NFF/
[5] http://water.usgs.gov/software/NSS/

WMS Basic Modeling Concepts

WMS is used primarily to set up and run hydrologic models. Though the software has expanded to provide additional hydraulic and hydrologic tools that engineers will find useful, the original focus of the software remains the same. This section discusses the fundamental concepts upon which WMS is built.

The distinguishing difference between WMS and other similar applications is its ability to manipulate digital terrain data for hydrologic model development within a GIS-based environment. WMS uses three primary data sources for model development:

1. Geographic Information Systems (GIS) Vector Data
2. Digital Elevation Models (DEMs) or Gridded Elevation Sets
3. Triangulated Irregular Networks (TINs)

WMS Downloads

Executable and Library Downloads

- Download WMS 9.0 Beta Installation (Full Version) [1] (64 Bit Full Version) [2]
- Download WMS 8.4 Installation (Full Version) [3] (64 Bit Full Version) [4]
- Download WMS 8.3 Installation (Full Version) [5]
- Download WMS 8.2 Installation (Full Version) [6]

General WMS

Category	Tutorial (PDF)	Required Files
Introduction	• Introduction to WMS [7] • Images [8] • Basic Feature Objects [9] • Advanced Feature Objects [10]	• [11] • [12] • [13] • [14]
Editing Elevations	• DEM Basics [15] • Using TINs [16]	• [17] • [18]
Watershed Modeling	• DEM Delineation [19] • Advanced DEM Delineation Techniques [20] • Time of Concentration Calculations and Computing a Composite CN [21]	• [22] • [11] • [23], [24]

Hydrologic Models

Category	Tutorial (PDF)	Required Files
HEC-1	• HEC-1 Interface [25]	• [26]
HEC-HMS	• HEC-HMS Interface [27]	
Rational Method	• Rational Method Interface [28]	• [29]
NSS (National Streamflow Statistics)	• National Streamflow Statistics Program (NSS) Interface [30]	• [23]
Maricopa County (Phoenix, AZ, USA) Hydrology	• Maricopa County: NSS and HEC-1 [31] • Maricopa County: Master Plan – Creating a Predictive HEC-1 Model [32]	• [33]
MODRAT	• MODRAT Interface – Schematic [34] • MODRAT Interface (GIS-based) [35] • MODRAT Interface (Map-based) [36]	• [37]
Orange County Rational and Unit Hydrograph Methods	• Orange County Rational Method [38] • Orange County Unit Hydrograph [39] • Orange County Small Area Hydrograph [40] • Orange County Hydrology – Using GIS Data [41] • Orange County Rational Method – GIS [42] • Orange County Unit Hydrograph – GIS [43]	• [44]
Spatial Hydrology-HMS ModClark and GSSHA	• HEC-HMS Distributed Parameter Modeling with the MODClark Transform [45] • Developing a GSSHA Model using the Hydrologic Modeling Wizard [46] • Using NEXRAD Rainfall Data in an HEC-HMS (MODClark) Model [47] • Using NEXRAD Rainfall Data in GSSHA [48]	• [49]

Distributed Hydrologic Modeling using GSSHA

Category	Tutorial (PDF)	Required Files
WMS Basics	• Loading DEMs, Contour Options, Images, and Coordinate Systems [50] • Watershed Delineation using DEMs and 2D Grid Generation [51] • Creating Feature Objects and Mapping their Attributes to the 2D Grid [52]	• Personal [53], GSSHA Images [54] • Personal [53], Raw Data [55] • Personal [53], GSSHA Tables [56], Raw Data [55], Watershed Delineation Files [57]
GSSHA Modeling	• GSSHA Initial Overland Flow Model Setup [58] • Correcting Overland Flow Problems [59] • Infiltration [60] • Stream Flow [61] • Developing a GSSHA Model Using the Hydrologic Modeling Wizard in WMS [62] • Post-Processing and Visualization of GSSHA Model Results [63]	• Personal [53], Raw Data [55], GSSHA Basic Model [64] • Personal [53], GSSHA Tables [56], GSSHA Digital Dams [65] • Personal [53], Raw Data [55], GSSHA Tables [56], GSSHA Digital Dams [65], GSSHA Infiltration [66] • Personal [53], GSSHA Tables [56], GSSHA Infiltration [66] • Personal [53], GSSHA Tables [56], Raw Data [55] • Personal [53], Raw Data [55], GSSHA Visualization [67]

Applications	• Precipitation Methods in GSSHA [68] • Analyzing the Effects of Land Use Change (Part - I) [69] • Analyzing the Effects of Land Use Change (Part - II) [70] • Long Term Simulations in GSSHA [71] • Simulating Sediment Transport [72] • Simulating Constituent Transport [73] • Overland Flow Boundary Conditions in GSSHA [74]	• Personal [53], GSSHA Tables [56], Raw Data [55], GSSHA Precipitation [75] • Personal [53], GSSHA Tables [56], GSSHA Scenarios [76] • Personal [53], GSSHA Tables [56], GSSHA Scenarios [76] • Personal [53], Raw Data [55], GSSHA Long Term [77] • Personal [53], GSSHA Sediment [78] • Personal [53], GSSHA Contaminants [79] • Personal [53], GSSHA Boundary Conditions [80]
Calibration	• Manual Calibration of GSSHA models [81] • Stochastic Simulations of GSSHA models [82] • Automated Calibration of GSSHA models [83]	• Personal [53], GSSHA Calibration [84] • Personal [53], GSSHA Calibration [84] • Personal [53], GSSHA Calibration [84]
Groundwater Modeling	• Groundwater Modeling in GSSHA [85] • Advanced Groundwater Modeling in GSSHA [86] • Subsurface Tile and Storm Drains [87]	• Personal [53], Raw Data [55], GSSHA Groundwater [88] • Personal [53], GSSHA Groundwater [88] • Personal [53], GSSHA Subsurface [89]

Hydraulics and Floodplain Modeling

Category	Tutorial (PDF)	Required Files
HEC-RAS	• HEC-RAS Analysis [90] • HEC-RAS – Managing Cross Sections [91]	• [92] • [93]
Floodplain Delineation	• Floodplain Delineation [94] • Stochastic Modeling Using HEC-1 and HEC-RAS [95]	• [96] • [97]
SMPDBK (Simplified Dam Break)	• Simplified Dam Break [98]	• [99]
HY-8 Culvert Modeling Wizard	• HY-8 Modeling Wizard [100]	• [101]
Modeling with FHWA's Hydraulic Toolbox	• Modeling with the Hydraulic Toolbox [102]	• [103]

Storm Drain Modeling

Category	Tutorial (PDF)	Required Files
EPA-SWMM and xpswmm	• SWMM Modeling [104]	• [105]
FHWA Storm Drain	• Storm Drain: Rational Design [106] • Storm Drain: Hydrographic Design [107]	
FHWA HY-12	• HY-12: Rational Design [108]	• [109]

Water Quality Modeling

Category	Tutorial (PDF)	Required Files
HSPF	• HSPF Interface [110]	• [111]
CE-QUAL-W2	• CE-QUAL-W2 Interface [112]	• [113]

Reference Papers

Main WMS page

References

[1] http://www.aquaveo.com/f.php5?s=wms&v=9.0&p=full
[2] http://www.aquaveo.com/f.php5?s=wms&v=9.0&p=full64Bit
[3] http://www.aquaveo.com/f.php5?s=wms&v=8.4&p=full
[4] http://www.aquaveo.com/f.php5?s=wms&v=8.4&p=full64Bit
[5] http://www.aquaveo.com/f.php5?s=wms&v=8.3&p=full
[6] http://www.aquaveo.com/f.php5?s=wms&v=8.2&p=full
[7] http://wmstutorials-9.0.aquaveo.com/1%20Introduction-Intro.pdf
[8] http://wmstutorials-9.0.aquaveo.com/2%20Introduction-Images.pdf
[9] http://wmstutorials-9.0.aquaveo.com/3%20Introduction-BasicFeatureObjects.pdf
[10] http://wmstutorials-9.0.aquaveo.com/6%20Introduction-AdvancedFeatureObjects.pdf
[11] http://wmstutorials-9.0.aquaveo.com/demedit.zip
[12] http://wmstutorials-9.0.aquaveo.com/images.zip
[13] http://wmstutorials-9.0.aquaveo.com/feature.zip
[14] http://wmstutorials-9.0.aquaveo.com/featureadv.zip
[15] http://wmstutorials-9.0.aquaveo.com/4%20EditingElevations-DEMBasics.pdf
[16] http://wmstutorials-9.0.aquaveo.com/5%20EditingElevations-UsingTINs.pdf
[17] http://wmstutorials-9.0.aquaveo.com/dembasics.zip
[18] http://wmstutorials-9.0.aquaveo.com/tins.zip
[19] http://wmstutorials-9.0.aquaveo.com/7%20WatershedModeling-DEMDelineation.pdf
[20] http://wmstutorials-9.0.aquaveo.com/8%20WatershedModeling-AdvancedDEMDelineationTechniques.pdf
[21] http://wmstutorials-9.0.aquaveo.com/9%20WatershedModeling-TimeConcAndCN.pdf
[22] http://wmstutorials-9.0.aquaveo.com/demdelin.zip
[23] http://wmstutorials-9.0.aquaveo.com/nss.zip
[24] http://wmstutorials-9.0.aquaveo.com/tr-55.zip
[25] http://wmstutorials-9.0.aquaveo.com/10%20WatershedModeling-HEC1Interface.pdf
[26] http://wmstutorials-9.0.aquaveo.com/hec-1.zip
[27] http://wmstutorials-9.0.aquaveo.com/11%20WatershedModeling-HECHMSInterface.pdf
[28] http://wmstutorials-9.0.aquaveo.com/12%20WatershedModeling-RationalMethodInterface.pdf
[29] http://wmstutorials-9.0.aquaveo.com/rational.zip
[30] http://wmstutorials-9.0.aquaveo.com/13%20WatershedModeling-NSSInterface.pdf
[31] http://wmstutorials-9.0.aquaveo.com/14%20WatershedModeling-MaricopaNSS.pdf
[32] http://wmstutorials-9.0.aquaveo.com/15%20WatershedModeling-MaricopaHEC1.pdf
[33] http://wmstutorials-9.0.aquaveo.com/MARICOPA.zip
[34] http://wmstutorials-9.0.aquaveo.com/16%20WatershedModeling-ModratSchematic.pdf
[35] http://wmstutorials-9.0.aquaveo.com/17%20WatershedModeling-ModratGIS.pdf
[36] http://wmstutorials-9.0.aquaveo.com/18%20WatershedModeling-ModratMap.pdf
[37] http://wmstutorials-9.0.aquaveo.com/MODRAT.zip
[38] http://wmstutorials-9.0.aquaveo.com/19%20WatershedModeling-OCRational.pdf
[39] http://wmstutorials-9.0.aquaveo.com/20%20WatershedModeling-OCHydrograph.pdf
[40] http://wmstutorials-9.0.aquaveo.com/21%20WatershedModeling-OCSmallHydrograph.pdf
[41] http://wmstutorials-9.0.aquaveo.com/22%20WatershedModeling-OCUsingGIS.pdf
[42] http://wmstutorials-9.0.aquaveo.com/23%20WatershedModeling-OCRationalGIS.pdf
[43] http://wmstutorials-9.0.aquaveo.com/24%20WatershedModeling-OCHydrographGIS.pdf

[44] http://wmstutorials-9.0.aquaveo.com/OrangeCounty.zip
[45] http://wmstutorials-9.0.aquaveo.com/36%20Spatial-HMSModClark.pdf
[46] http://wmstutorials-9.0.aquaveo.com/37%20Spatial-GSSHAWizard.pdf
[47] http://wmstutorials-9.0.aquaveo.com/38%20Spatial-NexradModClark.pdf
[48] http://wmstutorials-9.0.aquaveo.com/39%20Spatial-NexradGSSHA.pdf
[49] http://wmstutorials-9.0.aquaveo.com/spatial.zip
[50] http://wmstutorials-9.0.aquaveo.com/40%20Gssha-WMSBasics-DEMImageCoords.pdf
[51] http://wmstutorials-9.0.aquaveo.com/41%20Gssha-WMSBasics-DEMDelineation.pdf
[52] http://wmstutorials-9.0.aquaveo.com/42%20Gssha-WMSBasics-MappingFeatures.pdf
[53] http://wmstutorials-9.0.aquaveo.com/Personal.zip
[54] http://wmstutorials-9.0.aquaveo.com/GsshaImages.zip
[55] http://wmstutorials-9.0.aquaveo.com/RawData.zip
[56] http://wmstutorials-9.0.aquaveo.com/tables.zip
[57] http://wmstutorials-9.0.aquaveo.com/WatershedDel.zip
[58] http://wmstutorials-9.0.aquaveo.com/43%20Gssha-Modeling-InitialSetup.pdf
[59] http://wmstutorials-9.0.aquaveo.com/44%20Gssha-Modeling-CorrectingOverland.pdf
[60] http://wmstutorials-9.0.aquaveo.com/45%20Gssha-Modeling-Infiltration.pdf
[61] http://wmstutorials-9.0.aquaveo.com/46%20Gssha-Modeling-StreamFlow.pdf
[62] http://wmstutorials-9.0.aquaveo.com/47%20Gssha-Modeling-Wizard.pdf
[63] http://wmstutorials-9.0.aquaveo.com/48%20Gssha-Modeling-PostProcessing.pdf
[64] http://wmstutorials-9.0.aquaveo.com/BasicGSSHA.zip
[65] http://wmstutorials-9.0.aquaveo.com/DigitalDam.zip
[66] http://wmstutorials-9.0.aquaveo.com/Infiltration.zip
[67] http://wmstutorials-9.0.aquaveo.com/Visualization.zip
[68] http://wmstutorials-9.0.aquaveo.com/49%20Gssha-Applications-Precipitation.pdf
[69] http://wmstutorials-9.0.aquaveo.com/50%20Gssha-Applications-LandUseChange1.pdf
[70] http://wmstutorials-9.0.aquaveo.com/51%20Gssha-Applications-LandUseChange2.pdf
[71] http://wmstutorials-9.0.aquaveo.com/52%20Gssha-Applications-LongTerm.pdf
[72] http://wmstutorials-9.0.aquaveo.com/53%20Gssha-Applications-SedimentTransport.pdf
[73] http://wmstutorials-9.0.aquaveo.com/54%20Gssha-Applications-ConstituentTransport.pdf
[74] http://wmstutorials-9.0.aquaveo.com/55%20Gssha-Applications-OverlandBoundaryConditions.pdf
[75] http://wmstutorials-9.0.aquaveo.com/Precipitation.zip
[76] http://wmstutorials-9.0.aquaveo.com/Scenarios.zip
[77] http://wmstutorials-9.0.aquaveo.com/LongTerm.zip
[78] http://wmstutorials-9.0.aquaveo.com/Sediment.zip
[79] http://wmstutorials-9.0.aquaveo.com/Contaminants.zip
[80] http://wmstutorials-9.0.aquaveo.com/BoundaryCondition.zip
[81] http://wmstutorials-9.0.aquaveo.com/56%20Gssha-Calibration-ManualCalibration.pdf
[82] http://wmstutorials-9.0.aquaveo.com/57%20Gssha-Calibration-Stochastic.pdf
[83] http://wmstutorials-9.0.aquaveo.com/58%20Gssha-Calibration-AutomatedCalibration.pdf
[84] http://wmstutorials-9.0.aquaveo.com/Calibration.zip
[85] http://wmstutorials-9.0.aquaveo.com/59%20Gssha-Groundwater-GroundwaterModeling.pdf
[86] http://wmstutorials-9.0.aquaveo.com/60%20Gssha-Groundwater-AdvancedGroundwaterModeling.pdf
[87] http://wmstutorials-9.0.aquaveo.com/61%20Gssha-Groundwater-TileDrains.pdf
[88] http://wmstutorials-9.0.aquaveo.com/Groundwater.zip
[89] http://wmstutorials-9.0.aquaveo.com/SubSurface.zip
[90] http://wmstutorials-9.0.aquaveo.com/25%20HydraulicsAndFloodplainModeling-HECRASAnalysis.pdf
[91] http://wmstutorials-9.0.aquaveo.com/26%20HydraulicsAndFloodplainModeling-HECRASManagingCrossSections.pdf
[92] http://wmstutorials-9.0.aquaveo.com/hecras.zip
[93] http://wmstutorials-9.0.aquaveo.com/xsecs.zip
[94] http://wmstutorials-9.0.aquaveo.com/27%20HydraulicsAndFloodplainModeling-FloodplainDelineation.pdf
[95] http://wmstutorials-9.0.aquaveo.com/28%20HydraulicsAndFloodplainModeling-StochasticModeling.pdf
[96] http://wmstutorials-9.0.aquaveo.com/flood.zip
[97] http://wmstutorials-9.0.aquaveo.com/stochastic.zip
[98] http://wmstutorials-9.0.aquaveo.com/29%20HydraulicsAndFloodplainModeling-SimplifiedDamBreak.pdf
[99] http://wmstutorials-9.0.aquaveo.com/smpdbk.zip
[100] http://wmstutorials-9.0.aquaveo.com/30%20HydraulicsAndFloodplainModeling-HY8.pdf
[101] http://wmstutorials-9.0.aquaveo.com/HY8WizardTutorialFiles.zip
[102] http://wmstutorials-9.0.aquaveo.com/62%20HydraulicsAndFloodplainModeling-HydraulicToolbox.pdf

[103] http://wmstutorials-9.0.aquaveo.com/HydraulicToolbox.zip
[104] http://wmstutorials-9.0.aquaveo.com/31%20StormDrainModeling-SWMMModeling.pdf
[105] http://wmstutorials-9.0.aquaveo.com/stormrat.zip
[106] http://wmstutorials-9.0.aquaveo.com/32%20StormDrainModeling-StormDrainRationalDesign.pdf
[107] http://wmstutorials-9.0.aquaveo.com/33%20StormDrainModeling-StormDrainHydrographicDesign.pdf
[108] http://wmstutorials-9.0.aquaveo.com.s3.amazonaws.com/63%20StormDrainModeling-HY12RationalDesign.pdf
[109] http://wmstutorials-9.0.aquaveo.com.s3.amazonaws.com/HY12Rational.zip
[110] http://wmstutorials-9.0.aquaveo.com/34%20WaterQualityModeling-HSPF.pdf
[111] http://wmstutorials-9.0.aquaveo.com/hspf.zip
[112] http://wmstutorials-9.0.aquaveo.com/35%20WaterQualityModeling-CEQUALW2.pdf
[113] http://wmstutorials-9.0.aquaveo.com/cequal.zip

Where Can I Get Data?

(Click on *italicized links* to see a Flash instructional file)

Aquaveo maintains a website with useful links to locations that distribute geospatial data such as DEM, land use, soils, images, and other data useful for hydrologic modeling. The website is periodically updated by graduate students and research assistants at Brigham Young University. There are hundreds, if not thousands, of sites with geospatial data that are useful for modeling in WMS, but we have tried to distill from these the most significant sites. However, if you are aware of sites that might be more valuable, or as valuable, please contact technical support and let us know.

Visit the GeoSpatial Data Acquisition (GSDA) website now for further help with downloading and preparing digital data for use in WMS.

Download DEMs from the USGS NED site [1]

Download SSURGO data [2]

Related Topics

- Getting Started

References

[1] http://www.ems-i.com/WMS/captivate/DownloadDEM.html
[2] http://www.ems-i.com/WMS/captivate/SSURGO.html

2.1. Layout

Graphical User Interface (GUI)

One of the problems with learning any new software interface is the need to understand both what the program can do (functionality) as well as how the program works (GUI controls). It is a chicken and egg kind of problem because things like display/contour options, panning, zooming, saving files, etc. are meaningless without a purpose or reason, and yet it is hard to also learn how WMS does hydrology without knowing some of the basic GUI operations. Past experience tells us that "sneaking" in some of these operational ideas with the functionality of the tutorials can be confusing and so we have developed this section in order to provide some basic helps on some of the common questions that come up about display options, contour options, manipulating the display, and file management.

Interface Layout

The [WMS:Quick_Tour_-_Getting_StartedQuick Tour provides a good overview of the interface layout, as well as other things. If you haven't gone through this tour you should do so now. Specifically you can view the different parts of the WMS GUI, learn about the modules, tools, and macro palettes, and understand what the Edit and Help strips are.

Using the Project Explorer

The Project Explorer has become an important part of the WMS interface. It is used to create new data objects (i.e. coverages, TINs, etc.), to control the visibility of data objects, change the coordinate system, delete, and more. Learning how to interact with the Project Explorer is an important part of understanding WMS.

Pan, Zoom, and Rotate

Most graphical user environments have the ability to zoom in/out on data, pan, and rotate. Learn more about how these tools work in WMS in the Static Tool Palette article.

Display Options

The following display options can be set and controlled from the Display Options dialog:

- Symbol, line, and text color
- Symbol style and size
- Line style and thickness
- Polygon color
- Text font and style

Check boxes are used to determine whether or not individual entities (points, triangles, contours, etc.) are to be displayed. The check box next to a data object in the Project Explorer window sets the visibility of all entities for that object, regardless of the individual settings in the Display Options dialog.

A grid displayed for snapping vertices/points when digitizing, or to use as a scale bar can be defined using the Grid Options command from the Display menu.

Contour Options

Visualization of contours is an important part of any program that utilizes digital terrain data. The following links provide key information and examples about WMS's contouring capabilities.

- Setting contour options such as method, legends, and ranges.
- Selecting color schemes.
- Choosing the contour interval.
- Placing labels.

File Management

WMS incorporates the use of many different files and it can be a bit confusing (especially at first) how all of these files are opened and saved. The following are basic guidelines to file management in WMS with links to more detailed information.

- File types are typically identified by their three letter extension. When opening a file the extensions listed relate to the current module. For example if you are in the Map Module then it will look for files with *.map, *.dxf, *.dwg, *.tif, etc. since these are the data used primarily in this module. However, any WMS file can be opened from any module, regardless of whether it's extension is currently listed.
- Because WMS manages so many different file types it incorporates a project file which reads/saves all files and then stores the name and type of file in the project file. It is recommended that you save project files in their own separate directory.
- The type of file saved is identified by the extension. Unless saving a project file, only the specified file type is saved.
- Sometimes files with formats WMS is capable of reading do not have the same extension as used by WMS. In such cases the file can still be read without changing the extension, however you may have to explicitly define the type of file format it is since WMS will not recognize the extension.
- Some file types are native to WMS (the format is determined by WMS) and some file types are non-native (defined by some other program, or industry standard).
- If you drag a file from Windows Explorer into the main graphics window of WMS it will automatically be opened (as long as the extension matches a WMS supported file type).
- WMS will remember recently opened files and display them at the bottom of the File menu for quick access.

Documentation

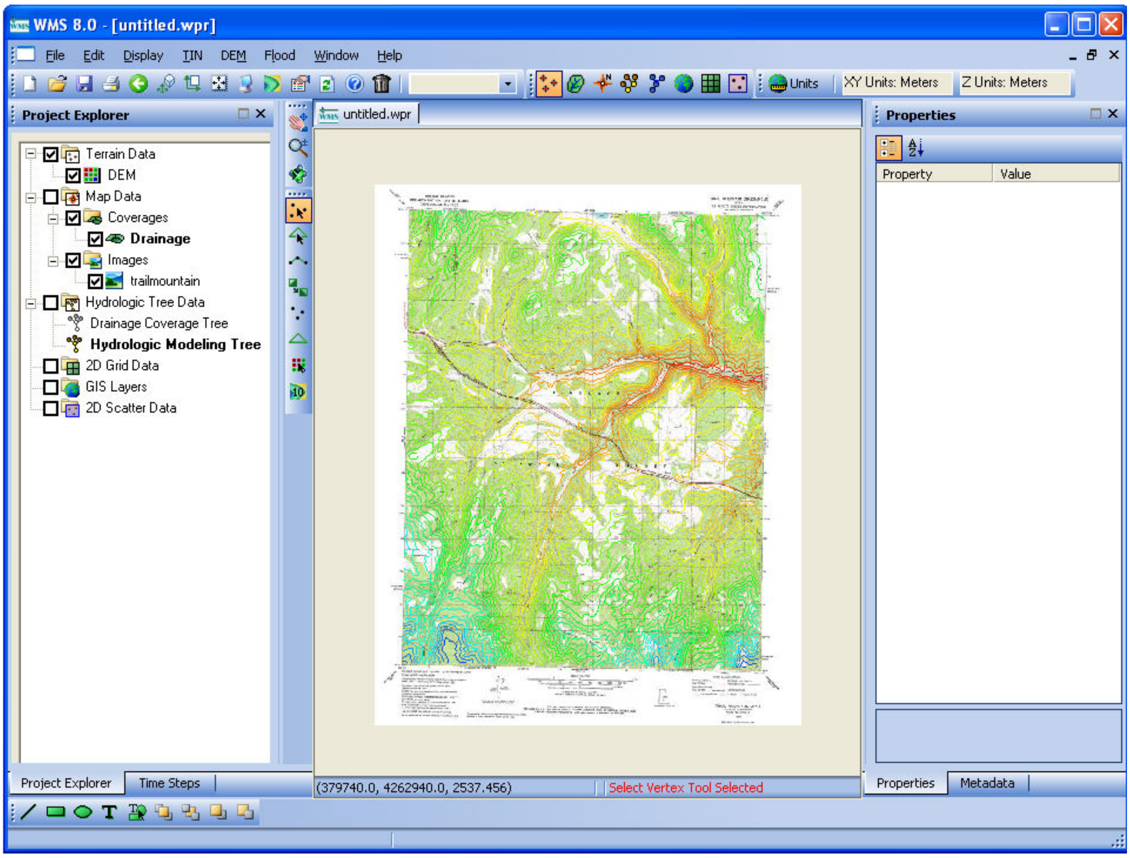
This help file comprises the reference information for the WMS interface. Separate *.pdf files for supported models, along with the tutorials for WMS can be found in a directory named "docs" under the main installation directory of WMS. You can open these files with Adobe Acrobat and print if you desire.

Related Links

- Getting Started with WMS

WMS Screen

The WMS Screen is divided into seven main sections: *Menu bar*, *Edit Window*, *Tool Palette* (modules, tools, and macros), *Help strip*, *Project Explorer*, *Cursor tracking* and the main *Graphics Window*. The different windows of the interface can be separated from the main panel as floating windows, or re-docked in other positions (i.e. the *Project Explorer* could be moved from the left to the right of the interface).



Related Topics

- Menu Bar
- Edit Window
- Module Palette
- Tool Palette
- Macros
- Help Strip
- Project Explorer

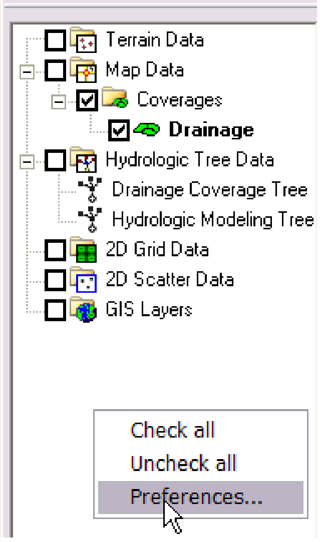
Overview

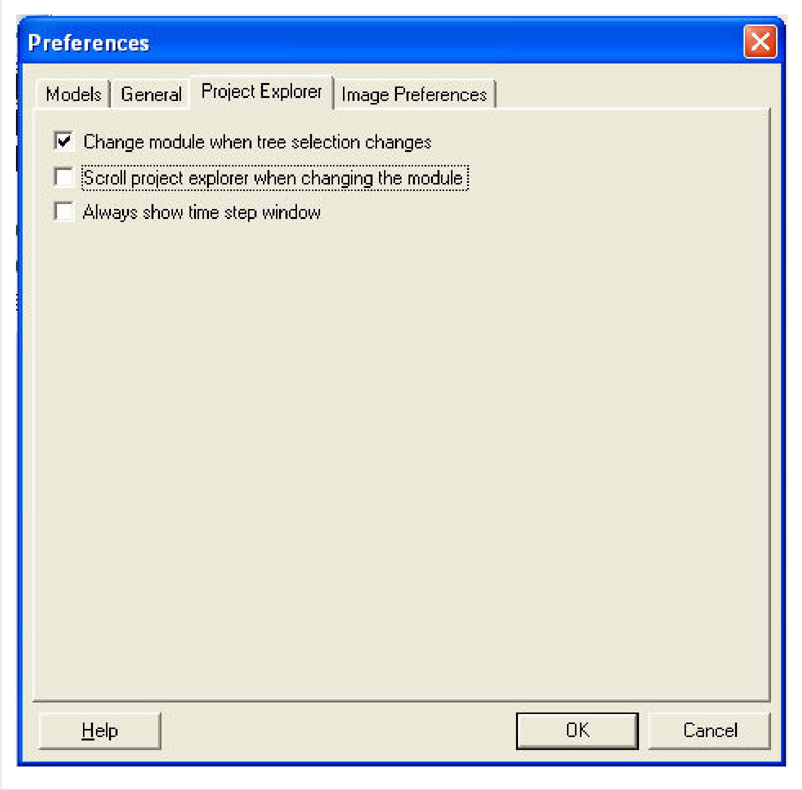
The *Project Explorer* is located at the left side of the WMS window by default, but it can be moved to anywhere on the window since it is a "dockable" toolbar. The *Project Explorer* contains a hierarchical representation of the data associated with a modeling project.

The *Project Explorer* manages the data in multiple ways, including creation of new data objects, deleting data objects, and the control of display.

Basic Project Explorer Manipulation

When WMS is initiated an empty folder for each of the data modules is created and displayed in the *Project Explorer* (previous versions only displayed the folder for the active module). New datasets (TINs, coverages, etc.) can be created by right-clicking on the folders or by opening existing files. A preferences dialog can be opened to control display and default settings/behaviors in the Project Explorer by right-clicking in the empty space of the *Project Explorer*.

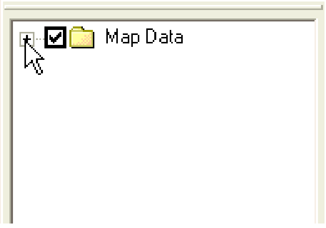




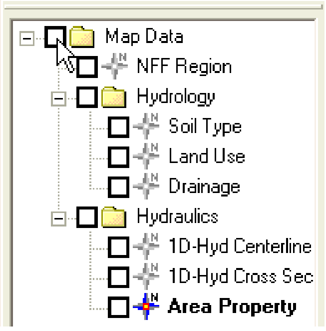
The *preferences* dialog can be used to control the following:

- By default WMS will not switch modules when selecting one of the folders, or data objects within a folder from the *Project Explorer*, but you can change your preferences so that each time a data folder/object is selected WMS automatically switches to that modules as if the module icon had been simultaneously selected
- The time step window used with transient datasets of TINs, grids, and scatter points is not shown at the bottom of the *Project Explorer* unless such a dataset exists and the TIN, grid, or scatter point set is selected. An option can be toggled on so that it always appears as it did in previous versions.
- The scroll project explorer when changing the module option forces WMS to scroll the *Project Explorer* such that the data folder corresponding to the active module appears at the top of the window.
- WMS 8.0 deals with images differently than past versions. The Image Preferences tab controls these options, including building pyramids and image conversion.

The data folders of each module behave similar to *Windows Explorer*, except there is an included check box which controls the display of the folder, or data object and all objects underneath. The *Project Explorer* can be expanded/contracted by selecting the + or - box at the left side as shown below.



The display of all objects within a folder, or individual data object can be turned on or off using the check box at the left side as shown below.



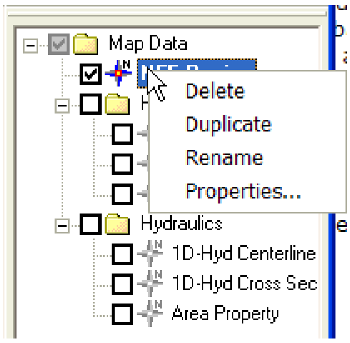
Creating Data Objects and Folders

New folders and data objects are created by right clicking the mouse on the folder where you wish to store it (they are created automatically when reading new files).

Data objects can be stored within folders, or directly under the main folder for each module. Folders and data objects can be renamed and moved within the *Project Explorer* structure of a given module. The entire *Project Explorer* structure is saved as part of a project file, and not as part of a module file such as a TIN file or a MAP file. So in order to restore the entire *Project Explorer*, it must be done as part of a project file.

Project Explorer Contents By Module

The contents of the *Project Explorer* vary, depending on which module is active. Pop-up menus are available for folders and files within the *Project Explorer* and these menus vary for the different data objects available. In general each folder and object has a pop up menu that can be accessed using the right mouse button that include the ability to create, delete, rename, and control properties.



Specific behaviors and contents for the *Project Explorer* are provided for each module:

- Terrain Data
- Drainage
- Map
- Hydrologic Modeling
- River
- GIS
- 2D Grid
- 2D Scatter

Related Topics

- WMS Screen

WMS Dialog Help

This is a special page that relates WMS dialogs to wiki pages. WMS reads this page when a user hits the *Help* button in a WMS dialog, and opens the wiki at the page indicated below.

Wiki Page | Dialog Number | Dialog ID

- Arc GIS Mapping | 140 | IDD_ARCGISMAPPING
- Assign Land Use | 4591 | IDD_ASSIGN_LANDUSE
- Assign Soil Type | 4590 | IDD_ASSIGN_SOILTYPE
- Assign Profile | 386 | IDD_ASSIGNPROFILE
- Bridge/Culvert Parameters | 662 | IDD_BRIDGECULVATTS
- Bridge/Culvert Cross Section Attributes | 652 | IDD_BRIDGECULVERT
- GSSHA Channel Routing Parameters | 194 | IDD_C2DCHANNELROUTINGPARAMETERS
- Polygon Branch Attributes | 4532 | IDD_CEQUAL_BRANCHATTS
- Constituent Control | 156 | IDD_CEQUAL_CONSTITUENTCONTROL
- Select Credible Interval | 729 | IDD_CEQUAL_CREDIBLEINTERVALS
- Define Gates | 4545 | IDD_CEQUAL_GATES
- CE-QUAL-W2 Layer Editor | 1401 | IDD_CEQUAL_LAYEREDITOR
- Node Attributes | 4537 | IDD_CEQUAL_NODEATTS
- Define Outlet Structures (NST) | 4540 | IDD_CEQUAL_OUTLETSTRUCTURES
- Define Pipes | 4543 | IDD_CEQUAL_PIPES
- Polygon Segment Attributes | 4497 | IDD_CEQUAL_POLY_SEG_ATTS
- Define Pumps | 4542 | IDD_CEQUAL_PUMPS
- Segment Profile Plot | 163 | IDD_CEQUAL_SEGMENTSPROFILE
- Selected Simulated Constituents | 463 | IDD_CEQUAL_SIMULATEDCONSTITUENTS
- Snap Shot Print Parameters | 4546 | IDD_CEQUAL_SNAPSHOTPRINT
- Define Spillways | 4541 | IDD_CEQUAL_SPILLWAYS
- Time Step | 4555 | IDD_CEQUAL_TIMESTEPJOBCONTROL
- Define Tributaries (NTR) | 4539 | IDD_CEQUAL_TRIBUTARIES
- Water Body Job Control | 161 | IDD_CEQUAL_WATERBODYJOBCONTROL
- Define Internal Weirs | 4544 | IDD_CEQUAL_WEIRS
- Define Withdrawl | 144 | IDD_CEQUAL_WITHDRAWAL
- Channel Calculations | 4392 | IDD_CHANNEL_CALCULATIONS
- View Data File | 204 | IDD_CHOOSETEXTEDITOR
- Film Loop | 4390 | IDD_CLOCKOPTIONS
- Color Ramp | 6 | IDD_COLOROPTIONS
- Clean Options | 452 | IDD_CLEAN_FEATURE_OBJECTS
- HEC-HMS | 4692 | IDD_COMPHMSGRID
- Compute GIS Attributes | 4479 | IDD_COMPUTEGISATTS
- Compute NSTPS | 3426 | IDD_COMPUTENSTPS
- Contour Label Options | 3442 | IDD_CONTLABELOPTIONS
- Contour Options | 4650 | IDD_CONTOUR_OPTIONS
- Contour Legend Options | 3441 | IDD_CONTOURLEGEND
- Convert TIN to DEM | 4674 | IDD_CONVERT_TIN
- DEM Conversion Options | 367 | IDD_CONVERTTODEMOPTS
- Coordinate Transformation | 361 | IDD_COORD_CONVERT
- User Defined Ellipsoid | 374 | IDD_COORD_ELLIPSE

- Coordinate Conversion | 4473 | IDD_COORD_SINGLEPT
- Coordinate System | 482 | IDD_COORD_SYSTEM
- Coverage Overlay | 4672 | IDD_COVERAGE_OVERLAY
- Create Grid | 4596 | IDD_CREATEGRID
- Create TIN Options | 4626 | IDD_CREATETIN
- Cross-Section Attributes | 3391 | IDD_CSATTS
- CsDb Management | 3351 | IDD_CSDB_MANAGEMENT
- CsDb Topo Management | 3352 | IDD_CSTOPOMANAGE
- Curb and Gutter Calculator | 3399 | IDD_CURBANDGUTTERCALC
- Data Calculator | 471 | IDD_DATA_CALCULATOR
- Data Service Options | 45002 | IDD_DATASERVICE_OPTIONS
- File Formats | 1142 | IDD_DATASETIMPORT
- Dataset Info | 4255 | IDD_DATASET_INFO1
- Delete TINs | 224 | IDD_DELETE_TINS
- Importing USGS DEMs | 4569 | IDD_DEM_IMPORT
- DEM Smoothing Options | 4570 | IDD_DEM_SMOOTHING_OPTS
- DEM Accumulation Options | 3361 | IDD_DEMACCUMOPTIONS
- DEM Attributes | 4261 | IDD_DEMATTRIBUTES
- DEM Point Attributes | 4271 | IDD_DEMPOINTATTS
- Update DEM/Basin Data | 4478 | IDD_DEMUPDATEDATA
- Elevation - Storage Capacity - Discharge | 4558 | IDD_DETBASINCURVES
- Detention Basin Hydrograph Routing | 4559 | IDD_DETENTIONBASIN
- Display Options | 4641 | IDD_DISPLAY_OPTIONS
- CAD Display Options | 4342 | IDD_DISPOPTS_DXF
- Units | 1204 | IDD_DRAIN_UNITS
- Drawing Depth | 3397 | IDD_DRAWINGDEPTH
- DSS File Editor | 4598 | IDD_DSS_FILEEDITOR
- New DSS Dataset | 4599 | IDD_DSS_NEWDATASET
- DSS - Edit Paired dataset | 4622 | IDD_DSSPAIREDDATASET
- Select CAD Layers | 4344 | IDD_DXF_SELECT_LAYERS
- Edit Dataset Values | 4257 | IDD_EDIT_DATASET
- Edit View | 4580 | IDD_EDIT_VIEW
- Edit Elevations | 4258 | IDD_EDITELEVATIONS
- Elevation Discharge Input | 4562 | IDD_ELEVATIONDISCHARGE
- Export WMS Superfile Data | 4675 | IDD_EXPORT_WMSSUPERFILE
- Export Shapefile | 4260 | IDD_EXPORTSHAPEFILE
- Extract Cross Sections | 4709 | IDD_EXTRACTCS
- File Import Filter Options | 4671 | IDD_FILE_IMPORT_FILTER_OPTS
- Add View | 4388 | IDD_FILMLOOP_ADDVIEW
- Flood Extent Coverage | 4492 | IDD_FLOOD_COVERAGE
- Flood Depth Coverage | 4494 | IDD_FLOOD_DEPTH_COVERAGE
- Flood Impact Coverage | 4676 | IDD_FLOOD_IMPACT
- Flood Extent Attributes | 4612 | IDD_FLOODEXTENT_ATTS
- Floodplain Delineation | 4268 | IDD_FLOODPLAIN_DELINEATION
- Flow Accumulation Options | 3406 | IDD_FLOWACCUMOPTS
- Frequency Distribution | 4508 | IDD_FREQUENCY_DISTRIBUTION
- Plot Options | 4652 | IDD_GAGE_PLOT_OPTIONS

- Gages | 4643 | IDD_GAGES
- Legend Options | 4702 | IDD_GENERALLEGEND
- Join Table | 388 | IDD_GISGENERICJOIN
- Grid Options | 1428 | IDD_GRIDOPTS
- GSSHA Contaminants | 381 | IDD_GSSHA_CONTAMINANTS
- GSSHA Job Control Parameters | 4708 | IDD_GSSHA_JOB
- GSSHA Output Control | 406 | IDD_GSSHA_OUTPUT
- Overland soil erosion | 385 | IDD_GSSHA_SEDIMENTS
- GSSHA Stochastic Batch Run | 4715 | IDD_GSSHA_STOCHASTICSNEW
- Embankment Arc Profile Editor | 3413 | IDD_GSSHAARCEMBANKMENTATTS
- GSSHA Clean Dams | 4551 | IDD_GSSHACLEANDAMS
- GSSHA Maps | 4727 | IDD_GSSHAINDEXMAPS
- GSSHA Map Table Editor | 4690 | IDD_GSSHAMAPTABLE
- GSSHA Nutrients | 4694 | IDD_GSSHANUTRIENTS
- GSSHA Polygon Attributes | 3410 | IDD_GSSHAPOLYATTS
- GSSHA Precipitation | 4693 | IDD_GSSHAPRECIP
- GSSHA Reclassify Map | 4719 | IDD_GSSHARECLASSIFY
- Save GSSHA Project File | 4637 | IDD_GSSHASAVEPROJECT
- Smooth GSSHA Streams | 4262 | IDD_GSSHASMOOTHSTREAMS
- GSSHA Thalweg | 4656 | IDD_GSSHATHALWEG
- GSSHA Observations | 383 | IDD_GSSHAOBSERVATIONS
- GSSHA Solution Analysis | 22044 | IDD_GSSHASOLUTIONANALYSIS
- GSSHA Groundwater | 4623 | IDD_GSSHA_SUBSURFACE
- HEC Gage Weights | 4640 | IDD_HEC_GAGEWEIGHTS
- Basin Time Computation | 4391 | IDD_HEC1_BASINCOMPUTATION
- HEC-1 Basin Data | 3428 | IDD_HEC1_BASINDATA1
- Modify Equation | 4395 | IDD_HEC1_BASINMODIFY
- Basin Geometric Attributes | 4396 | IDD_HEC1_BASINTCDATA
- HEC-1 Model Check | 4610 | IDD_HEC1_CHECKMODEL
- Define MA Data | 4475 | IDD_HEC1_DEFINEMA
- Depth/Area Storm | 3445 | IDD_HEC1_DEPTHAREASTORM
- HEC-1 Diversion Data | 3457 | IDD_HEC1_DIVERSIONDATA
- HEC-1 Run Options | 4625 | IDD_HEC1_GET_FILES
- HEC-1 Snow Melt Data | 3454 | IDD_HEC1_GETSNOWMELTDATA
- HEC-1 Job Control | 3447 | IDD_HEC1_JOBCONTROL
- Multiflood Storms | 3448 | IDD_HEC1_MULTIFLOODSTORMS
- HEC-1 Output Control | 3436 | IDD_HEC1_OUTPUTCONTROL
- Edit HEC-1 Parameters | 4669 | IDD_HEC1_PARAM
- HEC-1 Precipitation | 354 | IDD_HEC1_PRECIPITATION1
- RC Channel Record | 3432 | IDD_HEC1_RCCHANNELRECORD
- Reservoir Routing Data | 4560 | IDD_HEC1_RESROUTDATA
- HEC-1 Reservoir Routing Options | 4557 | IDD_HEC1_RESROUTOPTS
- RK/RD Channels | 3433 | IDD_HEC1_RKRDCHANNELS
- HEC-1 Unit Hydrograph Methods | 3449 | IDD_HEC1_UNITHYDRO
- HEC-1 Loss Methods | 3411 | IDD_HEC1LOSSMETHOD
- HMS Meteorological Model | 4639 | IDD_HMS_METMODELDATA
- HMS Properties | 453 | IDD_HMS_PROPERTIES

- Add Sub-Basin | 4602 | IDD_HSPF_ADD_SUBBASIN
- HSPF Global Options | 4697 | IDD_HSPF_GLOBALOPTS
- Mass Link Editor | 4603 | IDD_HSPF_MASSLINK
- Assign External Targets | 4632 | IDD_HSPF_OUTPUTTIMESERIES
- Run HSPF | 459 | IDD_HSPF_RUNSIMULATION
- HSPF Segments | 4699 | IDD_HSPF_SEGMENTS
- Assign External Sources | 4619 | IDD_HSPF_WDM_TIMESERIES
- HSPF | 533 | IDD_HSPFPARAM
- Analysis Results | 165 | IDD_HY8_ANALYSIS_RESULTS
- Energy Dissipation | 244 | IDD_HY8_ENERGY_DISSIPATION
- Input Data | 708 | IDD_HY8_INPUT_PARAMS
- Irregular Tailwater Channel | 714 | IDD_HY8_IRREG_TWCHANNEL
- Report Generator | 211 | IDD_HY8_REPORTGENERATOR
- FHWA Storm Drain Job Control Information | 3412 | IDD_HYDRAJOBCONTROL
- Pipe Data Properties | 4642 | IDD_HYDRAPIPEDATA
- Pressure Flow Simulation | 3415 | IDD_HYDRAPRESSUREFLOW
- Rainfall Data | 3423 | IDD_HYDRARAINFALL
- Run Storm Drain | 154 | IDD_HYDRARUNSIM
- Storm Drain Node Attributes | 146 | IDD_HYDRASTORMDRAINATTS
- Image Display Options | 3371 | IDD_IMAGEDISPOPTS
- Import DLG Files | 4677 | IDD_IMPORT_DLGS
- Import WMS Superfile Data | 4657 | IDD_IMPORT_WMSSUPERFILE
- Import Flow Directions | 3405 | IDD_IMPORTFLOWDIR
- 2D Interpolation Options | 4523 | IDD_INTERPOLATE2D
- Interpolate Cross Sections | 3393 | IDD_INTERPOLATECS
- Subset Definition | 4522 | IDD_INTERPOLATESUBSET
- 2D IDW Interpolation Options | 4521 | IDD_INTERPOLATIONOPTIONS
- Land Poly Atts | 4592 | IDD_LAND_POLY_ATTS
- Line Properties | 524 | IDD_LINE_DISPLAY_ATTS
- Line Attributes | 3407 | IDD_LINEARROW_ATTS
- Manage Cross Sections | 405 | IDD_MANAGE_CROSSSECS
- Materials Data | 68 | IDD_MATERIALS
- Merge TINs | 3373 | IDD_MERGE_TINS
- Dataset Metadata | 4684 | IDD_METADATA_DATASET
- Metadata | 3531 | IDD_METADATA_MAIN
- Profile Information | 3532 | IDD_METADATA_PROFILE
- Source Metadata | 3533 | IDD_METADATA_SOURCE
- Spatial Metadata | 3535 | IDD_METADATA_SPATIAL
- Generate Burned Simulation | 4525 | IDD_MODRATBURNED
- Compute MODRAT Time of Concentration Wizard | 4507 | IDD_MODRATCOMPUTETC
- MODRAT Job Control | 4710 | IDD_MODRATJOBCONTROL
- MODRAT Parameters | 4711 | IDD_MODRATPARAMS
- MODRAT Renumber | 4564 | IDD_MODRATRENUMBER
- Natural Neighbor Options | 4631 | IDD_NATURALNEIGHBOROPTIONS
- New Palette | 80 | IDD_NEWPALETTE
- Basin Regions | 101 | IDD_NFF_BASINREGIONS
- National Flood Frequency Regression Method | 4270 | IDD_NFF_RUN

- Flow Weighting Options | 226 | IDD_NFF_WEIGHTOPTS
- NFF Region Polygon Attributes | 3430 | IDD_NFFPOLYATTS
- Compute Fm | 4646 | IDD_OCCOMPUTEFM
- Edit Orange County Unit Hydrograph Parameters | 4526 | IDD_OCHEC1PARAM
- Orange County Precipitation | 4527 | IDD_OCPRECIPWIZ
- OC Rational Method - Job Control | 4571 | IDD_OCRATIONALJOBCONTROL
- Orange County Rational Method - Tree Mapping | 4576 | IDD_OCRATIONALMAPPING
- Orange County Rational Method - Edit Parameters | 4628 | IDD_OCRATIONALPARAM
- Orange County Method -- Rational Method results | 4561 | IDD_OCRATIONALRESULTS
- WMS:OC Rational Initial Sub-area Time of Concentration | 4530 | IDD_OCRATIONALTC
- Orange County Small Area Hydrograph | 4582 | IDD_OCSMALLHYDROWIZ
- Orange County Storm | 4552 | IDD_OCSTORM
- Orange County Unit Hydrograph | 4531 | IDD_OCUNITHYDRO
- Open File Format | 1471 | IDD_OPENFILE_FORMAT
- Color Table | 3356 | IDD_PALETTE_TABLE
- Pattern Attributes | 3350 | IDD_PATTERN_DISPLAY_ATTS
- Plot Wizard - Step 1 of 2 | 3357 | IDD_PLOTWIZ1
- Point Properties | 523 | IDD_POINT_DISPLAY_ATTS
- Input Variables for PREFRE IDF Computation | 130 | IDD_PREFREVARS
- Time of Concentration | 4703 | IDD_PRINCEGEORGES_TC
- Properties | 4254 | IDD_PROPERTIES
- Publish Data | 448 | IDD_PUBLISH_DATA
- Rain Gage Properties | 4689 | IDD_RAINGAGEPROPS
- Rational Method Hydrographs | 4563 | IDD_RATIONAL_METH_HYDRO
- Rational Method | 570 | IDD_RATIONAL_METHOD
- Rational Method -- IDF Computation | 3421 | IDD_RATIONALIDF
- Read Solution|Read Solution | 1900 | IDD_READSOLUTION
- Rectangle/Oval Attributes | 3408 | IDD_RECTOVAL_ATTS
- Redistribute Verticies | 1429 | IDD_REDISTVERT
- Register Image | 498 | IDD_REGIMAGE_GM2
- Register | 4586 | IDD_REGISTER
- Return to Feature | 4579 | IDD_RETURNTOFEATURE
- Hydraulic Model Boundary Conditions | 4698 | IDD_RIVHYD_FLOWBC
- River Cross Section Attributes | 4696 | IDD_RIVHYD_XSECTION_ATTS
- River Reach Attributes | 3394 | IDD_RIVHYD_REACHATTS
- HEC-1 Routing Data | 4556 | IDD_ROUTING
- Write DNMCNT | 4607 | IDD_RUN_DEDNM
- SAC-SMA Model | 369 | IDD_SACRAMENTO
- Scatter Data Color Options | 4281 | IDD_SCAT_COLOR_OPTS
- Lighting Options | 3343 | IDD_SHADINGOPTS
- Stochastic Run Parameters | 343 | IDD_STOCHASTIC_RUNPARAMS
- Storage Capacity Input | 1266 | IDD_STORAGECAPINPUT
- Stream Feature Arc Options | 3401 | IDD_STREAMFARCOPTS
- Text Attributes | 3409 | IDD_TEXT_ATTS
- Time Computation Arc Attributes | 4477 | IDD_TIMECOMP
- Tin Options | 3360 | IDD_TIN_OPTIONS
- Topaz Run Options | 4608 | IDD_TOPAZ_RUN_OPTS

- Edit TR-20 Parameters | 4621 | IDD_TR20_ATTRIBUTES
- TR-20 Base Flow | 4572 | IDD_TR20_BASEFLOW
- TR-20 Basin Data | 4617 | IDD_TR20_BASINDATA
- TR-20 Diversion Data | 4620 | IDD_TR20_DIVERSIONDATA
- Define Input Hydrographs | 4589 | IDD_TR20_INPUTHYDRO
- TR-20 Job Control | 4594 | IDD_TR20_JOBCONTROL
- TR-20 Output Control | 4577 | IDD_TR20_OUTPUTCONTROL
- TR-20 Precipitation | 4567 | IDD_TR20_PRECIPITATION
- TR-20 Reservoir Routing | 4605 | IDD_TR20_RES_ROUTING
- TR-20 Reservoir Data | 4486 | IDD_TR20_RESDATA
- TR-20 Routing Data | 4611 | IDD_TR20_ROUTING
- TR-20 Cross Section Data | 4487 | IDD_TR20_XSECTDATA
- TR-55 | 567 | IDD_TR55_PROPERTIES
- Transform TIN | 4566 | IDD_TRANSFORM_TIN
- Travel Time Computation | 4474 | IDD_TRAVELTIME
- Weir Calculations | 1258 | IDD_WEIRCALCULATION
- Set Window Boundaries | 3340 | IDD_WINDOWBOUNDS
- XY Series Editor | 131 | IDD_XYSERIES
- Dataset Zonal Classification | 22042 | IDD_DATASET_ZONAL_CLASSIFICATION
- Dataset Zonal Classification Display | 22043 | IDD_DATASET_ZONAL_CLASSIFICATION_DISPLAY
- Project Filename | 4722 | IDD_HMW_PROJECTFILENAME
- Define Project Bounds | 4723 | IDD_HMW_DEFINEPRJBOUNDS
- Watershed Data | 694 | IDD_HMW_WSDATA
- Download Data (Web Services) | 695 | IDD_HMW_DOWNLOADDATA
- Read Data (Catalog) | 696 | IDD_HMW_READDATA
- Compute Flow Directions and Flow Accumulations | 697 | IDD_HMW_COMPUTE
- Choose Outlet Locations | 698 | IDD_HMW_CHOOSEOUTLET
- Delineate Watershed | 699 | IDD_HMW_DELINWATERSHED
- Select Model | 700 | IDD_HMW_SELECTMODEL
- Define and Smooth Streams | 22041 | IDD_HMW_DEFINESTREAMS
- Create 2D Grid | 703 | IDD_HMW_CREATE2DGRID
- Job Control | 701 | IDD_HMW_JOBCONTROL
- Define Land Use and Soil Data | 22037 | IDD_HMW_DEFLANDSOIL
- Hydrologic Computations | 702 | IDD_HMW_HYDCOMPS
- Define Precipitation | 22039 | IDD_HMW_DEFINE_PRECIP
- Clean Up Model | 725 | IDD_HMW_CLEAN
- Feature Coverage Atts | 22040 | IDD_COVATTS
- Define Culvert Roadway Data | 22055 | IDD_HY8WIZ_DEFINE_ROADWAY_DATA
- Run Hydrologic Model | 22056 | IDD_HMW_RUN_HYDRO_MODEL
- Crossing Discharge Data | 22057 | IDD_HMW_CROSSING_DISCHARGE_DATA
- Setup Tailwater Channel | 22058 | IDD_HY8WIZ_DEFINE_TAILWATER_DATA
- Culvert and Site Data | 22062 | IDD_HY8WIZ_CULVERT_SITE_DATA
- Run Culvert Analysis | 22059 | IDD_HY8WIZ_RUN_CULVERT
- Define Flood Inundation Polygon | 22063 | IDD_HY8WIZ_DEFINE_FLOOD_POLYGON
- Storage Capacity Data | 22060 | IDD_HY8WIZ_STORAGE_CAPACITY
- Define Upstream Channel | 22061 | IDD_HY8WIZ_DEFINE_HEADWATER_DATA
- Delineate Inundated Area | 22064 | IDD_HY8WIZ_DELINEATE_FLOOD

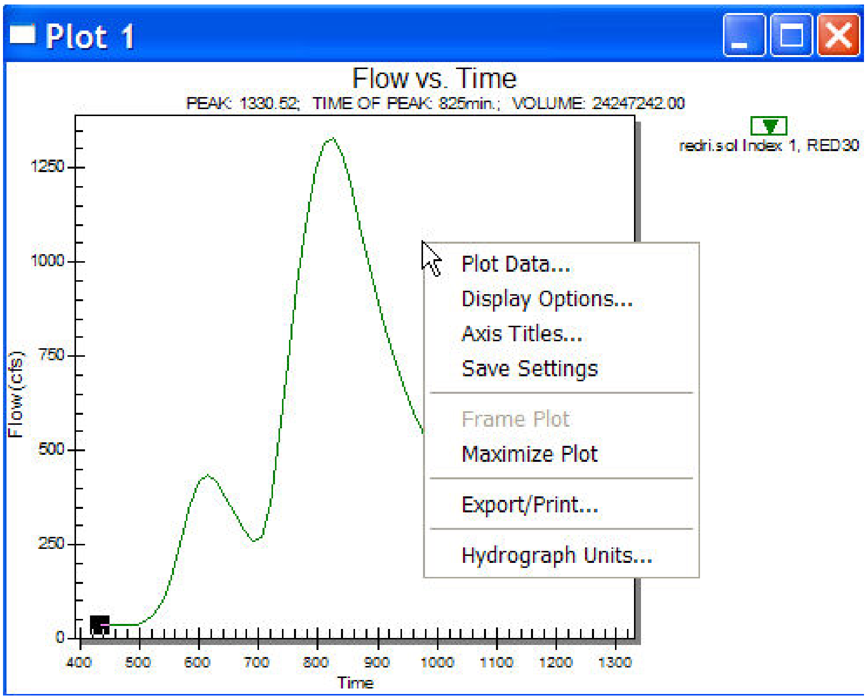
- GSSHA Pipe and Node Parameters | 22069 | IDD_GSSHA_PIPENODEPARAMS
- Get Online Maps | 43000 | IDD_GETONLINEIMAGES

Plot Windows

WMS uses a standard library for generating 2-Dimensional plot data such as time series data and hydrographs. A plot window is often embedded within a dialog to display time series data such as rainfall distributions or storage capacity curves, but the primary use of *Plot Windows* are to display the hydrographs that result from hydrologic modeling simulations.

Display Options

The display options for a plot window, whether embedded in a dialog or used as a stand alone window for a hydrograph, are accessed by right clicking the mouse in the plot window.



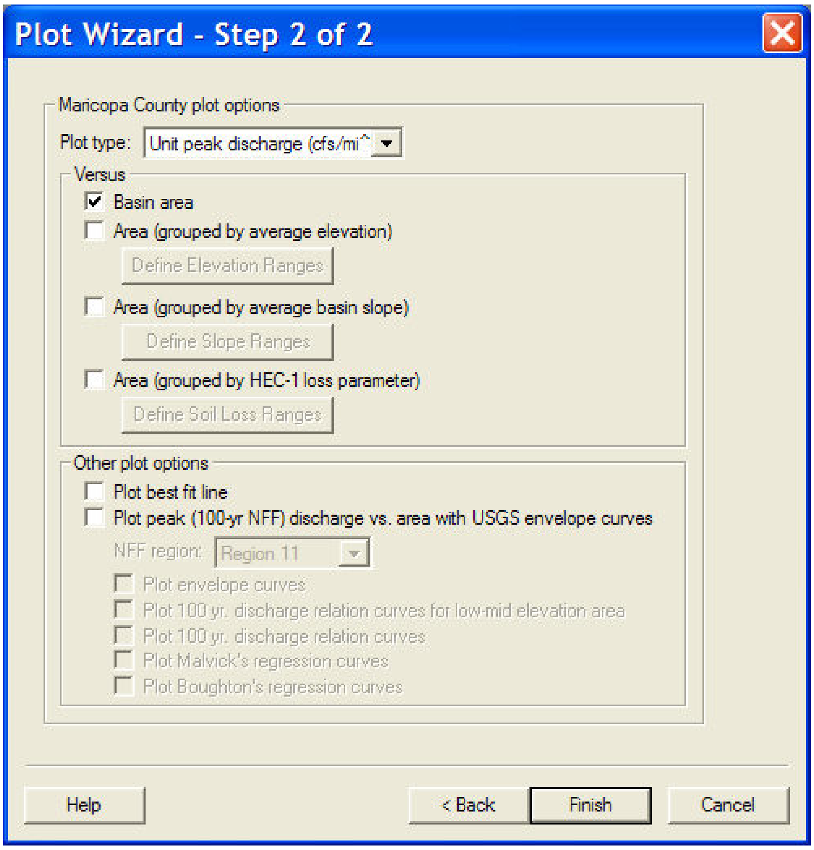
This is a standard menu provided by the plot library used in WMS, but not all of the menus will have effect on the plot. For example the *Plot Data* and *Save Settings* menus will not perform any action within WMS. The **Display Options** command will allow you to control General, Axis, Font, Color, etc. attributes of the plot. Axis Titles can be set, and Hydrograph Units defined. Most useful though may be the **Export/Print** command which will allow you to export your plot data to a spreadsheet, or print the plot directly to a printer. Exporting can be to an image or text file and can be placed in a file or sent to the Windows clipboard for pasting.

Plot Wizard

The Plot Wizard is intended to streamline the preparation for a series of standard plots that can be generated after performing certain types of analysis. The first six plot types are related to the two-dimensional runoff model, GSSHA, where a series of results for the finite difference grid, and observed values exist. At this time they are still under development and will not be discussed in detail here.

Maricopa County Scatter Plot

The Maricopa County Scatter Plot option was created specifically for reports generated by the Maricopa County Flood Control District, but these plots are useful for all users who regularly work with HEC-1. These plots include the ability to plot peak discharges or unit peak discharges (cfs/mi2) against basin areas that can be grouped according to average elevations, slopes, or loss parameters. They can also be compared against several standard regression curves or best fit lines. After selecting the Maricopa County Scatter Plot option from the Plot Wizard the second step shows the following dialog:



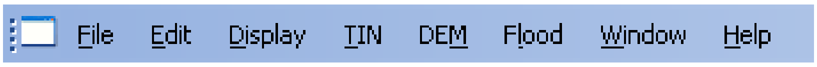
The options for the plot are chosen from this dialog. A scatter point for the plot is created from each hydrograph icon (basins and outlets) in the model and then any other plot option lines are generated for reference.

FHWA Storm Drain HGL Plot Results of a storm drain analysis can also be generated using the Plot Wizard. Hydraulic Grade Line vs. invert and/or ground elevations can be generated from the results of a Storm Drain analysis.

Related Topics

- GSSHA Overview
- Plot Windows

Menu Bar



The commands in WMS are accessed through pull down menus located in the menu bar. Each menu can be accessed with the mouse or by holding down the Alt key and pressing the underlined letter in the menu title. Once a menu is visible the individual commands can be selected with the mouse or by holding down the Alt key and pressing the underlined letter in the menu command.

When the active module is changed, the menus change to a set of menus associated with the selected module. The first four menus (File, Edit, Display, View) are the same for every module, as are the last two (Window, Help). The remaining menus are dependent on the selected module.

If a menu item is dimmed, then it can not be used until the proper conditions exist. Typically this means the appropriate data has not yet been created or selected.

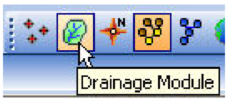
Some menus for creation and manipulation of data objects appear when right-clicking in the Project Explorer.

Related Topics

- WMS Screen
- Project Explorer

Help Strip

There are two ways to get context sensitive help. If you hold your cursor over a button or tool the name will appear in a small pop-up text box below the cursor.



Also The **Help Strip** is at the bottom of the WMS application window and is used for user prompts and to display context sensitive help messages. Some commands require selection in the middle of execution, at these times, WMS chimes and a prompt appears in the Help Strip with specific instructions. Once the instructions have been completed the prompt message is removed. Context sensitive help messages appear in the Help Strip as the cursor is moved over tools, macros, menu items, or dialog items.

Related Topics

- Edit Window

3. General Tools

Toolbars

While the toolbars are automatically updated when switching between modules, the tools available on various palettes can still be made available regardless of the active module. This can be done by selecting a toolbar from **Display | Toolbars**. When this is done, a floating toolbar appears that can be moved or docked. Furthermore, the shape and orientation of these floating toolbars can be adjusted by dragging the handles (sides or corners of the toolbar) when double-headed arrows appear.

Related Topics

- Tool Palettes
- Module Palette
- Macros
- Units Toolbar
- Digitize Toolbar
- Static Tool Palette
- Dynamic Tool Palette
- Drawing Tools
- Get Data Tools
- WMS Screen

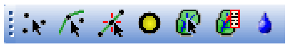
Tool Palettes

Each set of tools within a dynamic tool palette corresponds to a module, or data objects within a module. For example the tools used to create, edit, and select TIN data are different from those used for feature objects or DEMs. You can click on each link to get a further description of the tools in that palette.

Terrain Data Module

Terrain Data 

Drainage Module

Drainage 

Map Module

Map 

Hydrologic Modeling Module

Hydrologic Modeling 

River Module

Hydraulic Modeling 

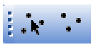
GIS Module

GIS 

2D Grid Module

2D Grid 

2D Scatter Module

Scattered Data 

Module Palette

The *Module Palette* is used to switch between modules. Only one module is active at any given time. However, the data associated with a module (ex. a DEM or TIN in the Terrain module) is preserved when the user switches to a different module. Activating a module simply changes the set of available tools and menu commands.

By changing the default preferences for the Project Explorer the module can also be changed each time a data folder or data object is selected in the *Project Explorer*.

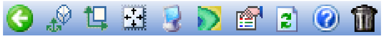
Related Topics

- Modules
- Terrain Data Module
- Drainage Module
- Map Module
- Hydrologic Modeling Module
- Hydraulic Modeling Module
- GIS Module
- 2D Grid Module
- Scatter Point Module

Macros

Standard Windows Macros such as New, Open, Save, and Print appear at the top left of the WMS screen directly under the menus.

Many of the more frequently used menu commands can be accessed through the macro buttons to the right of the *Standard Windows Macros*.



These commands include from left to right above:

View Previous – redraw everything in the graphics window. Selecting the **View | Previous View** command of the *Display*, or the *View Previous* macro  , restores the *Graphics Window* viewing parameters to those in place before the last viewing command was issued (rotate, zoom, pan, etc.). A macro for this command appears in the *Macro Tool* palette.

Perspective View – draw all data in a 3-Dimensional oblique view. Selecting the **Display | View | Oblique View** command, or the *Prespective View* macro  , restores the bearing and dip angles to their previously defined values and causes the image to be viewed from an oblique perspective. The **rotate** tool  can be used to alter the angle of view.

Plan View – draw all data in plan view (looking down from above). Selecting the **Display | View | Plan View** menu command, or the *Plan View macro*  , changes the viewing angles so that the image is displayed such that the user is looking down the z-axis with the x-axis horizontal and the y-axis vertical.

Frame – center and redraw all data in the graphics window. After altering the image display using the Zoom or Pan tools, the image can be centered by selecting the **Frame Image** command in the *View* menu. This command adjusts the window boundaries so that all currently visible objects just fit in the *Graphics Window*. It does not affect the viewing angle.

Display Options – set the display options. See the article Display Options for more information.

Contour Options – set the contour display options. See the article Contour Options for more information.

Properties – accesses the *Dataset Info* dialog

Refresh – redraw everything in the graphics window. When editing the image in the Graphics Window it occasionally becomes necessary to update the display or refresh the screen by redrawing the image. Whenever possible, WMS automatically updates the display. However, in several cases small parts may be obscured by editing procedures, and the display will need to be refreshed by selecting the **Refresh** command from the *Display* menu.

NOTE: The process of redrawing can be aborted in many cases by pressing the *ESC* key.

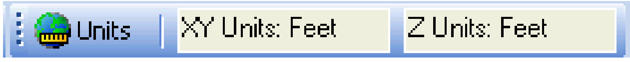
Help – accesses *WMS Help*

Delete – Delete selected objects (must have something selected). The **Delete** command is used to delete any selected objects. This command is also equivalent to hitting the *DELETE* or *BACKSPACE* keys. A macro for the **Delete** command  is found in the *Macro* tool palette.

Related Topics

- Tool Palette
- Menu Bar
- Display Menu

Units Toolbar



The planimetric and elevation units are displayed in the Units toolbar, on the far right of the top toolbars The Units button can be used to access/change the current units and coordinate system definition. The units of the data are important when making area, length, slope, and other geometric calculations. WMS must have the correct units defined for the raw data (DEM, TIN, etc.) in order to make the appropriate conversions to model units such as square miles, kilometers squared (area) and miles, kilometers, feet, meters (distance).

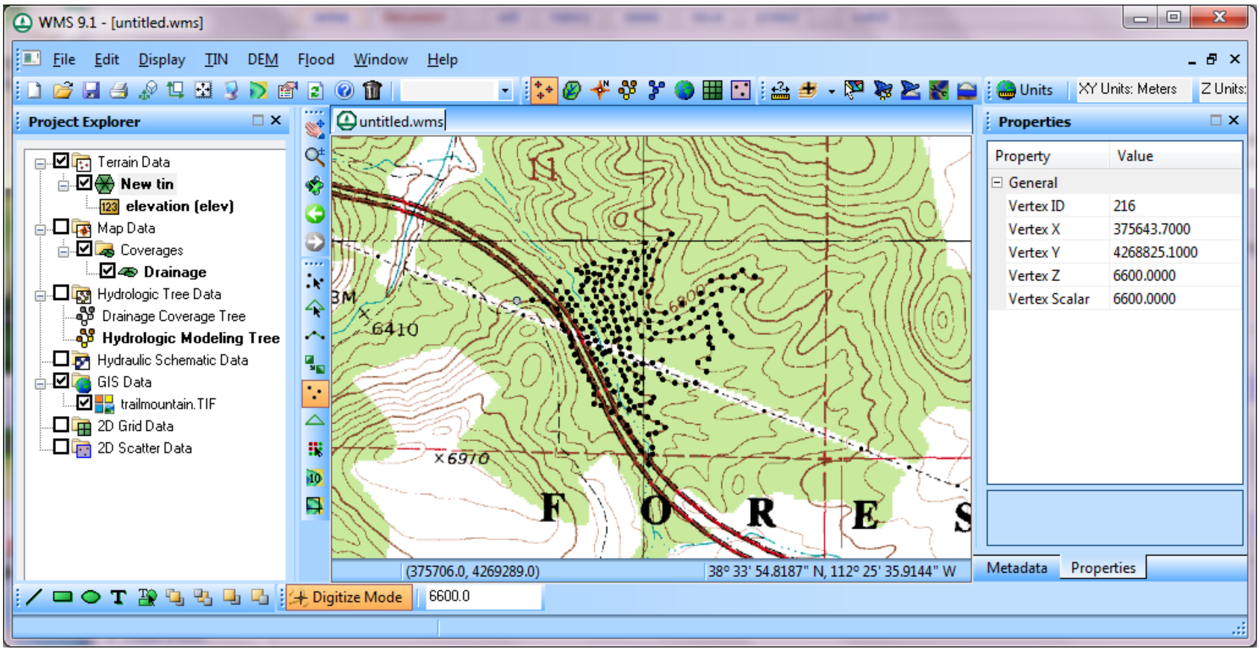
Related Topics

- Units

Digitize Toolbar

The WMS *Digitize Toolbar* provides a quick way to create TIN vertices if you have known elevations and have no source of accurate elevation data other than a digital map.

For example, if you have an image of a contour map with known contour elevations, you can turn on the *Digitize Toolbar* and then turn on **Digitize Mode**. Define an elevation value and then click on points along the contour to define vertices on a TIN, as shown below.



Once you have defined all the vertices on your TIN, you can turn off the digitize mode and triangulate your TIN to use your TIN in your hydrologic or hydraulic model.

Related Topics

- Creating TINs

Static Tool Palette

The tools that are available in every module are located in the *Static Tool Palette*. These tools are used for basic operations such as panning, zooming, and rotating. The static tools are as follows:

Pan Tool

The **Pan** tool is used to pan the viewing area of the *Graphics Window*. When the **Pan** tool is active, clicking the mouse in the *Graphics Window* has the following results:

If a point is clicked, the viewing area is shifted so that the point clicked corresponds to the center of the window.

If the cursor is dragged while holding the mouse button down, the viewing area is shifted to simulate moving the image the direction and distance specified by the line defined while dragging the cursor. The image isn't updated until the mouse button is released.

- **Mouse Click –** by clicking on a location on the screen the entire display will be panned so that the point clicked on is in the center of the graphics window.
- **Mouse Drag –** while clicking and holding the mouse cursor down you can drag the point clicked on to any other location in the graphics window.

Zoom Tool

The viewing area can be magnified/shrunk using the **Zoom** tool. When this tool is active, the following actions can be used to redefine the viewing area of the *Graphics Window*:

A rectangle can be dragged around a portion of the display to zoom in on a particular region. The display is refreshed and the area inside the rectangle is expanded to fill the entire screen.

If a point is clicked, the display is zoomed in around the point by a factor of two.

If a point is clicked while the *SHIFT* key is held down, the display is zoomed out about that point by a factor of two.

- **Zooming In –** To zoom in select the zoom tool and click on the part of the model to zoom in about, or drag a rectangle around the area you wish to fit in the graphics window.
- **Zooming Out –** To zoom out select the zoom tool and hold down the *SHIFT* key (you will notice that the plus sign inside of the zoom tool changes to a minus sign as you move the cursor). Click on the point you wish to zoom out about.
- **Framing –** Sometimes you just want to change the zoom so that everything just fits within the graphics window. You can do this at any time by selecting **Display | Frame Image**, or by selecting the Frame macro .
- **Previous View –** After zooming in/out it is sometimes useful to return the previous view. This can be done by selecting **Display | View | Previous View**, or by selecting the Previous View Macro .

 Rotate Tool

The **Rotate** tool provides a quick way to rotate the image on-screen about the x and z axes. Two rotation methods are available:

Single Update – Holding down the mouse button and dragging the cursor in the *Graphics Window* rotates the object in the direction specified in dynamically A horizontal movement rotates the image about the z axis. A vertical movement rotates the image about the x axis. The amount of rotation depends on the length the cursor moves while the mouse button is down.

Continuous Update – Holding down both the *CTRL* key and the mouse button while dragging the cursor in the *Graphics Window* creates an arrow vector indicating the direction and magnitude of rotation. This is useful because the image is not updated until after the mouse button is released.

Related Topics

- Tool Palette
- Dynamic Tool Palette
- Macros

Dynamic Tool Palette

When the active module is changed, the tools in the *Dynamic Tool Palette* change to the set of tools associated with the selected module. Each module has a separate set of tools.

Selection Tools

Many of the module-specific tools in the dynamic portion of the *Tool Palette* are selection tools (tools used to select objects such as triangles or vertices). For many commands it is necessary to first select some objects before issuing the command. For example, to delete a set of triangles in the Terrain Data module, the **Select Triangles** tool is chosen, the set of triangles to be deleted are selected and the Delete command is selected from the Edit menu.

Most of the selection tools follow a standard selection protocol. Single items can be selected by clicking on the item. With this method, only one item can be selected at a time. When a new item is selected, any other currently selected items are unselected.

In many cases, multiple items need to be selected. If the *SHIFT* key is held down while clicking on individual items, the items are added to the set of selected items. A previously selected item can be unselected by holding down the *SHIF*T key and clicking on it again. This removes the item from the set of selected items without affecting other selected items. Multiple objects can also be selected by dragging a box around the items to be selected, or by choosing the **Select All** or **Select With Polygon** commands from the *Edit* menu.

You can clear the selection list at any time by clicking on a portion of the graphics window where no objects exist. This effectively clears the selection list because whatever is currently selected becomes unselected, and since you click in a location where no objects exist, nothing is placed in the selection list.

Related Topics

- Static Tool Palette
- Tool Palette
- Select All
- Select With Polygon
- Delete

Drawing Tools

The following drawing object tools are in the dynamic portion of the Tool Palette when the Map , Terrain, or Drainage modules are activated. Only one tool is active at any given time.

Contour Labels

The **Contour Label** tool manually places numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contouring options are changed, until they are deleted using the Contour Label Options dialog, or until the Graphics Window is refreshed. Contour labels can also be deleted with this tool by holding down the SHIFT key while clicking on the labels. This tool can only be used when the TIN is in plan view.

Select Drainage Unit or Basin

The **Select Drainage Unit or Basin** tool is used to select basins which can then be either merged together or split. In addition to selecting basins, this tool can be used to select one of the basin icons.

Move Basin Labels

Computed drainage data may be displayed for each basin. However, when there are many basins, the screen can become cluttered with data. The Move Basin Labels tool allows basin data to be placed at a position other than the centroid, which is the default location. When moving a label, click in the desired basin and while holding down the mouse button, drag the cursor to the desired position on the screen and then release the button. An arrow will be drawn from the final position to the point first clicked in the basin.

Flow Path

The Flow Path tool allows the flow paths for specified points to be drawn. When this tool is active, clicking in the graphics window at a location on a TIN, or a DEM after TOPAZ data are computed will cause a flow path to be initiated from that point and followed "downstream" until a pit or local minima is reached, or until the path leaves the TIN/DEM. This tool can be very useful in checking portions of an edited TIN before stream and basin definition is completed.

The length and slope of overland and stream flow is displayed in the help window each time a new path is drawn. This can be helpful in obtaining parameters used to compute lag times with some empirical formulas. Stream distances are shown only after a stream has been created. In other words, channel flow is not counted in the stream distance unless a "stream" has been created along the channel.

Flow paths initiated from the centroid of each triangle or DEM cell can be displayed using the **Draw Flow Patterns** command in the TIN/DEM menu within the drainage module.

Create Text

The Create Text tool is used to create a single line text string. The location clicked on defines where the text string will be placed. After clicking on a location, the Text Attributes dialog appears allowing you to enter the text string and choose the font, color, etc.

Create Rectangle

The Create Rectangle tool is used to create wire frame or filled rectangles. Rectangles can be used to represent buildings, frame a series of text strings, etc.. Rectangles are created with this tool by dragging a rectangle with the mouse at the location on the screen where you wish to place the rectangle. A square can be created by holding down the control key while dragging.

Create Oval

The Create Oval tool can be used to create wire frame or filled ovals. Ovals are created by dragging a rectangle with the mouse at the location on the screen where you wish to place the oval. The rectangle width and height determine the major and minor axes of the oval. A circle can be created by holding down the control key while dragging.

Create Line

The Create Line tool can be used to create single line segments or polylines (a series of connected segments). An arrowhead can be placed on either end of the line. Lines are typically used in conjunction with text strings to highlight key features in a plot. A line is created by clicking on a series of points on the screen with the mouse and double-clicking to end to end the line. The color, line style, and arrowhead options of a line are edited with the Attributes dialog described below.

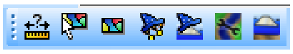
Select Drawing Objects

The Select Drawing Objects tool is used to select previously created text, rectangles, ovals, and lines. Once selected, a drawing object can be moved to another location by clicking on the object and dragging it to a new location. Lines, rectangles, and ovals can be resized by dragging the handles that appear on the corners or ends of the object when the object is selected. The Select Drawing Objects tool is also used to edit the graphical attributes.

Related Topics

- Tool Palettes
- Drawing Objects

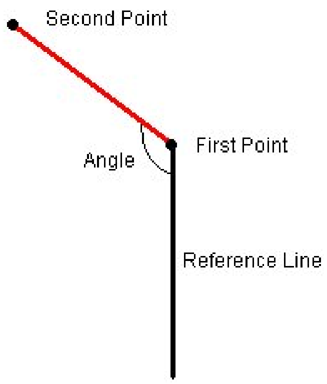
Get Data Toolbar

The *Get Data Toolbar* () is used to get data.
Common problems when beginning a project in WMS include the need to locate the necessary data (e.g. DEMs, Topo Map images, shape files of land use and soils, etc.), make sure the file format is correct, open in WMS, and then insure that all of the data are in a common coordinate system. This can be tedious, especially if many users of the same organization frequently use the same data and therefore repeat this process over and over again on the same set of files. For agencies like flood control districts, departments of transportation, counties, etc. that manage large data for a community of users it is convenient to catalog the necessary files. WMS can read a database catalog and then automatically open any files that fit within a user specified bounding box.

Measure Tool

The **Measure** tool can be used to measure the distance and slope (providing a TIN or DEM is present) along any user defined path. When this tool is active a series of line segments can be defined that make up a path. When the path is terminated by double-clicking, the distance and slope of path are reported in red text in the help message window at the bottom of the screen. Slope is computed as the distanced-weighted average of the slopes of each segment. The slope of each segment is determined by dividing the change in elevation between the beginning point and ending point by the plan (xy) distance of the segment. The units of the distance will be the same as the units of the coordinates (feet or meters) and the slopes are in ft/ft or m/m.

If a line only has two points then an angle (in radians) is also reported. The angle is determined as the angle from a reference line that is in a direction south of the first point, to the line segment defined by the two points. This is illustrated in the following picture.



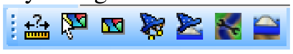
Get Data Tool

The **Get Data** tool is used to get data from the web or from a catalog file.

Common problems when beginning a project in WMS include the need to locate the necessary data (e.g. DEMs, Topo Map images, shape files of land use and soils, etc.), make sure the file format is correct, open in WMS, and then ensure that all of the data are in a common coordinate system. This can be tedious, especially if many users of the same organization frequently use the same data and therefore repeat this process over and over again on the same set of files. For agencies like flood control districts, departments of transportation, counties, etc. that manage large data for a community of users it is convenient to catalog the necessary files. WMS can read a database catalog and then automatically open any files that fit within a user specified bounding box.

See the article Get Data Tool for more on how to use the **Get Data** tool.

Hydrologic Modeling Wizard

The WMS *Hydrologic Modeling Wizard* () is a simple tool that walks you through all the steps involved in creating a hydrologic model. It can be accessed by selecting the hydrologic modeling wizard tool in the *Get Data Toolbar* ().
See the article Hydrologic Modeling Wizard for more information.

Get Data Tool

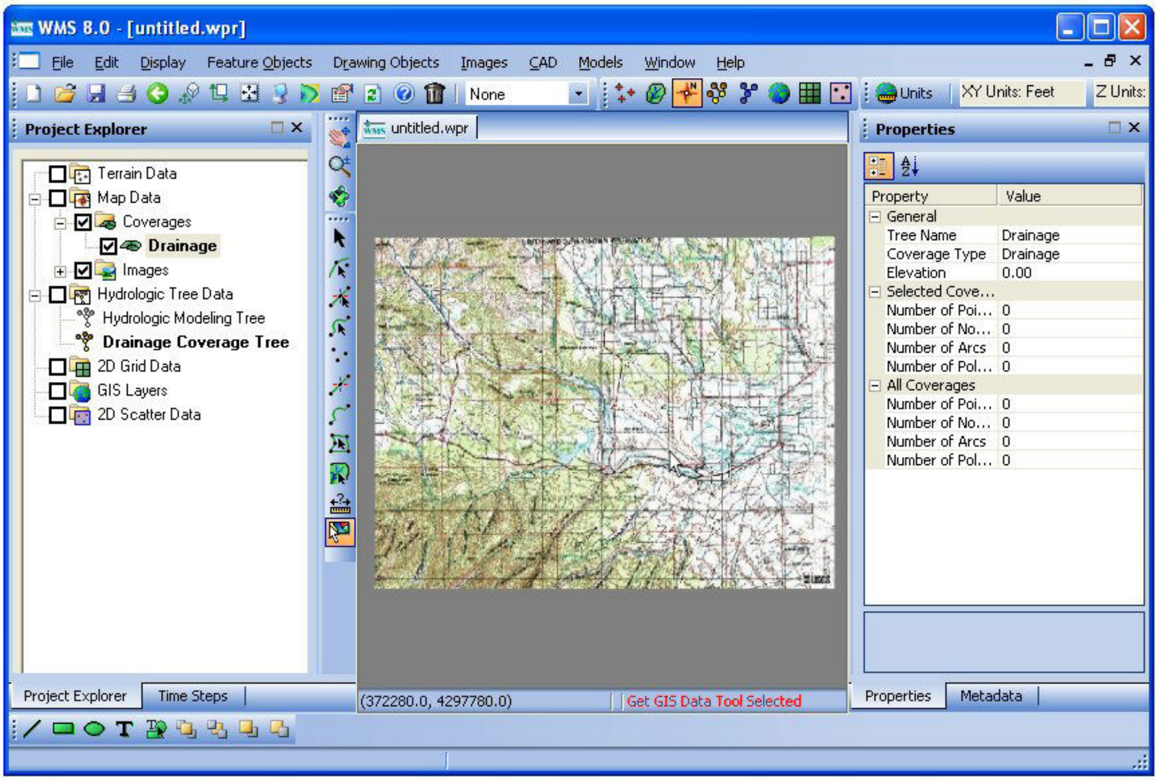
The **Get Data** tool is used to get data from the web or from a catalog file.

Common problems when beginning a project in WMS include the need to locate the necessary data (e.g. DEMs, Topo Map images, shape files of land use and soils, etc.), make sure the file format is correct, open in WMS, and then ensure that all of the data are in a common coordinate system. This can be tedious, especially if many users of the same organization frequently use the same data and therefore repeat this process over and over again on the same set of files. For agencies like flood control districts, departments of transportation, counties, etc. that manage large data for a community of users it is convenient to catalog the necessary files. WMS can read a database catalog and then automatically open any files that fit within a user specified bounding box.

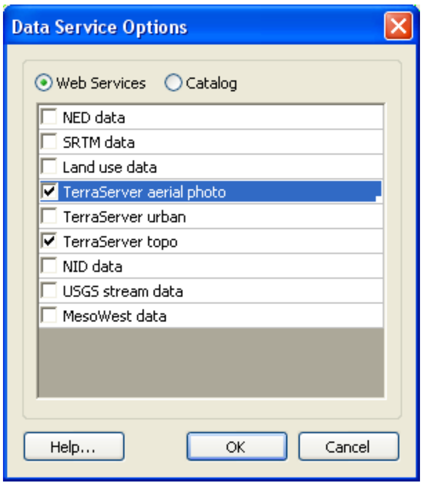
Using the Get Data Tool

The **Get Data** tool can be used to select a geographic area of interest for a modeling project. Available files, as defined by a data catalog file, within the bounding box are opened for use within WMS. The following steps are followed when opening data with the **Get Data** tool:

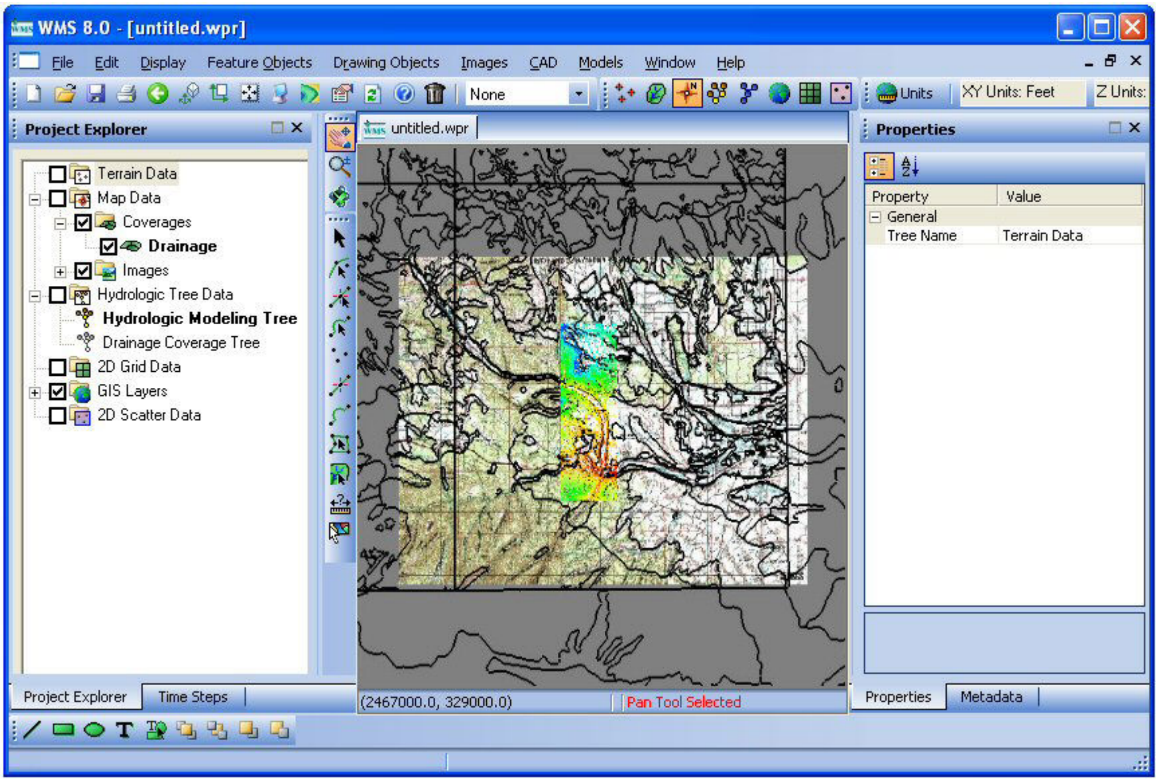
1. Open a georeferenced image file that covers the extents of the data defined within the catalog
2. Select the **Get Data** tool and drag a rectangle in the main graphics area that contains the area for your project



4. Either choose *Web Services* and select the data types desired, or choose the *Catalog* option and select the catalog text file. From the dialog listing available file types, choose those you wish to obtain/open.



The data files whose extents overlap the defined rectangle will then be opened and loaded into WMS.



Creating a Catalog

In order for the **Get Data** tool to work a text file defining the location and extents of each file must be created first and then the data files (DEMs, Images, etc.) archived in a directory or CD/DVD so that it can be accessible to those who will use it. The following shows a small example of what this text file should look like:

```
DEM-30Meter   XMIN    XMAX    YMIN    YMAX
C:\temp\Catalog\DEMs\40110a4.dem  542548.3247 553288.3247 4427875.406 4441795.406
C:\temp\Catalog\DEMs\40110a5.dem  531898.5074 542608.5074 4427845.404 4441735.404
C:\temp\Catalog\DEMs\40110a6.dem  521248.6896 531928.6896 4427785.355 4441675.355

IMAGE-TOPO  XMIN    XMAX    YMIN    YMAX
C:\temp\Catalog\QuadImages\pq1831.tif 542548.3247 553288.3247 4427875.406 4441795.406
C:\temp\Catalog\QuadImages\pq1830.tif 531898.5074 542608.5074 4427845.404 4441735.404
C:\temp\Catalog\QuadImages\pq1829.tif 521248.6896 531928.6896 4427785.355 4441675.355
```

```
IMAGE-PHOTO  XMIN   XMAX   YMIN   YMAX
C:\temp\Catalog\PhotoImages\q1831_83.jpg 542548.3247 553288.3247 4427875.406 4441795.406
C:\temp\Catalog\PhotoImages\q1830_83.jpg 531898.5074 542608.5074 4427845.404 4441735.404
C:\temp\Catalog\PhotoImages\q1829_83.jpg 521248.6896 531928.6896 4427785.355 4441675.355

SHAPEFILE-LANDUSE  XMIN   XMAX   YMIN   YMAX
C:\temp\Catalog\LandUse\salt_lake_city.shp 414534 584057 4428200 4539297

SHAPEFILE-SOILTYPE  XMIN   XMAX   YMIN   YMAX
C:\temp\Catalog\SoilType\soils_Project.shp 226700 674700 4099900 4654900
```

A catalog file is a simple space/comma/tab delimited text file. For each kind of data defined in the file there should be a two-part identifier. The first part of the identifier specifies the file type and the current valid types are:

DEM –

This can be any DEM file type supported by WMS, but within groups you cannot mix file types. You can have two separate groups of DEM files, for example 10 meter and 30 meter resolution, or different formats. Each would have to exist as a group under a separate heading and only one choice at a time can be made within a WMS project.

IMAGE –

This can be any set of geographically referenced images but typically this would be the USGS quadrangle topographic maps and aerial photography.

SHAPEFILE –

This can be any shapefile format. At the present all shapfiles are loaded as GIS layers and not WMS coverages. You will still need to map the desired shapes to WMS feature objects in their proper coverages.

The second part of the identifier is just a description that is used in the catalog services dialog to identify that group of data. For example if you have 10 meter and 30 meter DEMs in the same dialog the second part of the descriptor is used in the combo box that allows you to specify which group you will load for the given project.

The bounding-box coordinates for all files must then be given in four separate fields as defined by the header line.

Creating a Background Image

In order to make the catalog easily accessible a georeferenced base map is useful. Such a map need not (and probably it is best if it does not) contain a lot of detail, but rather enough information that a modeler can get a general sense of where a potential modeling area lies.

Related Topics

- Get Data Toolbar

Setting up Film Loops

One of the most powerful 2D visualization tools in WMS is animation. An animation sequence can be generated for a grid with a transient dataset to illustrate how contours, vary as a function of time. Each frame of the animation is stored as an image as part of an AVI file. The entire set of frames in an animation sequence is referred to as a film loop.

Animation film loops are generated by selecting the **Film Loop** command in the **Data** menu of the 2D Grid module. This command brings up the *Film Loop* dialog. The *Film Loop* dialog is used to control the playback of film loops. A new film loop can be generated by selecting the *Setup* button. Once a film loop has been generated, it can be saved to an AVI file using the Save button. Previously saved film loops can be read from disk using the Read button (they can also be run using any AVI playing software, or included in presentation software documents).

Dataset

Film loops are always generated using the active dataset. The *Scalar Dataset* button at the top of the dialog can be used to change the active scalar dataset. The current active dataset is displayed to the right of this button.

Display Mode

The display mode is used to control whether each frame is generated as a wire frame image or a shaded image using the current shading options.

Image Size

By default, each frame that is generated in a film loop occupies the entire *Graphics Window*. This results in film loops composed of large images which require a significant amount of memory and which are difficult to playback at a high speed. To reduce the size of the film loop, the individual frames can be generated at a specified fraction of the default size. The memory required for a film loop is quadratically proportional to the fractional size. For example, an image generated at 50% of the Graphics Window size requires 25% as much memory as an image generated at full size.

Transient Animation

Transient animation can be used with 2D grids and a transient dataset. As each frame is generated, a set of values corresponding to the current time is loaded into memory and the image is redrawn using the current display options. Thus, if the contour display option is selected, the contours will vary from frame to frame.

The strip on the right of the transient animation section of the Film Loop Setup dialog is used to specify what range of the available time steps are to be used for animation. The range of time steps can also be entered directly in the edit fields below the time step strip. The range displayed in the strip corresponds to the scalar data set.

The total number of frames generated in the film loop can be defined by either matching the time steps (one frame per time step) or by using a constant interval (e.g., one frame for every two hour interval). If the Match Time Steps option is chosen, extra frames can be created between each time step if necessary using linear interpolation of the data values at the specified time steps.

Saving Film Loops

Saving and reading film loops is useful since some film loops may take a significant amount of time to generate depending on the complexity of the image. The film loops are saved to disk in a compressed AVI format.

Film Loop Playback

Once a new AVI film loop has been generated or an AVI film loop has been read from disk, several options are available for playing back the film loop. The buttons at the upper left of the *Film Loop* dialog are designed to mimic the buttons on a VCR or CD player. The **Play** button causes the film loop to cycle continuously. The **Stop** button halts the playback. The **Step** buttons can be used to advance the film loop forward or backward one frame at a time. In addition, the frame scroll bar can be used to interactively move the frames forward or backward.

The speed of playback can be adjusted using the **Speed** scroll bar. The maximum speed depends on the speed of the computer and the size of the image being animated. The smaller the image, the faster the maximum playback speed.

Two options are available for cycling the film loop playback. The continuous playback option starts a new cycle at the first frame in the loop after the last frame is encountered. The oscillation option plays the loop in the forward direction to the end of the loop and then in the reverse direction back to the beginning of the loop.

3.1. Menus

File Menu

The **File** menu is one of the standard menus and is available in all of the modules. The commands in the **File** menu are used for file input and output for the WMS file types, for printing, and to exit the program.

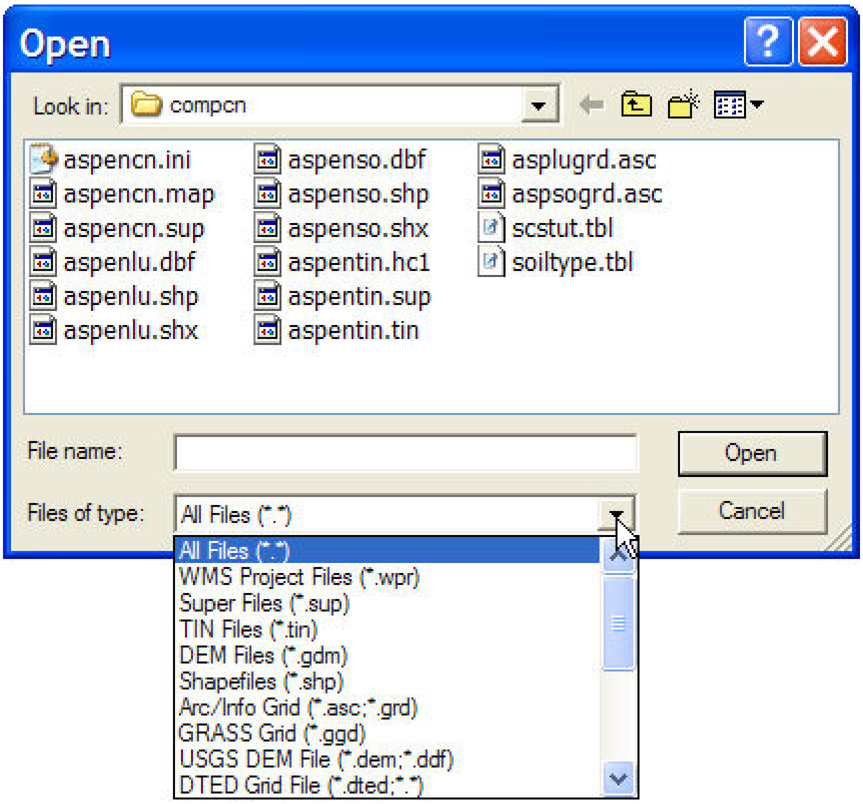
The following commands are contained in the **File** menu:

New

The **New** command (*Ctrl+N*), or *New* macro , deletes all data associated with all data types and all modules. As a result, a warning message asking if you want to save changes to the project will appear, giving a last chance to save. This command should be selected when an entirely new modeling project is started. The default settings are not restored so if you wish to restore all of the settings you should exit WMS completely and restart.

Open

The **Open** command (Ctrl+O), or Open macro , is used to read all files. Unlike previous versions of WMS where the Open command was used to open WMS native file types and the Import command was used to open non-native files, in version 7.0 all files are opened with the Open command where the file filter in the "**Files of type:**" drop down indicates the type of file to be opened.



While any file can be opened from any module, the extensions listed tend to relate primarily to the active module. For example when in the terrain module you will see the extensions for .tin and .dem, but they do not appear in the map and other modules.

If a project file is selected, all of the files listed in the project file will be read in.

The five most recently opened files are listed at that bottom of the File menu. Selecting a file from this list will reopen it.

Drag and Drop

You can drag any WMS supported file from a Windows Explorer to the main window in WMS and it will open, or initiate the specific Import dialog such as DEMs, or shapefiles.

Import from Database

The *Database Import Wizard* allows users to access data stored in a database and import it. The wizard is invoked by selecting the *Import from Database* command from the *File* menu. The data types that can be imported by the *Database Import Wizard* are the same data types that are supported by the Text Import Wizard. Like the *Text Import Wizard*, database data must be formatted in columns to be imported.

The *Database Import Wizard* has four steps:

Step 4 is identical to the last step in the *Text Import Wizard*.

Step 1 – Connecting to a Database

Step 1 of the wizard lets you set up a connection to a database. To connect to a database either on the same computer or on a network press the **Connect to Database** button. Once a database connection is created, a path to the database and the different tables in the database are displayed. When a table is selected its columns are displayed along with the number of rows in that table.

Step 2 – Querying Information from a Database Table

Step 2 allows you to create, copy, delete, and import queries that retrieve data from a database. To help in writing the queries, the tables in the database are displayed, as well as the columns in the selected table.

Some SQL Basics

The query is an SQL (Structured Query Language) statement. The SQL statement is entered in the *Query SQL statement* edit field. An example of an SQL statement would be: “SELECT x, y, z, toluene FROM multipledatasets”. This statement means that columns x, y, z, and toluene from the table multipledatasets will be retrieved from a database. SQL statements are case sensitive. SQL statements also require brackets around table or column names that have spaces. For example, to query data from a table titled “x coordinate” in the SQL statement it would be written as [x coordinate]. A full explanation of SQL is beyond the scope of this document.

You can write a short description for each query in the *Query description* edit field. The **New** button creates a new query that has a default name, description, and SQL statement. The **Copy** button creates a copy of the currently selected query. The **Delete** button deletes the currently selected query.

The queries you define are saved automatically by WMS in a file called wmsquery.ini, located in the folder where WMS is installed. The **Import** button allows you to import a list of queries from any file that follows the same format as the wmsquery.ini file.

Step 3 – Viewing the Results of the Query

Step 3 displays the results of the database query. Only the first 20 rows are displayed in a spreadsheet. If the results are not what you wanted, you can go back to Step 2 and modify the query.

Step 4 – Assigning Column Types

Step 4 is identical to Step 2 of the Text Import Wizard.

Save

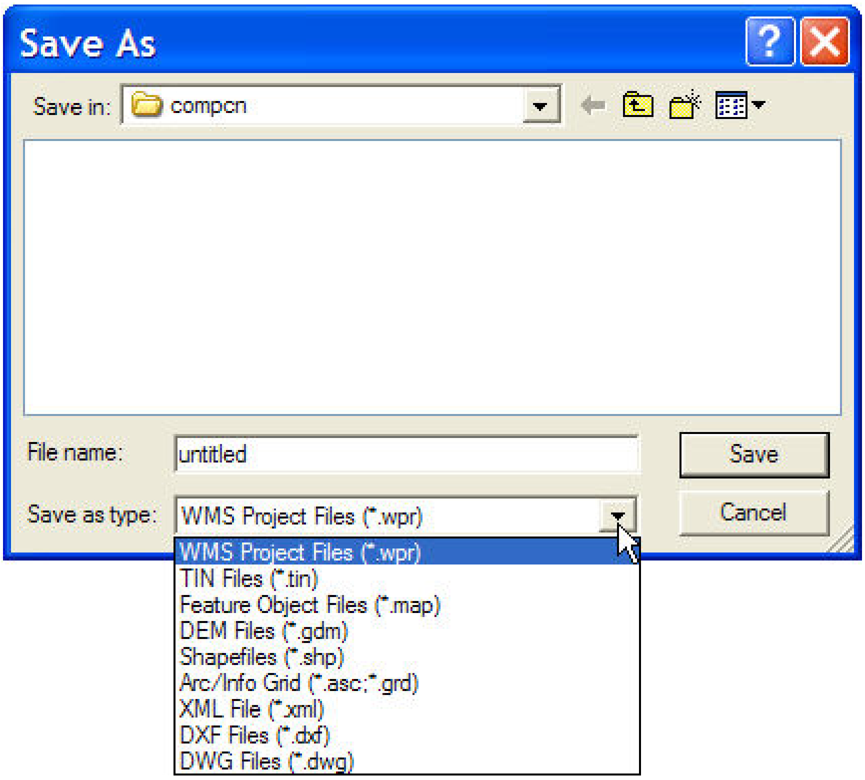
The **Save** command (*Ctrl+S*),or *Save* macro , is used to save WMS projects. A project contains all of the files associated with a modeling project.

When a WMS project is saved, all files associated with the data currently in memory are saved. This includes any model simulations which are open. The model simulations are saved to the path most recently specified using the **Save As** command in the model interface menu.

Save As

The **Save As** command is used to designate the path and prefix file name for saving a WMS project. It can also be used to save a single file as specified by the extension indicated in the "Save as type:" drop down combo.

Previous versions had a separate command for exporting files (saving files in formats not native to WMS), but exporting is now handled by specifying the appropriate extension in the *Save As* dialog.



Publish to Web

The **Publish to Web** command is a specialized command for the US Army Corps of Engineers that allows them to share results of models by publishing a graphic of the current window along with some metadata explaining the project. Alternatively the WMS project files can be zipped and uploaded with the published content. This command first requires the user to authenticate to a web service with CDF capabilities. After successful authentication you can specify the metadata (description) and choose whether or not to upload the WMS screen capture and/or the project data files.

Print

Printed copies of WMS window displays can be generated by using the **Print** command. WMS will print to any printer supported by Windows.

When the **Print** command is selected the standard Windows *Print* dialog comes up allowing you to pick your printer, paper orientation and the typical printing parameters.

If you wish to control the size and location of the print on the paper then you should access the Page Layout command from the *File* menu.

Printer Setup

The **Printer Setup** command allows you to control the orientation of the printed image on the sheet of paper and the paper size. For Windows versions the standard *Printer Setup* dialog for the currently selected printer device will be brought up which allows you to change other relevant parameters of the currently selected printer.

Page Setup

The **Page Setup** dialog allows you to change the size and position of the printed image on the paper. It also allows you to specify margins, paper size, and orientation.

The image size is controlled by specifying the model units for an inch (or centimeter) of the paper. The scale toggle determines whether or not a scale legend is printed. The text color and font of the scale legend can be selected by clicking on the colored rectangle to the left of the toggle. If the Maintain Aspect Ratio toggle is not on then a scale legend cannot be printed.

Edit File

The **Edit File** command allows you to examine (or edit) any text file from within WMS. This command is particularly useful if errors occur while running a simulation using one of the hydrologic models supported by WMS. If a model does not run to a successful completion, errors can usually be found by examining the ASCII output file.

You will be prompted for the name of a text editor (Notepad is the default) and then the file is brought up in the specified editor. You can use the Find Other button to locate a different word processing .exe file and the "Never ask this again" toggle can be set so that you are not prompted for the word processing program each time.

Register WMS

In WMS, components (modules, interfaces) can be licensed individually depending on the needs and interests of the user. The components of WMS are licensed using a password system. The **Register WMS** command is used to enter a password that enables the licensed components. This command can be used to enable the program after initially installing WMS, or for adding additional modules to the program at a later time.

Exit

The **Exit** command (*Ctrl+X*) will exit WMS and terminate the program. Any data not yet saved to files will be lost and so you will be prompted before exiting to confirm that you have saved your most recent edits.

Related Topics

- Edit Menu
- Display Menu

Edit Menu

The **Edit** menu is one of the standard menus and is available in all of the modules. The commands in the **Edit** menu are used to select objects, delete objects, and set basic object and material attributes.

The **Edit** menu contains the following commands:

Delete

The **Delete** command is used to delete any selected objects. This command is also equivalent to hitting the *DELETE* or *BACKSPACE* keys. A macro for the **Delete** command  is found in the Macro tool palette.

Confirm Deletions

Whenever a set of selected objects is about to be deleted, the user is prompted to confirm the deletion. This is meant to ensure that objects are not deleted accidentally. Selecting the **Confirm Deletions** item from the *General* tab of the *Preferences* dialog (accessed by the **Preferences** command of the **Edit** menu) can turn this option off. The check mark in front of the command is present when this option is turned off and is not when it is turned off.

Delete All

The **Delete All** command is used to delete all of the data associated with the active module, whether or not there are selections. It is similar to the **New** command in the **File** menu except that the **New** command deletes all data in all modules.

Select All

The **Select All** command (*Ctrl+A*) selects all items associated with the current selection tool, providing that the tool supports the **Select All** option.

Select With Polygon

The **Select With Polygon** command allows you to enter an irregular polygon enclosing the items to be selected (one of the selection tools must be active). To enter the polygon, click on both the polygon's starting point and each intermediate point defining the polygon and double click on the ending point. All items within the polygon will be selected. If an error occurs while entering a polygon, the following keys can be used:

- *BACKSPACE* or *DELETE* – Back up one line segment.
- *ESC* – Abort entering the polygon, and selection by polygon.
- *CONTROL* – Holding the *CONTROL* key down while moving the cursor causes all previously entered segments of the polygon to be moved simultaneously.

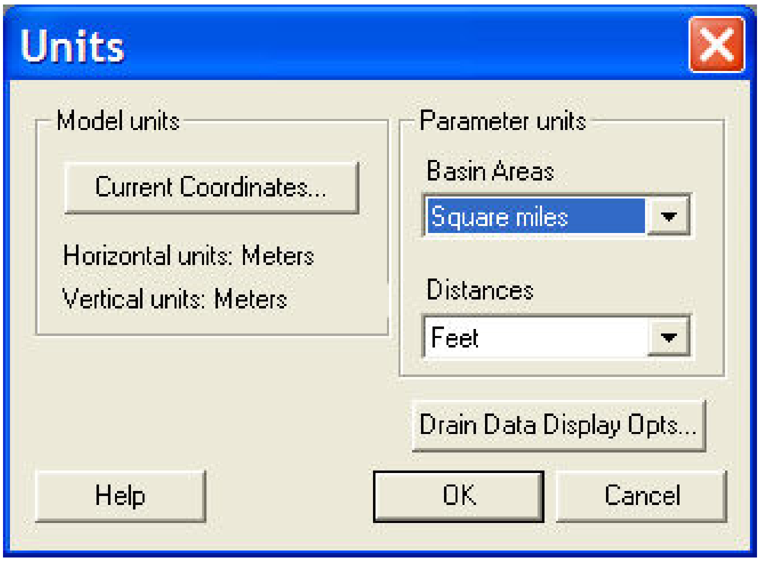
In addition to defining a polygon using the cursor, a polygon created in the Map module may be chosen to define a selection polygon.

Units

Units are determined by the data that you read into WMS. For example if you read a TIN in which has coordinate vertices in meters then you are responsible to tell WMS that you are working with metric units so that proper conversions are made when computing areas, slopes, and other geometric values.

Units are specified in the *Units* dialog. Model units are the units of the geometric data (i.e. TIN, DEM, Feature Objects, etc.) you are using to develop hydrologic and hydraulic models whereas parameter units are the units you wish to have areas and distanced converted to when performing calculations. **IMPORTANT:** WMS requires that the Horizontal and Vertical units be the same in order to correctly estimate slopes.

Units are closely related to the Current Coordinate System and so access to this dialog is given in order to change the current units setting.



Current Coordinates

See Coordinate Systems for more information.

Coordinate Conversion

Converts data from one coordinate system to another. See Coordinate Conversions for more information.

Single Point Conversion

The **Single Point Conversion** dialog allows a user to convert a coordinate (xyz point) between to coordinate systems. This dialog can be accessed from two locations within XMS programs:

1. *Edit* menu, *Convert Single Coordinate* item
2. Register Image Dialog

The dialog consists of two sides. On the left side, the original (or Convert from) coordinate system is entered. On the right side, the final (or Convert to) coordinate system is entered. The options for the two sides are described on the Coordinate Systems page.

The other items in the dialog include:

- **Enter coordinates –** Enter the original (or from) coordinates.
- **New Coordinates –** View the final (or to) coordinates.
- **Convert –** Perform the conversion.
- **Create Mesh Node/Feature Point –** Create a mesh node (if in the mesh module) or a feature point (if in the map module) at the final coordinates when OK is pushed.

Materials

When doing 1D Hydraulic modeling in WMS it is useful to define material types. Cross section segments have material ID's associated with them and relate to the Materials defined with this command. These material types often represent different types of bed material or areas of fluid properties. A global list of material attributes is maintained that can be edited using the **Materials Data** command. The command brings up the *Materials Data* dialog where each material is assigned an ID number. This dialog can be used to delete unused materials, create new materials, and assign a descriptive name, color, and pattern to a material. This general information is saved in the material file.

The Legend toggle controls the display of a legend of the materials in the *Graphics Window*. The options for the legend are edited in the *Legend Options* dialog. These options include:

- The name to be displayed on the legend.
- The font to be used in the legend.
- The specification of which corner of the screen the legend will appear in.
- The size of each entry in the legend.
- Whether all materials or active materials will be included in the legend.

When a new cross section is created, a user specified default material is assigned to the new section.

Material ID's can be assigned to polygons of an Area coverage and used to map to cross section segments when automatically extracting them from TINs.

Model specific material properties such as Manning's n are edited using commands local to the model menu.

Copy to Clipboard

The contents of the Graphics or Hydrograph windows can be saved to the Windows clipboard by selecting the **Copy to Clipboard** command (Ctrl+C). The clipboard can then be “pasted” into other Windows programs, such as word processors or drawing packages.

Other dialogs and plot windows have separate commands available for copying text results or plots to the clipboard for inclusion in other documents.

Paste

The **Paste** command in the *Edit* menu allows the user to paste tabular data (generally this is xyz data used to create a TIN or scattered data set) directly into WMS. The tabular data can be copied from a text file or from a spreadsheet and then pasted into WMS.

When this command is executed the Text Import Wizard is launched.

Related Topics

- File Menu
- Display Menu

Display Menu

The *Display* menu is one of the standard menus and is available in all of the modules. The commands in the *Display* menu are used to control how attributes of a TIN, DEM, Feature Objects, etc., are being displayed, to set up a drawing grid, to control how contours are displayed, and to generate shaded images. The following commands are found in the *Display* Menu

- Display Options
- Contour Options
- Grid Options
- **Refresh** – When editing the image in the Graphics Window it occasionally becomes necessary to update the display or refresh the screen by redrawing the image. Whenever possible, WMS automatically updates the display. However, in several cases small parts may be obscured by editing procedures, and the display will need to be refreshed by selecting the **Refresh** command from the *Display* menu.

  NOTE: The process of redrawing can be aborted in many cases by pressing the *ESC* key.
- **Frame Image** – After altering the image display using the Zoom or Pan tools, the image can be centered by selecting the **Frame Image** command in the *View* menu. This command adjusts the window boundaries so that all currently visible objects just fit in the *Graphics Window*. It does not affect the viewing angle.
- **View**
  - **View Angle** – The objects in the *Graphics Window* can be rotated and viewed in three dimensions. Two angles, bearing and dip, are used to rotate the view. The bearing and dip values correspond to a rotation about the z and x axes. The bearing affects the horizontal angle (rotating the object in the xy plane), and the dip changes the vertical angle (shifting the viewing angle on the object to a higher or lower perspective). The object cannot be tilted sideways. Using only two viewing angles rather than three limits the viewing angles, but it is simpler and more intuitive. The bearing and dip angles can be explicitly defined in the *View Angle* dialog accessed by selecting the **View Angle** command from the *View* menu. The viewing angles can be manipulated interactively with the Rotate tool.
  - **Set Window Bounds** – The region of the real world coordinate system that is mapped to the *Graphics Window* can be altered using the Pan and Zoom tools. It is also possible to precisely control the visible region by selecting the **View | Set Window Bounds** command from the *Display* menu. The *Set Window Bounds* dialog box appears and the x and y limits of the viewing area can be set.

    If the **X range** to be specified (preserves aspect ratio) option is selected, the x coordinate at the left and right and the y coordinate at the bottom of the *Graphics Window* are specified. The y coordinate at the top of the *Graphics Window* is not specified in order to maintain the aspect ratio.

    If the **Y range** to be specified (preserves aspect ratio) option is selected, the y coordinate at the top and bottom and the x coordinate at the left of the *Graphics Window* are specified. The x coordinate at the right of the Graphics Window is not specified in order to maintain the aspect ratio.

    If the **X** and **Y range** to be specified (alters aspect ratio) option is selected, the x coordinate at the right and left and y coordinate at the top and bottom of the *Graphics Window* are specified. Since all four coordinates are specified, the aspect ratio of the scene may be altered.
  - **Plan View** – Selecting the **Display | View | Plan View** menu command, or the *Plan View macro*  , changes the viewing angles so that the image is displayed such that the user is looking down the z-axis with the x-axis horizontal and the y-axis vertical.
  - **Front View**
  - **Side View**

  - **Oblique View –** Selecting the **Display | View | Oblique View** command, or the *Prespective View* macro , restores the bearing and dip angles to their previously defined values and causes the image to be viewed from an oblique perspective. The rotate tool can be used to alter the angle of view.
  - **Previous View –** Selecting the **View | Previous View** command of the *Display*, or the *View Previous* macro , restores the *Graphics Window* viewing parameters to those in place before the last viewing command was issued (rotate, zoom, pan, etc.). A macro for this command appears in the Macro Tool palette.
  - **Z Magnification –** Occasionally an object may be very long and wide with respect to its overall depth (z dimension). It is possible to exaggerate the z scale so that the variation in the z value is more apparent by selecting **View | Z Magnification** from the *Display* menu and changing the magnification factor to something larger than the default of 1.0.
- Open Hydrograph Plot
- Plot Wizard

Related Topics

- File Menu
- Edit Menu

Properties Window

The *Properties Window* provides both information and opportunities to edit properties of WMS' entities.

Properties Window Overview

The properties of the currently selected entity are shown in the *Properties Window*. If more than one entity is selected, some properties that show the relationship between the entities or the number of entities selected may be shown in the *Properties Window*. Sometimes, selecting one entity will also select a related entity. For example, if you select a feature polygon representing a basin boundary, WMS will also select the associated basin in the hydrologic tree.

When you select a transient (time-varying) dataset in the *project explorer*, all the time steps associated with the transient dataset are shown in the *Properties Window*. You can select the time step of interest or click through the time steps to visualize contours of data values at each selected time step.

The following list describes the entities and their properties shown in the WMS *Properties Window*:

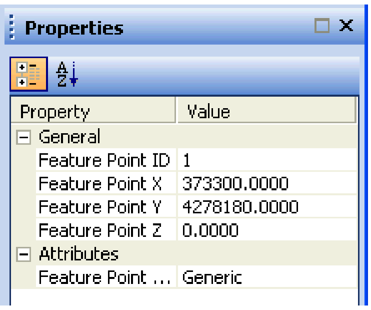
1. Selected tree items
   1. Name
   2. Various other properties
2. Selected DEM cell properties
   1. I value
   2. J value
   3. DEM cell elevation
3. Feature Arcs
   1. ID
   2. XYZ Length
   3. Plan (XY) Length
   4. Arc Type
   5. Arc Basin ID

6. Basin Name
7. Basin Area
8. Cross Section Attributes (for cross section arcs)

4. Feature Polygons
   1. Polygon ID
   2. Polygon Area
   3. Polygon Type
   4. Polygon Basin ID
   5. Basin Name
   6. Basin Area
5. Feature Points/Nodes
   1. ID
   2. X Coordinate
   3. Y Coordinate
   4. Z Coordinate
   5. Point Type
   6. HY-8 Crossing Name (if applicable)
6. TIN/2D Scatter Point Vertices
   1. ID
   2. X Coordinate
   3. Y Coordinate
   4. Z Coordinate
   5. Scalar Value
7. TIN Triangles
   1. Number (ID)
   2. Area
8. Hydrologic Tree Outlet Points
   1. Outlet Name
   2. Upstream Area
   3. Downstream Slope
   4. Downstream Length
   5. Tree X Coordinate
   6. Tree Y Coordinate
   7. Tree Z Coordinate
9. Hydrologic Tree Basins
   1. Model-specific Hydrologic Properties
   2. Area
   3. Other Geometric Properties
10. Hydrologic Diversions
    1. Diversion Name
11. Hydrographs
    1. Peak Flowrate
    2. Time of Peak
    3. Volume
12. Hydraulic Reaches

   1. River Name
   2. Reach ID
   3. Reach Name
   4. Computational Feature Length
   5. Measured Length
   6. Start Station
   7. End Station
   8. Hydraulic Schematic X Location
   9. Hydraulic Schematic Y Location
   10. Hydraulic Schematic Z Location
13. GIS Shapes
   1. Layer Name
   2. Layer Type
   3. Filename
   4. Extents (Left, right, top, and bottom)
   5. Listing of the fields for the selected shape and their values
14. 2D Grid Cells
   1. I Value
   2. J Value
   3. Scalar Value/Index Map ID

Editing Coordinates and Other Values



Coordinate edit boxes, located on the right side of the WMS window, are used to edit the coordinates of a selected TIN vertex, feature point, or scatter point. Coordinates are changed by typing in new values and hitting the *ENTER* or *TAB* key. Other values, such as 2D grid dataset values, can also be edited in the properties window by changing the value and hitting the *ENTER* or *TAB* key to accept the entered value.

Metadata Window

Metadata, or data about data, is accessible by choosing the *Metadata* tab at the bottom of the *Properties Window*. This window will display information the currently selected object in the WMS Project Explorer. This data may include when the file was created or modified, contact information, horizontal and vertical coordinate system information, or other object specific data.

3.2. Coordinate Systems

Coordinate Systems

XMS programs work in a single coordinate system. The user selects what system will be used using the *Edit | Coordinate System* menu command.

Both a horizontal and vertical system are specified. Many numerical models work in a local system, so there may not be a problem if the user does not know what coordinate system the data is referenced to. If the desired model requires a global system, or the base data is referenced to more than one system, the modeler must determine the coordinate systems involved.

Since data can be gathered and referenced to various coordinate systems, XMS allows the user to convert data from one coordinate system into another. The Coordinate Conversions page describes how to do this. The available systems and units include:

Horizontal System

The options to the right of the dialog change as the horizontal system is changed. For example, the hemisphere is required for a Geographic system and a UTM zone is required for a UTM system.

- Local
- Geographic
- UTM
- State Plane
- CPP

Local is the default horizontal and vertical system. Unless the user desires to convert data to another system, it is not necessary to change the system from *Local*.

Horizontal Units

The horizontal units can be specified for all systems except Geographic, which uses decimal degrees. **The units set here are the units used by the models such as RMA2**.

- U.S. Survey Feet
- International Feet
- Meters

Ellipsoid

The Ellipsoid options are only available for non-NAD/HPGN/CPP systems. The ellipsoid can be selected for a region of the world. Changing the ellipsoid changes the minor and major radii of the earth, measurements used in performing conversions. To set the radii for an ellipsoid not included in XMS, select the *User Defined* ellipsoid and a button appears titled *Define*. Clicking on the button brings up a dialog where the radii are defined. The Major and Minor Radii (a and b, respectively) are defined in meters only. The Minor Radius can be input directly or defined by specifying the ellipsoid flattening (1/f) or eccentricity squared (e2) variables, where

$$\frac{1}{f} = \frac{(a-b)}{a} \quad \text{and} \quad e^2 = 1 - \frac{b^2}{a^2}$$

Vertical System

The supported vertical options are for North America. If a non-NAD/HPGN horizontal system is selected, this option is disabled.

- Local
- NGVD 29
- NAVD 88

Vertical Units

The vertical unit is only used for conversions. Area and volume calculations performed by XMS use the horizontal units.

- U.S. Survey Feet
- International Feet
- Meters

External Links

- Introduction to Tidal and Geodetic Vertical Datums and Datum Computations [1] – Presentations available here [2]

References

[1] http://www.ngs.noaa.gov/corbin/class_description/Tidal_Geodetic_Datums.shtml
[2] ftp://ftp.ngs.noaa.gov/pub/corbin/datum_training/

Coordinate Conversions

Converting data from one coordinate system to another can be done using the ***Edit|Coordinate Conversions*** menu command. The command may also be accessed by right-clicking on a single entity (grid, mesh, scattered data set ...) in the Project Explorer. This allows the user to convert just that entity from one system into another (typically to the current project coordinate system).

All data will be converted from the system on the left of the dialog to the system on the right.

"Convert From" System

The "Convert From" system defines the coordinate system the data is currently referenced to. When working from the *Edit* menu, this is the system XMS is working in and is dimmed by default because it is assumed you have already specified this system. When working from the Project Explorer, this is the coordinate system of the selected entity and must be selected.

"Convert To" System

The "Convert To" system defines the system you will be working in after the conversion. When the dialog is invoked from the *Edit* menu, this is selected by the user and all data is converted from the current system to this new system. When the dialog is invoked from the Project Explorer, the "Convert To" system is dimmed because this is assumed to be the system XMS is working in and all other data is already in this system.

Restrictions

Some conversions are not allowed, such as converting between a NAD and non-NAD system. A warning is issued when conversions are not allowed.

Projections

Related Versions	
GMS	v7.0
version note	

"Projection" refers to a map projection like UTM [1]. In XMS software, a projection can be associated with a project, and data can be reprojected from one projection to another. XMS software utilizes the Global Mapper (TM) [2] library which supports hundreds of standard projections.

Previous XMS software versions referred to projections as "coordinate systems" and reprojection as "coordinate conversion".

Project Projection

The current projection, or the projection currently associated with the project, can be specified via the *Edit|Projection* menu command. Changing the projection does not alter the XYZ coordinates of the project data.

Local Projection

Many numerical models work in local systems, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). XMS software allows for local projections that are unrelated to any standard projection.

Reproject

Reprojecting means to convert data from one coordinate system to another. For example, a 2D mesh representing the ground surface may have XYZ coordinates in a UTM system and they need to be converted to a State Plane system to be consistent with other data. Reprojecting usually results in the XYZ coordinates of the data changing, although conceptually the data is in the same place with respect to the Earth, just in a different coordinate system.

There are three basic reprojection tasks that you can do:

- Reprojecting the entire project from one system to another
- Reprojecting one object from one coordinate system to the project coordinate system
- Single point reprojection, which allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection.

Either operation brings up a dialog with two projections specified. On the left, the "Convert From" projection defines the projection the data is currently in. On the right, the "Convert To" projection defines the projection the data will be in after the operation.

Reproject everything

Reprojecting everything can be done by selecting the ***Edit|Reproject*** menu command. This will convert all the data loaded into the XMS application from one projection to another. In this operation, the "Convert From" projection (left side of the dialog) is dimmed by default. The user selects a new projection on the right side that all data in the application will be converted to. This operation changes the "current" projection that is being used by XMS to the newly specified "Convert To" projection.

Reproject object

This command is done on a specific geometric object (grid, mesh, scatter set, ...) by right-clicking on the entity in the Project Explorer. The object is reprojected from some projection into the current projection being used by the XMS application. In this operation, the "Convert To" projection (right side of the dialog) is dimmed by default because it represents the current projection used by the project. It is assumed that the desire is to reproject data from another projection into the current projection. The current projection is not changed in this operation.

Single Point Reprojection

Single Point Reprojection allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection. It also lets you create a feature point at the new location. This operation is accessed via the *Edit|Single Point Reprojection* menu command. It's also available in the Register Image dialog.

Restrictions

Some reprojections are not allowed, such as reprojecting between a NAD and non-NAD system. A warning is issued when the reprojection is not allowed.

Supported Projections

XMS software utilizes the Global Mapper (TM) [2] library which supports hundreds of standard projections.

References

[1] http://en.wikipedia.org/wiki/Universal_Transverse_Mercator_coordinate_system
[2] http://www.globalmapper.com/

CPP Coordinate System

A CPP (Carte Parallelo-Grammatique Projection) system is a local system. The origin of the system must be defined in latitude/longitude decimal degrees.

The conversion from of a point from latitude/longitude to CPP is:

$$\text{newpoint}_x = R * \left(\text{point}_{\textbf{longitude}} - \text{origin}_{\textbf{longitude}}\right) * \cos\left(\text{origin}_{\textbf{latitude}}\right)$$

$$\text{newpoint}_y = \text{point}_{\textbf{longitude}} * R$$

The conversion of a point from CPP to latitude/longitude is:

$$\text{newpoint}_{\textbf{longitude}} = \frac{\text{origin}_{\textbf{longitude}} + \text{point}_x}{R * \cos\left(\text{origin}_{\textbf{latitude}}\right)}$$

$$\text{newpoint}_{\textbf{latitude}} = \frac{\text{point}_y}{R}$$

R = 6378206.4 m. (Clarke 1866 major spheroid radius)

Geographic Coordinate System

A Geographic system is a latitude/longitude system defined in decimal degrees. Supported Geographic systems include:

- NAD (North American Datum) 1927 and NAD 1988
- 33 world ellipsoids and a user defined ellipsoid (i.e., Clarke 1866, WGS 1984, etc.)

The hemispheres are defined for non-NAD systems. The hemisphere cannot be changed for NAD systems (Northern, Western hemispheres).

Local Coordinate System

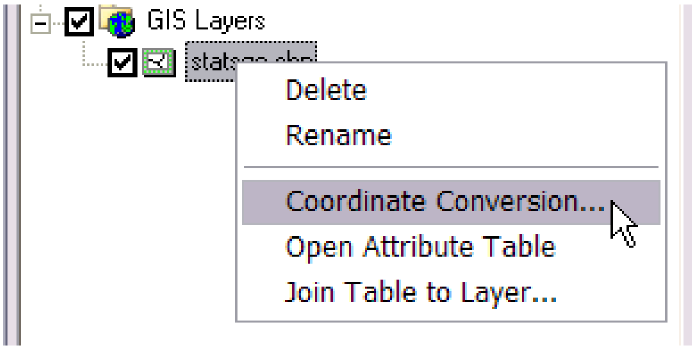
A local coordinate system is a system defined for a survey. Many numerical models work in local systems, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). If a portion of the data for a project is referenced to a global coordinate system, and a portion is in a local system, a transformation to convert all the data to a single system must be defined. This can be done by defining the location and orientation of the origin of the local system with reference to the global system. This allows the data to be referenced back to the global system or to another local system.

Three scenarios are possible when dealing with local coordinate systems:

1. Global to Local – When converting from a global to a local coordinate sytem, the global coordinates of the origin of the local coordinate system must be defined using the Local Origin button in the lower portion of the dialog. This dialog can also be used to enter the angle of rotation of the local coordinate axes relative to the global coordinate axes. The angle is measured ccw from the positive x-axis.
2. Local to Local – When converting from one local coordinate system to another local coordinate system, the same approach is used as when converting from a global to local coordinate sytem. The Local Origin dialog is used to define the coordinates of the origin of the new coordinate system relative to the old coordinate system.
3. Local to Global – When converting from a local to a global coordinate sytem, the Local Origin dialog is used to coordinates of the local coordinate system (which is the old system in this case) relative to the new global coordinate sytem.

Converting Coordinate Systems

You can use the standard WMS coordinate conversion on any shapefile loaded without ArcObjects enabled by right-clicking on the layer and choosing the Coordinate Conversion option. You will need to know the current coordinate system of the data layer so that you can specify it and the coordinate system you wish to convert to.



Related Topics

- Coordinate Systems of GIS Layers
- Coordinate Conversions
- Map Properties

Coordinate Tracking

The cursor tracking box at the bottom of the graphics window displays the x, y, and z coordinates of the cursor as it is moved in the graphics window. If there are no elevation data (TIN or DEM), then no z value will be displayed. If you have a TIN or DEM loaded, then the z value will be interpolated as long as you are in either the Terrain Data or Drainage modules. For example even if you have a TIN loaded, but are in the map module the z value will not be displayed.

The s: value displayed corresponds to the function value of the active data set. When there is only an elevation data set loaded, this should be the same value as the elevation.

Related Topics

- WMS Screen
- Help Strip

Coordinate Systems of GIS Layers

Coordinate systems of GIS layers are managed differently in WMS depending on whether or not ArcObjects is enabled. If ArcObjects is not enabled then the standard WMS coordinate conversion option appears as one of the menus when right-clicking on the data layer in the Project Explorer. When ArcObjects is enabled then the standard ArcMap Map Properties dialog is used to define the coordinate system for display and mapping of GIS layers.

Related Topics

- Coordinate Conversions of Shapfile Layers
- Map Properties of GIS Layers

Editing XYZ Coordinates

Two methods of editing vertex positions and z values are available. To manipulate vertex positions and z values, the Select Vertex tool must be selected.

- A vertex can be moved to a new position by clicking on the vertex and holding down the mouse button while dragging the vertex to the desired position.
- If the current view is plan view, dragging the vertex will cause it to move in the xy plane. WMS will not allow the vertex to be dragged to a position where one of the surrounding triangles would become inverted.
- If the current view is not the plan view, the vertex will move along the z-axis.
- The vertex position and z value can also be manipulated by selecting the vertex and changing the xyz values that will appear in the x, y, and z edit boxes.

Display options such as contours are updated automatically as a vertex's position is altered as long as these options are selected from the TIN Display Options dialog.

Related Topics

- Locked/Unlocked Vertices
- Edit Window

Local Origin Dialog

A local system is a system defined for a survey that can be referenced back to a global system (UTM, State Plane) or to another local system.

Three scenarios are possible when dealing with local coordinate systems:

1. Global to Local - When converting from a global to a local coordinate sytem, the global coordinates of the origin of the local coordinate system must be defined using the Local Origin button in the lower portion of the dialog. This dialog can also be used to enter the angle of rotation of the local coordinate axes relative to the global coordinate axes. The angle is measured ccw from the positive x-axis.
2. Local to Local - When converting from one local coordinate system to another local coordinate system, the same approach is used as when converting from a global to local coordinate sytem. The Local Origin dialog is used to define the coordinates of the origin of the new coordinate system relative to the old coordinate system.
3. Local to Global - When converting from a local to a global coordinate sytem, the Local Origin dialog is used to coordinates of the local coordinate system (which is the old system in this case) relative to the new global coordinate sytem.

Single Point Conversion

The Single Point Conversion dialog allows a user to convert a coordinate between two coordinate systems. The single point conversion dialog is accessed from two locations within WMS:

1. Edit Menu, Convert Single Coordinate Item
2. Register Image Dialog

The dialog consists of two sides. On the left side, the original (Convert From...) coordinate system is entered. On the right side, the final (Convert To...) coordinate system is entered. The options for the two sides are listed in the Coordinate Systems page. The other items in the dialog include:

- **Enter coordinates.** Enter the original (or from) coordinates.
- **New Coordinates.** View the final (or to) coordinates.
- **Convert.** Convert from the original to final coordinates.
- **Create Mesh Node/Feature Point.** Create a mesh node (if in the mesh module) or a feature point (if in the map module) at the final coordinates when the OK button is selected.

Related Topics:

- Coordinate Conversions
- Coordinate Systems

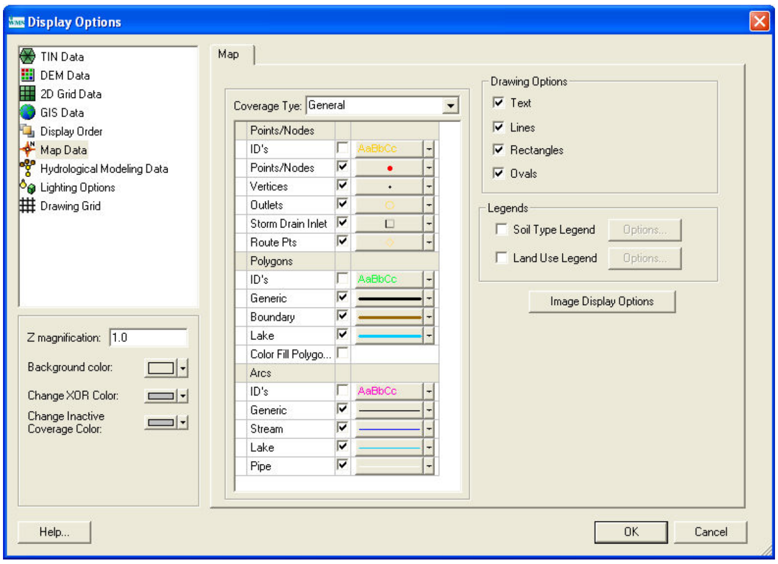
3.3. The Display Options

Display Options

Most of the data types (modules) in WMS have a set of display options that can be modified using the **Display Options** command in the *Display* menu. The **Display Options** command brings up the *Display Options* dialog shown below. The default tab in the dialog depends on which module is active. However, you can change the display options of any module by changing to the appropriate tab. Some modules really use multiple tabs such as the Terrain Data Module which in addition to the *TIN* tab has the *DEM* tab and the *TIN-Drainage* tab. Also the *Drainage Data* tab controls the display of text with computed geometric parameters for watersheds done with either TINs or DEMs. The *General* tab has a few things that do not fit as part of any module such as the background color, the color used to contrast selections (XOR), as well as a few others.

The check box next to the feature named can be toggled on or off to control whether or not the feature is to be displayed. The combo box will either display a symbol, line, text, or polygon in the currently selected color depending on the type of feature. Colors can be changed quickly by selecting the drop-down button, and other styles, fonts, sizes of attributes can be changed by selecting the button displaying the current setting (see descriptions below).

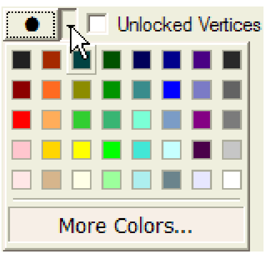
Additionally, entire data objects can be turned off by un-checking the toggle box next to the data object from the *Project Explorer*.



Display Options dialogue

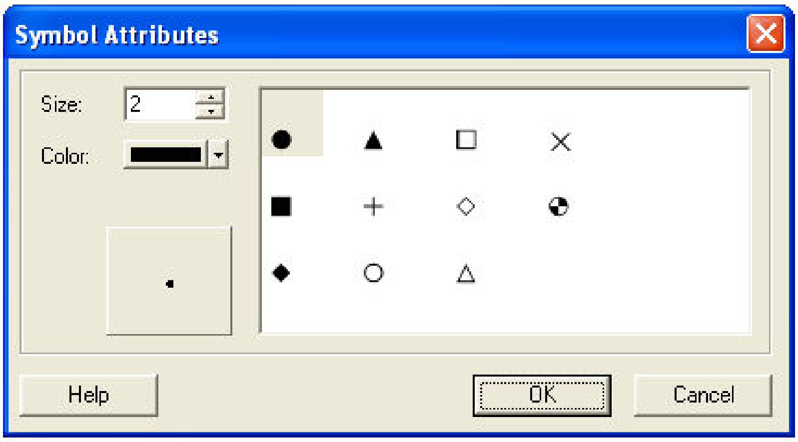
Changing Color

If you select the drop-down arrow a simplified color chooser appears that allows you to change the color only of the attribute.



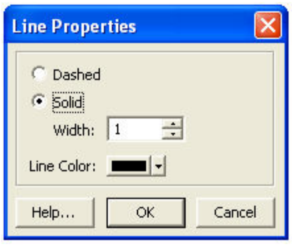
Changing Symbols

If you wish to change the symbol style or size select the symbol button.



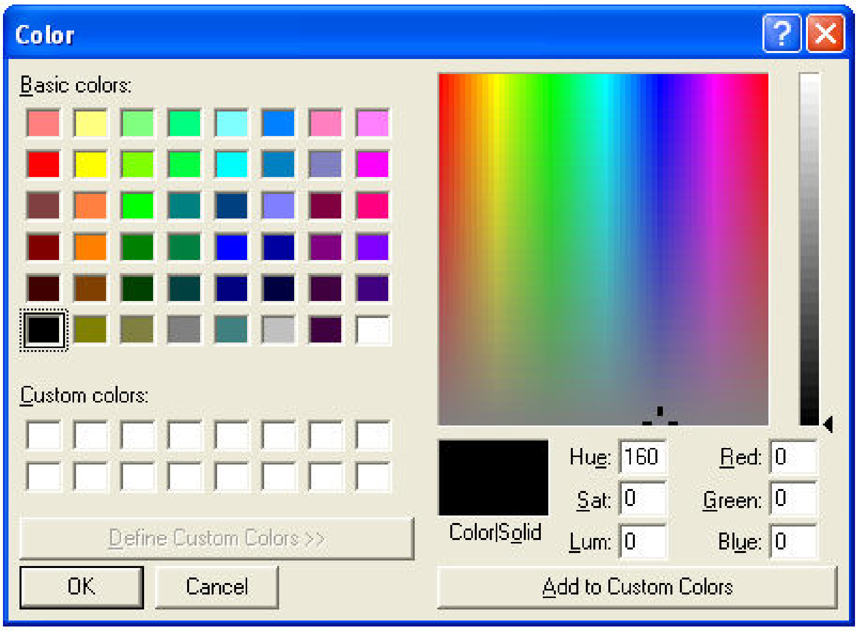
Changing Line Styles

If you wish to change a line thickness or style select the **line symbol** button.



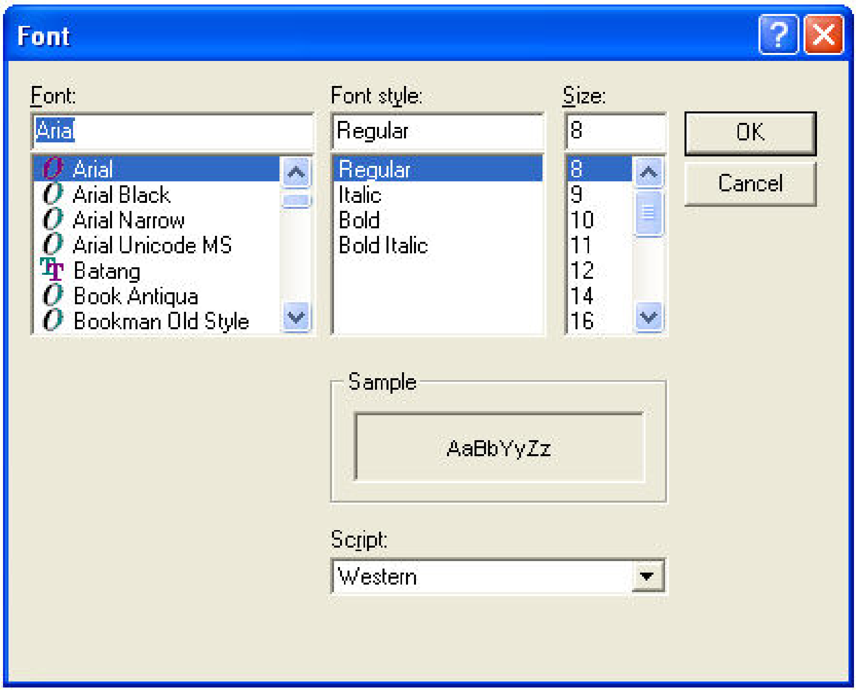
Changing Polygon Colors

The more advanced color chooser is used to specify a polygon color, this is the same dialog that comes up when selecting *'More Colors...* from the basic drop-down color chooser, or when selecting a color from within the *Line or Symbol Attributes* dialogues.



Changing Text Fonts and Styles

If you wish to change a font or font style select the **text symbol** button.

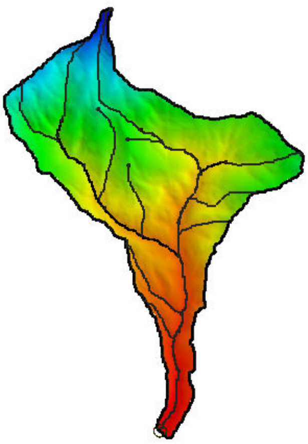


Lighting Options

The *Lighting Options* option in the *Display Options* dialog can be used to set the light and shading of the image in the *Graphics Window*. When the *Use light source* option is selected, the image is shaded according to the intensity and lighting angle. When the *Smooth features* option is selected, the shading is smoothed. The image shows an example of a watershed that has color filled contours and the lighting and shading options applied.

The default display mode for a TIN or 2D Grids in the *Graphics Window* is a wire-frame image. Color shading and hidden surface removal can be applied to either TINs or 2D Grids in order to generate a realistic image. Hill shaded images of DEMs can also be created with a DEM present.

Images can be "draped" over a TIN if properly registered. This is controlled in the *Image Display Options* (found as a button on the *Map* tab in the *Display Options* dialog) by choosing to map the image to the TIN. If an image is mapped it will be draped and shaded over the TIN when using *Lighting Options*.



Related Topics

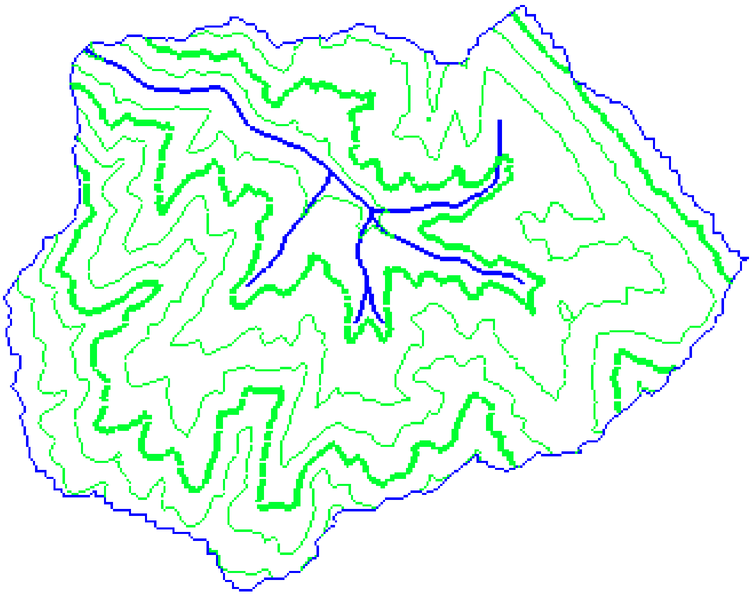
- Contour Options
- Drawing Objects
- Display Order

Color Ramp

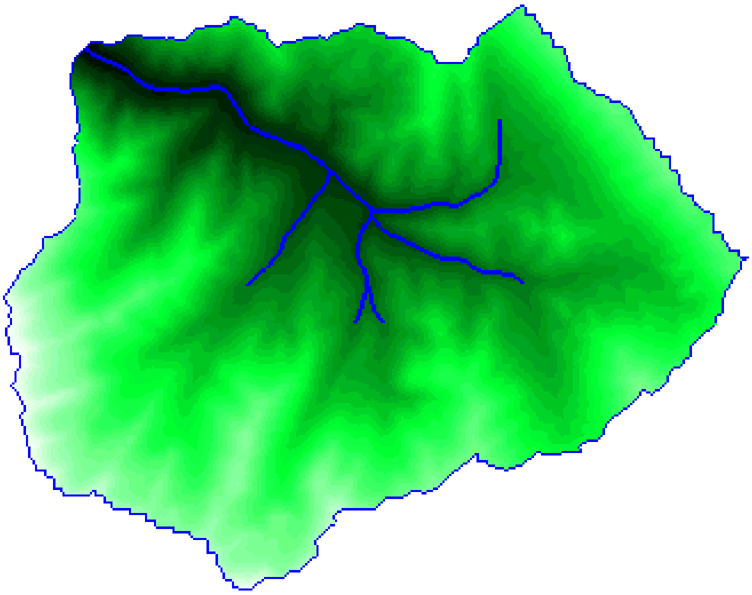
Contours are displayed according to a specified color ramp. The color ramp can be edited by selecting the **Options** button in the *Colors* portion of the *Contour Options* dialog. The *Color Ramp Options* dialog contains the following options:

Palette Method

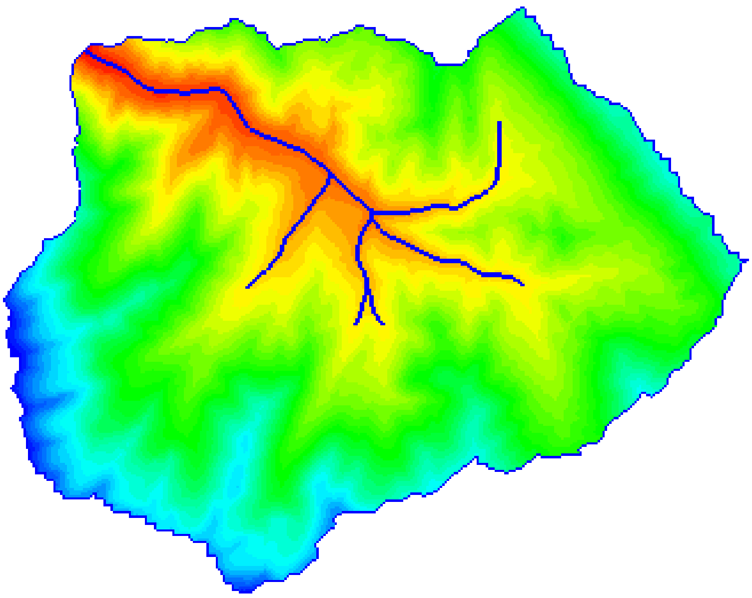
Solid Color will display all contours according to the specified color at the bottom of the palette method group.



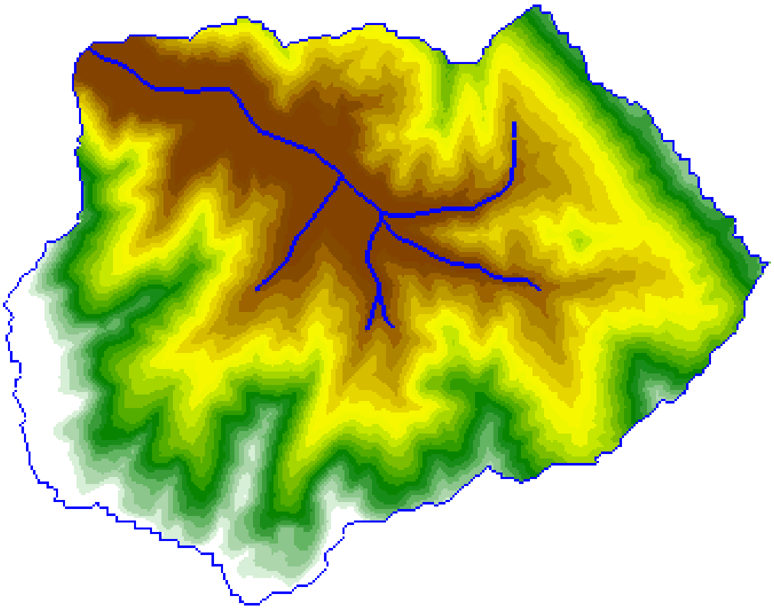
Intensity Ramp defines a ramp of colors corresponding to a varying intensities of the specified color at the bottom of the palette method group. The portion of the range (from black to white) can be established by adjusting the arrow marks corresponding to 0.0 and 1.0 on the color bar at the bottom of the dialog. The ramp can also be reversed so that the bright colors are min and the dark max using the **Reverse** button below the *Palette Preview* color bar.



Hue Ramp is similar to intensity ramp, but uses a ramp of color hues (red-yellow-green-blue-magenta) rather than a single color. The range of colors can be set by adjusting the arrow marks corresponding to 0.0 and 1.0 on the color bar at the bottom of the dialog. The ramp can also be reversed using the **Reverse** button.



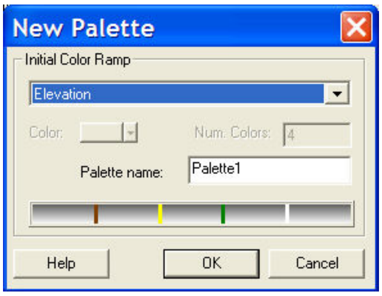
User Defined palettes can also be defined as discussed below.



User-Defined Palettes

The user-defined color palettes are listed in the upper right corner of the *Color Options* dialog. A new palette is created by selecting the **New** button. This button brings up a dialog listing a set of options for defining the initial color palette. These colors can be edited using the *Color Palette* section of the *Color Options* dialog. An existing palette can be deleted using the **Delete** button.

When specifying a new color palette you can choose from an intensity or ramp palette, or use the predefined elevation (from white for elevations through green-yellow and finally brown for the lowest elevations as shown in the previous example figure above).



Once a set of user-defined color palettes are created, they are saved with the project to a palette file (*.pal). The **Import** and **Export** buttons can be used to share user-defined palettes between projects.

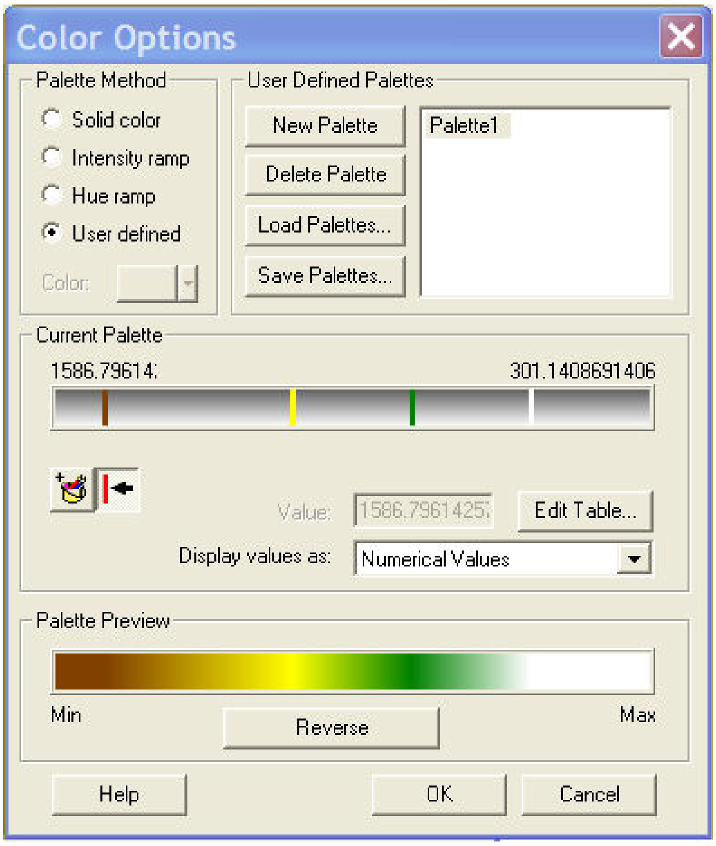
Color Palette

The current color palette is displayed in the Color Palette section. The min and max value of the color ramp can be set by clicking and dragging the two triangles just below the color palette. For user-defined color palettes, new colors can be added, colors can be deleted, and the color associated with a color entry can be edited using the tools just below the palette.

The data value associated with a selected color can be edited either by dragging the color or by entering a new value directly. The values can be displayed as either percentages or direct values (corresponding to the active data set). The **Edit Table** button can be used to edit the colors and corresponding values directly in a tabular format.

Preview

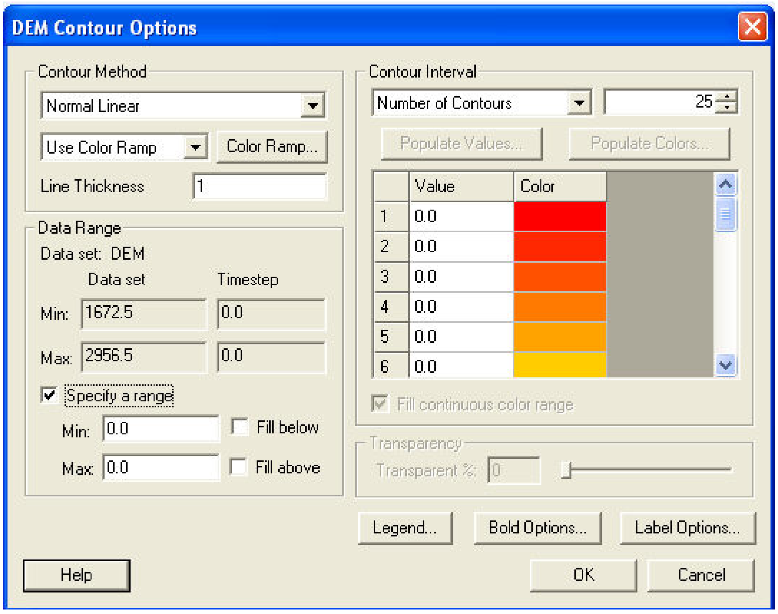
The *Preview* section at the bottom of the *Color Ramp Options* dialog displays the color ramp defined by the current palette and max and min values. The **Reverse** button can be used to reverse the direction of the color ramp (for example, to switch from red-yellow-green-blue to blue-green-yellow-red).



Related Topics

- Contour Options
- Contour Labels

Contour Options



The options used to generate contours can be edited by selecting the *Contour Options* command in the *Display* menu. Each object that can be contoured (i.e. DEMs, TINs, Grids, Flood Plains) has its own set of contour options. This means that you can have color filled contours for a DEM and linear contours on a TIN at the same time. The set of contour options you are editing is determined by either the current module and/or the display options dialog from which the contour options are activated. For 2D grids there is set of contour options associated with dataset and the options of each dataset can be specified by right-clicking on the dataset folder from the *Tree* window, or by selecting an active dataset and choosing the **Contour Options** command or Macro.

The items in the *Contour Options* dialog are as follows:

Dataset Information

The values shown in the upper left corner of the dialog correspond to the maximum and minimum values in the active dataset (TIN, DEM, etc.). These values are useful when choosing an appropriate contour interval.

Contour Interval

Three options are provided for defining contour intervals in the *Contour Options* dialog. The options are as follows:

Number of Contours

With the *Number of Contours* option, an integer is entered representing the total number of contours. The contour interval is adjusted based on the current active dataset so that the contours are evenly spaced and the number of contours correspond exactly to the specified value.

Specified Interval

With the *Specified Interval* option, the contour interval (5, 10, 20, etc.) is entered directly by the user.

Specified Values

The *Specified Values* option allows the user to enter a list specific contour values. Contours are only generated at these values.

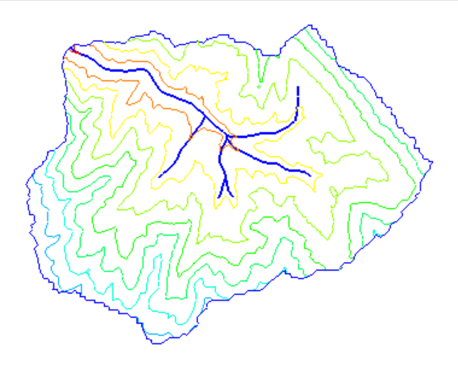
If the *Log Scale* option is selected, WMS automatically assigns contour intervals as multiples of 10. Furthermore, if the color-fill or color ramp option is being used, the colors assigned to the contours are distributed in a logarithmic fashion, rather than linearly interpolated from the low to high values.

Contour Method

The options at the lower left of the dialog control how the contours are computed. Three contouring methods are available:

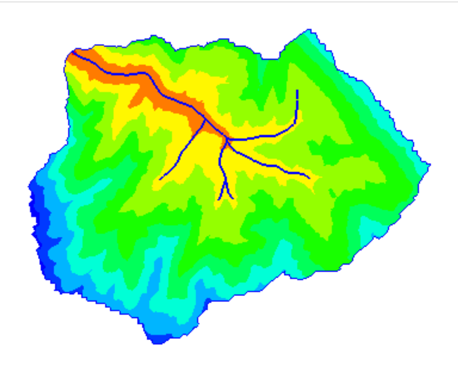
Normal Linear Contours

The default method is Normal linear contours and results in the contours being displayed as piece-wise linear strings.

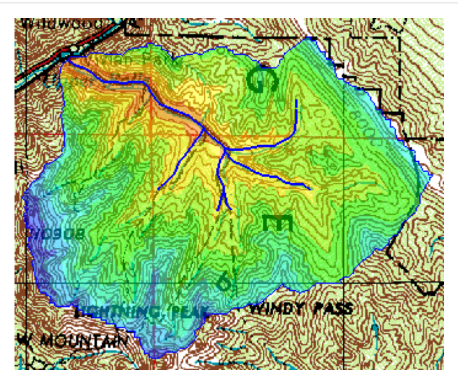


Color Fill Between Contours

If the **Color fill between contours** button is selected, the region between adjacent contour lines is filled with a solid color. For DEMs each DEM grid cell is filled with the color corresponding to the elevation of that cell. For 2D Grids the color fill can be smoothed across cell boundaries (similar to the TINs method) or cell filled (similar to the DEM method).



A transparency value can be set for color filled contours so that an image or other data can be seen through the color filled contours.



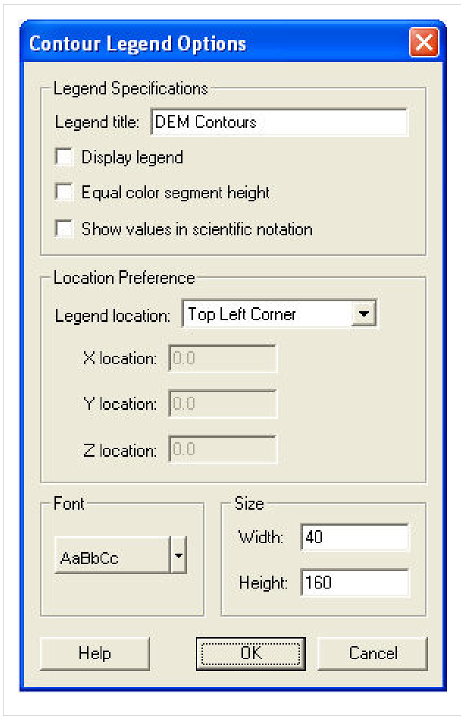
Cubic Spline Contours

If the **Cubic spline contours** button is selected, the contours are computed in strings and drawn as cubic splines. In some cases, drawing the contours as splines can cause the contours to appear smoother.

Occasionally, loops appear in the splines or the splines cross neighboring contour splines. These problems can sometimes be fixed by adding tension to the splines. A tension factor greater than zero causes the cubic spline to be blended with or converge to a linear spline based on the same set of points. A tension factor of unity causes the cubic spline to coincide with the linear spline.

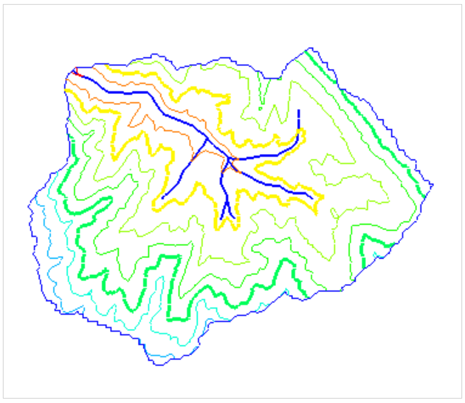
Legend

A legend displaying the range of colors and associated values can be turned on for the current set of contour options. The legend options include a title, font, size and position. By default the legend will be placed in the upper left of the graphics window, but can be positioned in the other three corners as well. This is important when displaying contours of more than one dataset at a time.



Bold Contours

The *Bold every...* option can be used to display contours at selected intervals with a thicker line width.

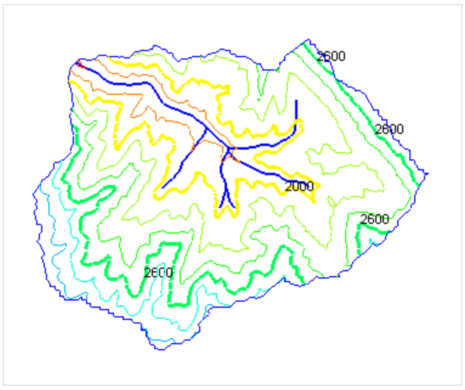


Contour Labels

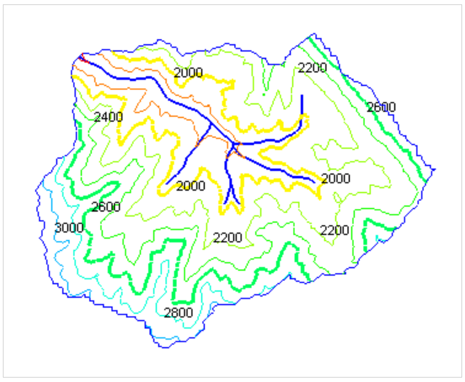
The *Label every...* option can be used to plot labels on contours at selected intervals. The contour label options are edited using the *Contour Labels* dialog.

The *Contour Label Options* tab in the *Contour Options* dialog is used to set the label color and font, the number of decimal places used to plot the label, and the spacing used when the labels are generated automatically. The default spacing value controls the placement of labels when labels are generated automatically. Labels can be added to contours in one of two ways:

- If the contour label option is selected in the *Contour Options* dialog, labels are automatically placed on the contours.



- In some modules, contour labels can be added manually to contours by selecting the **Contour Labels** tool in the Tool Palette and clicking on the contours where labels are desired. By default, the data set value corresponding to the point that was clicked is computed and a label corresponding to the nearest contour value is drawn centered at the point that was clicked. An option can be set in the *Contour Label Options* dialog to use the exact value at the point that is clicked as opposed to using the nearest contour value. This option is useful to post data set value labels in regions where there are no contours.



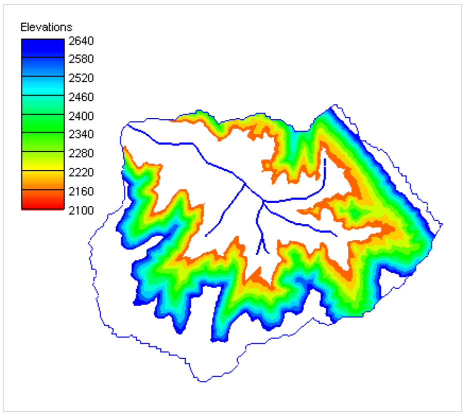
If the mouse button is held down, a box showing the outline of the label is drawn. The box can then be positioned precisely with the mouse. A line is drawn from the box to the point that was clicked to help you keep track of the contour that was selected. Contour labels can be deleted by holding down the *SHIFT* key while clicking on a label.

The orientation of labels can either be parallel with the contours or always horizontal on the screen

The **Remove Current Labels** button will eliminate all contour labels, whether they were created with the labeling tool or automatically.

Contour Specified Range

Regardless of which option is selected for the contour interval, a maximum and a minimum contour value can be specified and the contouring can be restricted to the specified range.



Contour Colors

The options in the lower right corner of the Contour Options dialog define what colors are used when the contours are displayed. The Color Options button is used to set the default contour color.

Related Topics

- Display Options
- Contour Labels
- Color Ramp

Drawing Objects

The Map module includes a set of tools for adding simple graphics and annotation to a plot. *These tools are not intended to be a full-featured drawing package as would be found in products like AutoCAD or Corel Draw*. However, they can be very useful for adding titles, arrows, and other annotation to a plot so that the plot can be directly included in a project report without the need to import the plot into an external drawing package prior to report generation.

The types of drawing objects that can be created in the Map module are text, lines (including arrows), rectangles, and ovals. Drawing objects are created and edited using the Drawing Object Tools in the Tool Palette. Drawing objects are saved in the Map file along with feature objects.

Drawing Object Attributes

Each type of drawing object has a set of graphical attributes that can be edited by selecting the object with the Select Drawing Objects tool and selecting the **Attributes** command in the **Drawing Objects** menu. The attributes can also be edited by double-clicking on an object.

Text Attributes

If a text object is selected, the **Attributes command** in the **Drawing Objects** menu brings up the Text Attributes dialog. The dialog can be used to change the font, the color, or the text string itself. An option is also provided to fill a rectangle just containing the text with a user-specified color. This option can be useful to help the text stand out from the objects being drawn behind the text.

Rectangle and Oval Attributes

The attributes for both rectangles and ovals can be edited with the Rectangle/Oval Attributes dialog. Rectangle and oval attributes include line style, line color, and line width. An option can also be set to either draw only the outline of the rectangle or oval (no fill) or fill the object with a user-specified fill pattern and color.

Line Attributes

The attributes for lines are edited with the Line Attributes dialog. The line attributes include line color, line width, and line style. The arrowheads associated with a line can also be edited. The length and width of the arrowhead can be defined along with the placement of the arrowheads. The arrowheads can be placed at the beginning of the line, the end of the line, or at both ends of the line.

Default Attributes

When a new object is created, it inherits the default attributes for that object type. The default attributes are defined by selecting one of the drawing object tools (line, rectangle, oval or text) and selecting the **Attributes** command in the **Drawing Objects** menu.

Drawing Depth

In some drawing packages, the drawing takes place in a purely two-dimensional medium. However, since the objects in WMS can be three-dimensional, the drawing objects must be positioned in three-dimensional space. This is accomplished by utilizing a drawing depth when drawing objects are created. When drawing objects are first created, they are created in a plane that is orthogonal to the current viewing angle. For example, in plan view, objects are drawn in an xy plane. When drawing in a plane, a depth must be defined for the plane. For example, when drawing in the xy plane, the drawing depth is the z coordinate of the xy plane.

The drawing depth is designated with the **Drawing Depth** command in the **Drawing Objects** menu. Two options are provided in the associated dialog: the objects can either be drawn at the average depth of the visible objects or drawn at a user-specified depth.

Drawing Object Order

The order in which drawing objects in the Map module are displayed becomes important whenever a rectangle or oval is displayed in the color fill mode. The order of drawing objects can be controlled using the **Move to Front**, **Move to Back**, **Shuffle Up**, and **Shuffle Down** commands of the **Drawing Objects** menu.

Move to Front

The **Move to Front** command causes the selected drawing object to be drawn last. In other words it appears on top or in front of all other drawing objects.

Move to Back

The **Move to Back** command causes the selected drawing object to be drawn first. In other words it appears at the bottom or in back of all other drawing objects.

Shuffle Up

The **Shuffle Up** command causes the selected drawing object to be displayed one object later than it is currently displayed. This causes it to appear in front of the object which is currently being displayed just ahead of it.

Shuffle Down

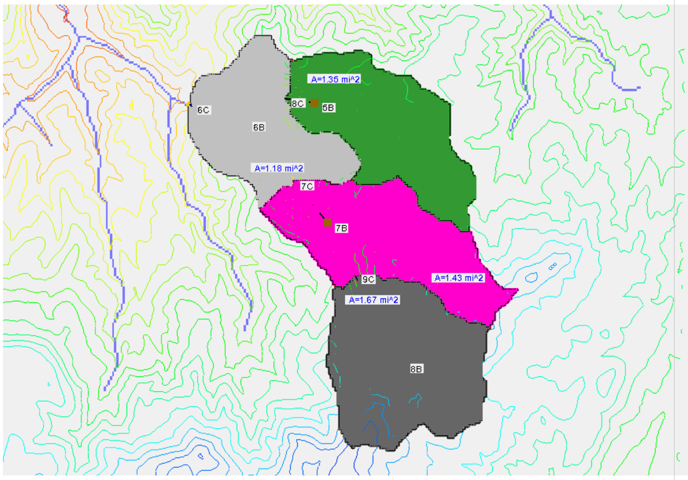
The **Shuffle Down** command causes the selected drawing object to be displayed one object sooner than it is currently displayed. This causes it to appear behind the object that is currently being displayed just behind it

Related Topics

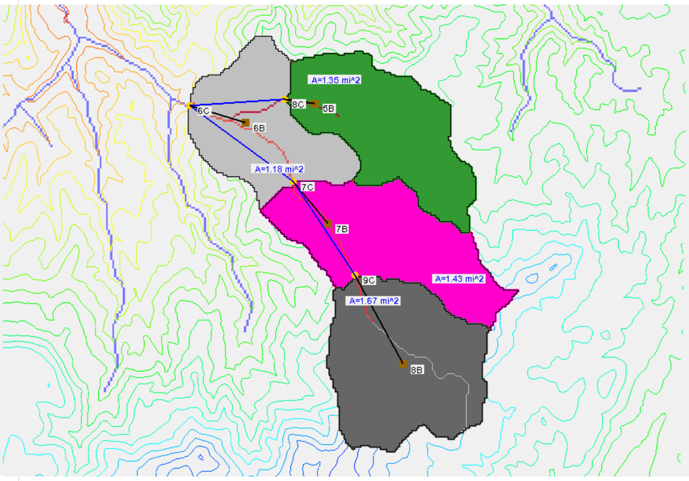
- Map Module
- Drawing Tools
- Display Options

Display Order

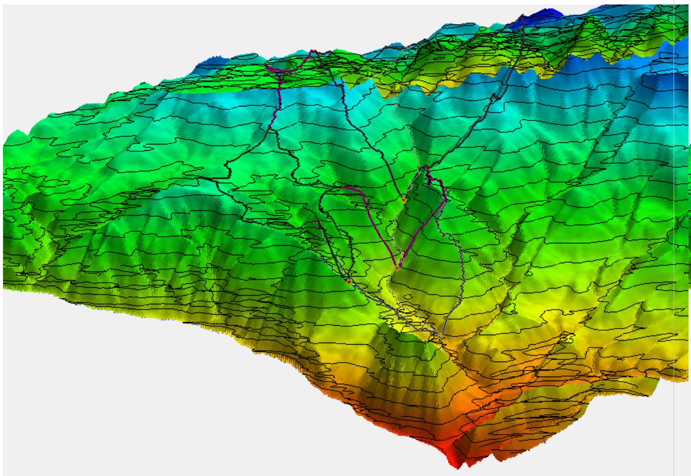
The display order specifies the order that objects are displayed in WMS. The WMS default is to turn the display order on, which means objects are displayed in the order specified in the WMS Display Options dialog. When the display order is turned off, objects that have XYZ coordinates that are closer to your eye are displayed in front of objects that are further from your eye. Turning the display order off can often have undesirable consequences when displaying layers of watershed data in plan view. However, it can be useful to turn the display order on when displaying texture mapped TINs or shaded data in oblique view. See the adjacent images for examples of data displayed in plan and oblique view with display order turned off and on.



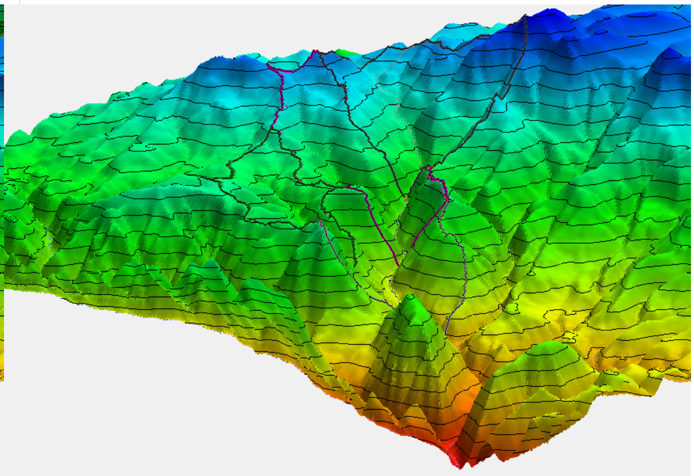
A plan view of a delineated watershed with display order turned **off**. Notice several contours and other objects "bleed" through the polygons to make things look cluttered.



A plan view of a delineated watershed with display order turned **on**. Notice the cleaner look of the watershed map.



An oblique view of a delineated watershed with display order turned **on**. Notice how the contours look cluttered because entire objects are displayed using display order instead of closer entities being displayed in front of entities that are further away.



An oblique view of a delineated watershed with display order turned **off**. Notice the cleaner look of the watershed map.

Related Topics

Display Options

3.4. CAD Menu

CAD Data

Opening CAD Files

WMS uses a third party set of DLLs to read CAD files. This is done so that WMS can stay current with evolving changes in DXF and other file formats. Previously, only limited DXF support was provided, and only for files exported from AutoCAD release 12 or earlier. This interface not only provides expanded and up to date support of DXF, but AutoCAD *.dwg files are also supported. Future versions will likely support the MicroStation *.dgn format as well.

CAD files are not yet managed from the WMS *Project Explorer*, but this will likely be supported in the future as well. WMS can use CAD files as base map (background) information to your project, and to convert some CAD features into feature objects of the current coverage.

Saving CAD Data

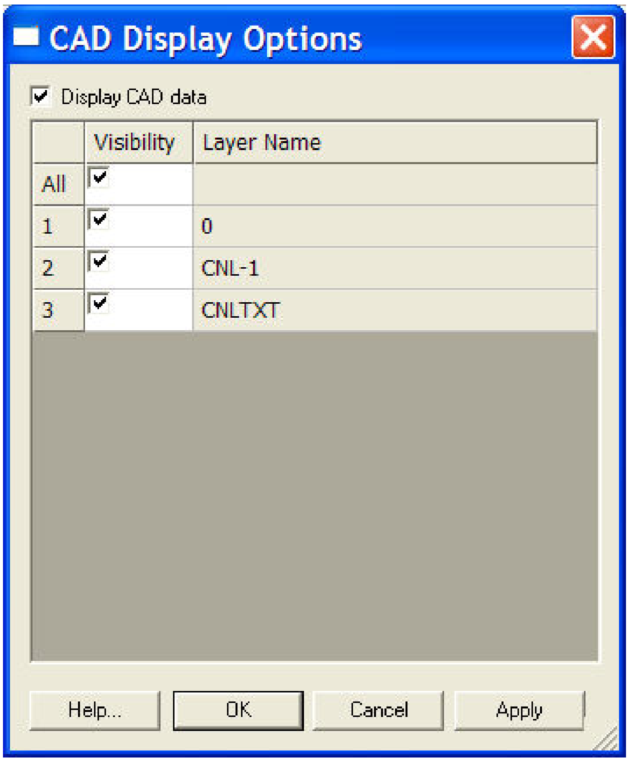
Most of the data objects within WMS (e.g. coverages, contours, etc.) can be converted to CAD layers. Once converted to CAD layers these data can be saved to either a DXF or DWG (AutoCAD native file format) using the **Save As** command in the *File* menu. The *Save As* option will only save the current CAD layers to the file so you must convert your WMS data to CAD prior to saving if you wish to have data created in WMS saved to a CAD format.

Deleting CAD Data

The **Delete** command of the *CAD* menu is used to delete all CAD data. It is not currently possible to delete individual layers, but the visibility of a layer can be controlled from within the *display options* dialog.

CAD Display Options

The objects in CAD files are organized into layers. The display of layers in a CAD drawing is controlled using the **Display Options** command in the *CAD* menu. This command brings up the *CAD Display Options* dialog, shown below.



List of Layers

The names of the layers in the drawing are shown in the box on the left of the dialog. A check mark appears to the left of the names. The visibility of a layer is controlled using these toggles. All of the layers can be made toggled on/off with the option for *All* that is at the top of the list.

Future versions of WMS will likely list the CAD data and control the display from within the *Project Explorer*.

Colors/Styles

With the current implementation of the new CAD data colors and styles are inherited from the original drawing and cannot be changed from within WMS.

Converting WMS Data to CAD

If you want to convert WMS data objects such as feature object coverages, TINs, contours, etc. you can choose the **Data → CAD** option in the *CAD* menu. This will convert the various WMS data objects into separate CAD layers which can be controlled from the *Display Options* dialog. The colors of the layers will be inherited based on the current WMS display options settings for the individual objects.

At the current release polygon data types are not supported in the conversion from WMS to CAD, however arcs that make up polygons, or triangles that make up color-filled contours are saved as layers.

Example

To export a delineated watershed with only the watershed boundary displayed, the user would follow these steps:

- 1. Select *Display* menu then **Display Options...** .
- 2. Select **Map Data** on the left side of the *Display Options* dialog.

- 3. Make sure that only the *Generic* (under *Arcs*) box is checked in the *Map* tab of the *display options*.
- 4. Right click on the *DEM* item under *Terrain Data* in the *Project Explorer*.
- 5. Select **Delete**.
- 6. Switch back to the map module by either selecting *Map Data* in the *Project Explorer* or the *Map module* macro at the top of the screen.
- 7. Select the **Select feature line branch** tool in the map module toolbar.
- 8. Delete each of the stream arcs by selecting each with the **Select feature line branch** tool.
- 9. Make sure you are back in the Map module (see step 8).
- 10.Select **CAD | Data → CAD**.
- 11.Select **File | Save As...** and save the file as a *.dwg/*.dxf file.

How to convert a floodplain boundary to a CAD drawing

1. Right-click on your flood depth dataset and set the contour options to normal linear, specified values, and a single contour with a value of 0.001 .
2. Right-click on your TIN, select convert, select **TIN Contours→Feature**.
3. Remove any unwanted arcs from your floodplain boundary.
4. Delete any data you don't need to export (TIN, area property coverage, 2D scattered data).
5. In the map module, go to the *CAD* menu and select **Data→CAD**.
6. Select **File | Save as...** and save your file as a DWG file.

Converting to Feature Objects

Many times you will want to use your CAD data layers to create streams, watersheds, or other feature objects. To do this you must first convert the CAD data to feature objects. This is done by choosing the **CAD → Feature Objects** command in the *CAD* menu and then specifying which objects in the layer you want converted. When converting data you may either add it to the currently active coverage, or have a new coverage created. Once the data have been converted it is a good idea to delete the CAD data or at least toggle of the visibility.

Converting to TIN

Sometimes digital elevation data are stored in CAD files in the form of 3D points and 3D faces. These CAD objects can be converted to TINs using the **CAD→TIN** command in the *CAD* menu. After converting the data it is a good idea to delete the CAD layers, or at least toggle off their visibility.

Related Topics

- Feature Objects
- Triangulation

3.5. Hydrologic/Hydraulic Calculators

Hydrologic/Hydraulic Calculators

The **Calculators** menu contains several utility functions that assist in developing hydrologic modeling input parameters, and perform design and analysis of channel hydraulics, detention basins, and culverts once a hydrologic model has been developed. The following is a list of calculator tools available in WMS:

- Compute GIS Attributes – A dialogue that allows computations from standard GIS formatted files.
- Lag Time and Time of Concentration – WMS provides two powerful methods (*basin data equations* and *travel time equation*) of computing travel times for lag time and time of concentration from the geometric data you are using for basin delineation and parameter estimation.
- HY8 Culvert Analysis [1] – A web-based program with a variety of calculators that can be used with WMS.
- Detention Basin Calculator – The effects of a detention basin on an inflow hydrograph can be analyzed and an output hydrograph created in WMS using the *Detention Basin* calculator. This same calculator is also used in to define detention basin parameter input for HEC-1, the Rational Method and other hydrologic models as part of an overall analysis for a planned development.
- Channel Calculator – WMS, beginning in version 9.1, uses *Hydraulic Toolbox* to perform the channel calculations. It is useful to be able to analyze the conveyance and other properties of channels using Manning's equation.
- Weir Calculator – Head or flow over a weir can be determined using the *Weir Calculations* dialog.

References

[1] http://www.xmswiki.com/xms/WMS:HY-8

Lag Time and Time of Concentration

Lag time and time of concentration are variables often used when computing surface runoff using unit hydrograph methods available in the hydrologic models supported in WMS. These variables indicate the response time at the outlet of a watershed for a rainfall event, and are primarily a function of the geometry of the watershed. WMS provides two powerful methods of computing travel times for lag time and time of concentration from the geometric data you are using for basin delineation and parameter estimation.

The first method is to use one of several empirical equations (or user-defined) based primarily on the basin data computed by WMS when using a DEM or TIN for basin delineation. Many different equations have been developed for different watersheds, and most of these equations are a function of the geometric parameters computed from digital terrain models.

The second method allows you to create a time computation coverage in the map module and then define the "representative" flow paths within each basin using arcs that are used to determine lag or time of concentration. A travel time equation can then be assigned to each arc (length and slope are automatically determined from the arc when a DEM or TIN is present) and the sum of the arc travel times within a basin used for time of concentration or lag time. Pre-defined equations such as are used by the FHWA or in TR-55 can be selected or user-defined equations developed.

There is no distinct advantage of one method over the other. Each allows a certain amount of customization and the ability to generate a summary report in a text file or by copying to the clipboard so that these critical input data can be well documented. In general, if time of travel can be determined from a single empirical equation then computing using the basin data will be more convenient whereas if the time of concentration or lag time is determined by combining the time of travel across one or more flow path segments, (overland flow, shallow concentrated flow, channel flow, etc.) then the map data method will likely work best.

Each of the hydrologic models supported by WMS that require a lag time, time of concentration, or channel travel time allow you to pick either of the two methods. Buttons adjacent to the input fields allow access to the different methods and the computed result is used to define the input value for the model you are working on (i.e. TR-55 time of concentration, TR-20 lag time, etc.). You may also compute travel times for selected basins or outlets using the **Compute Travel Time** command from the **Calculators** menu. This dialog will allow you to choose between the two methods for a selected basin (only the Map module method is available for a selected outlet). The computed travel time can then be assigned to the relevant input parameter for the selected hydrologic model (the hydrologic model corresponds to the current default model and can be changed using the drop-down combo box).

Related Topics

- Hydrologic/Hydraulic Calculators
- Travel Times from Map Data
- Travel Times from Basin Data
- Time Computation Coverage

Compute GIS Attributes

WMS allows you to define coverages, grids, or GIS layers that define boundaries for different soils, land uses, rainfall depths, and DPA zones. Typically this information is imported from standard GIS formatted files. This GIS data is overlaid with drainage basin boundaries to compute area-weighted composite model parameters for each sub-basin.

In summary the following data are used for computing composite model parameters:

- Basin boundaries from the drainage boundary polygons on a drainage coverage.
- Land use IDs are supplied from a land use coverage in the map module or as DEM (a gridded) attributes.
- Soil IDs are supplied from a soil type coverage in the map module or as DEM (a gridded) attributes.
- A user-defined table relating land use IDs to the parameters being mapped (for example SCS curve numbers, Green & Ampt parameters, etc.).
- A user-defined table relating soil IDs to the parameters being mapped (for example runoff coefficients, Green & Ampt parameters, etc.).

Any combination of data sources for computation can be used (i.e. drainage coverage, land use grid, soil type coverage, etc.). If a land use or soil coverage is used, then the parameters for each polygon ID can be defined using the **Attributes** command in the *Feature Objects* menu (in the Map module) with the proper coverage being active. However, if grid attributes are used for the soil or land use ID definitions, then one way to define the parameters for each ID is by creating the mapping file with a text editor and then importing in the *Compute GIS Attributes* dialog (see Mapping File Formats).

NRCS soils files that are available for download on the internet often contain the hydrologic soil group attribute in a separate database file than the feature polygons themselves. The tools in the GIS module that allow you to join tables or specific attributes from tables to your feature polygons can be used to link the hydrologic soil attribute to the polygons. See more information in the section on joining tables.

Once the polygon coverages and/or grid files for land use or soil types are defined, and the mapping tables set up, you are prepared to compute parameters for one of the available methods.

When computing GIS attributes the results are automatically stored with every applicable model supported in WMS.

NRCS Curve Numbers (CN)

A composite curve number for a basin can be computed by taking an area-weighted average of the different curve numbers for the different regions (soil type and land use combinations) within a basin.

Required Inputs:

- Land use data
- Soil type data (hydrologic soil group A, B, C, or D, where the infilatration capacity decreases from A to D)
- Table relating land use IDs to curve numbers for each hydrologic soil group

One good source for curve numbers is the TR55.pdf file included in the \docs folder of the WMS installation.

Runoff Coefficients

Composite runoff coefficients are computed using an area-weighted average of all runoff coefficients that overlay each drainage basin. Soil data can also be used to infer runoff coefficients.

Required Inputs:

- Soil data
- Table relating soil IDs to runoff coefficients

OR

- Runoff coefficient coverage

Green & Ampt Losses

Maricopa County, Arizona, and other regions often use the Green & Ampt infiltration options within HEC-1.

Required Inputs:

- Land use data
- Table relating land use IDs to initial abstraction and percent impervious
- Soil data
- Table relating soil IDs to hydraulic conductivity, soil moisture deficit, and wetting front suction

The parameters required to define these values must be entered for the appropriate coverage.

HSPF Segments

Required Inputs:

- Land use data
- Table identifying land use IDs as either pervious or impervious

Rainfall Depth

A composite rainfall depth is computed using a rainfall depth grid.

Required Input:

- Rainfall depths

Debris Production (Los Angeles County)

Debris production and bulking rates for burned simulations in Los Angeles County are computed by overlaying DPA zones and land use data with drainage basins.

Required Inputs:

- DPA zones are used for determining debris production and bulking rates
- Land use data is used for determining whether or not debris will be produced according to the percent impervious value

Orange County (CA) Losses

WMS can compute both the Fm and Ybar loss parameters used for hydrology in Orange County, CA.

Required Inputs:

- Land use data
- Soil type data
- Table relating land use IDs percent impervious and to curve numbers for each hydrologic soil group

Maricopa County (AZ) $m$and bValues

Required Inputs:

- Land use data
- Table relating land use IDs to $m$and $b$values

HEC-HMS SMA Losses

Required Inputs:

- Land use data
- Table relating land use IDs to canopy and surface storage
- Soil data
- Table relating soil texture to soil and groundwater storage, infiltration rates, and percolation rates

Related Topics:

- Computation Step
- Hydrologic/Hydraulic Calculators
- Coverage Overlays
- Importing Shapefiles
- Land Use
- Soil Type
- Coverages
- Mapping Tables
- Joining Tables

Detention Basin Calculator

An important aspect of any hydrologic study is the development of on-site storage facilities. The effects of a detention basin on an inflow hydrograph can be analyzed and an output hydrograph created in WMS using the *Detention Basin* calculator. This same calculator is also used in to define detention basin parameter input for HEC-1, the Rational Method and other hydrologic models as part of an overall analysis for a planned development.

A level pool routing technique is used to determine the effects of storage-routing on an input hydrograph for given detention basin/reservoir parameters. Using the principle of conservation of mass, the change in reservoir storage, S, for a given time period, Δ_t, is equal to the average inflow, I, minus average outflow, 0.

$$\frac{S_2 - S_1}{\Delta_t} = \frac{I_1 + I_2}{2} - \frac{O_1 + O_2}{2}$$

The defined storage vs. discharge relationships are used to iteratively solve for the end of period storage and outflow.

The detention basin calculator requires three sets of input:

1. A hydrograph.
2. A storage-capacity (volume-elevation) relationship.
3. An elevation-discharge relationship.

When computing an outflow hydrograph an initial storage is used to account for any volume of water that may be in the detention basin prior to the arrival of the inflow hydrograph. If depth or elevation is known, the elevation vs. volume storage capacity curve must be used to determine the initial storage. The units of the initial storage should be the same as the units defined for the storage-capacity relationship

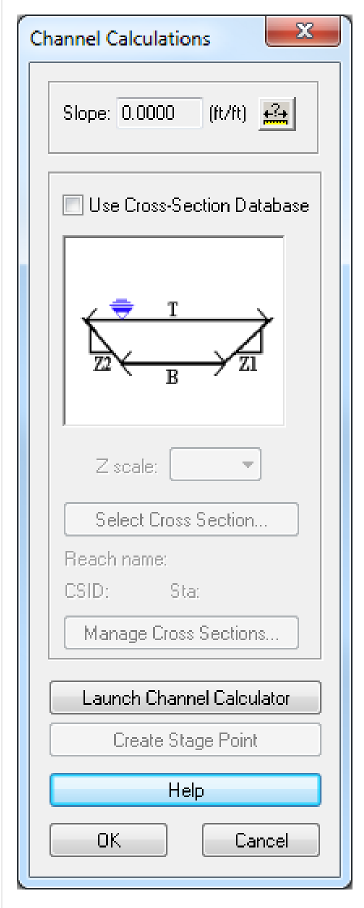
The storage-capacity and elevation-discharge curves (no matter how they are defined) are plotted in the detention basin calculator. They can also be displayed in the hydrograph window by selecting the respective **Plot to Hydrograph Window** buttons. Each of the curves can be printed by selecting the **Print** button. The **Plot Options** button accesses the plot options dialog in order to allow for control in the overall appearance of the defined curves.

The detention basin calculator uses the *Hydraulic Toolbox* to perform the calculations.

Related Topics

- Hydrologic/Hydraulic Calculators
- Storage Capacity Curves
- Elevation Discharge Relationship
- Hydrographs

Channel Calculator



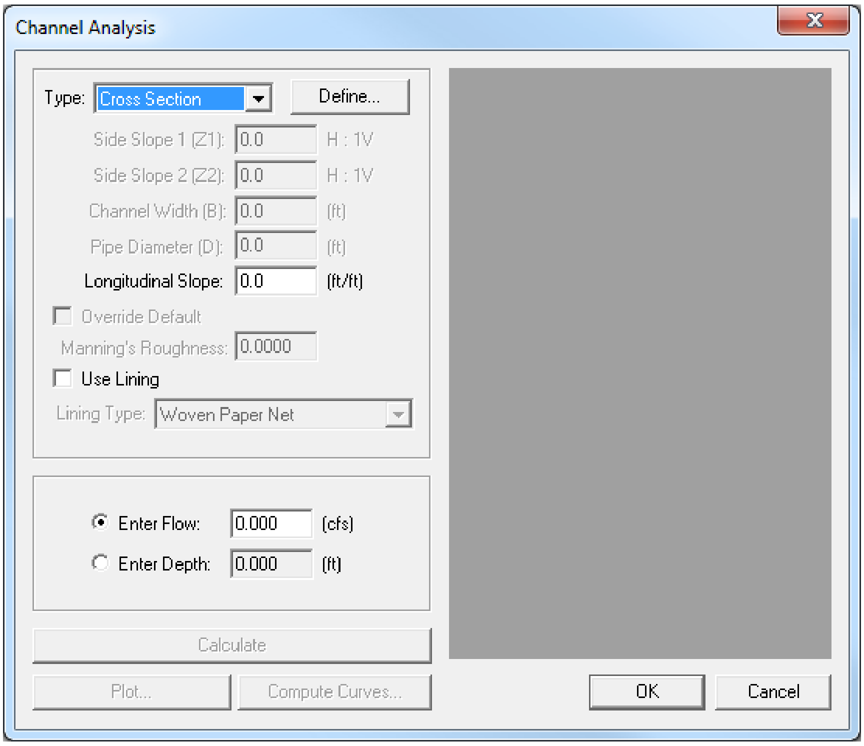
WMS, beginning in version 9.1, uses *Hydraulic Toolbox* to perform the channel calculations. It is useful to be able to analyze the conveyance and other properties of channels using Manning's equation. The first dialog of the *Channel Calculator*, shows tools for slope, cross sections, launching Hydraulic Toolbox's channel calculator, and creating a stage point.

The user can use the measure tool to determine the slope across a specified arc, as long as the user has elevation data loaded into WMS.

The user can then turn on cross sections. You select the channel by selecting the **Select Cross Section** button. The image showing channel geometry, becomes updated with the selected cross section and the Z scale can be adjusted to better view the channel. The *Reach name*, *Cross Section ID*, and *station* are all updated when a cross section is selected.

Clicking **Create Stage Point** is used floodplain delineation, which is described below.

Clicking **Launch Channel Calculator** will show the Channel Calculator from Hydraulic Toolbox with any information entered in WMS.



The Channels calculator allows for the definition of rectangular, trapezoidal, triangular, circular, and user-defined cross sectional channels.

Once channel input geometry is specified, either depth or flow can be computed after supplying a value for the other.

User-defined cross-sections are defined from a cross section coverage and can be interpolated from a background TIN or DEM.

All calculations (except Froude Number) are made using Manning's Equation:

$$Q = \frac{1.49}{n} A R^{\frac{2}{3}} S^{\frac{1}{2}}$$

where:

Q – Flow in cfs

n – Manning's roughness

A – Cross-section area of flow

R – Hydraulic radius

S – Slope

The Froude Number is computed from:

$$N_F = \frac{V}{\sqrt{gy}}$$

where:

N_F – Froude Number

V – Velocity

g – acceleration due to gravity

y – equivalent depth of flow for a rectangular channel.

The equivalent depth of flow for a rectangular channel is computed by dividing the cross sectional area of flow by the top width of the water surface.

Flood Plain Delineation Computations

Besides the ability to analyze the hydraulic properties of channels, results can be used to perform basic flood plain delineation. If you have computed flowrates from one of the supported hydrologic models, you can estimate stage or water surface elevation using the channel calculator.

When computing the depth using a cross section arc you can have a new scatter point created along the arc that contains the water surface elevation (depth calculated plus the lowest elevation of the cross section). If a centerline arc in a 1D-Hydraulic Centerline coverage exists the new scatter point will be created at the intersection of the centerline and cross section when selecting the **Create Stage Point** button. A set of scattered points create in this fashion can be used to perform a flood plain delineation. The scatter points should be interpolated along the cross sections and centerlines prior to delineating the flood plain in order to provide the delineation algorithm with more points so that the interpolation functions work better. Some studies would require a more complete hydraulic analysis using a 1D (HEC-RAS) hydraulic modeling program.

Related Topics

- Hydrologic/Hydraulic Calculators
- Preparing Stage Data for Floodplain Delineation
- Interpolating Stages
- Flood Plain Delineation
- Hydraulic Modeling

Weir Calculator

Head or flow over a weir can be determined using the *Weir Calculations* dialog. If flow is to be calculated then head over the weir must be entered as an input. If head is to be computed then a flow rate must be entered as input. The weir calculator uses the standard equation for computing flow over a weir:

$$Q = C_w L h^{\frac{3}{2}}$$

where:

Q – discharge flow (volume/time)

C_w – weir coefficient

L – width of weir (distance, ft or m)

h – head (distance, ft or m)

If a hydrograph has been computed using one of the supported hydrologic model, the peak flow for the hydrograph will be used as the default flow value if the hydrograph is selected prior to opening the dialog.

Selection of one of the predefined weir types automatically assigns the appropriate weir coefficient. A user-defined weir coefficient can also be entered, or the default value for one of the weir types listed modified. Weir calculations can also be used in combination with the detention basin calculator to define any outlet works of the basin/reservoir.

The weir calculator uses the *Hydraulic Toolbox* to perform the calculations.

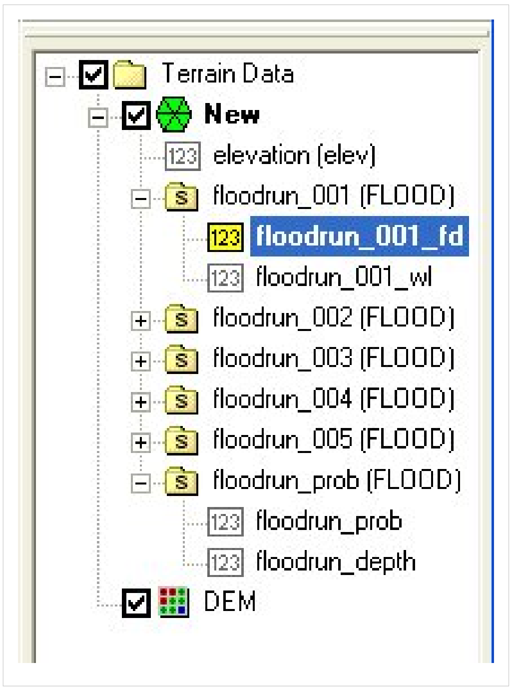
Related Topics

- Hydrologic/Hydraulic Calculators

3.6. Other Tools

Datasets

WMS was designed as a general purpose watershed modeling system. One of the main purposes of WMS is to provide a consistent interface for a variety of models and grid types. In order to accomplish this goal, input data for models and solution data (other than those models defined using topological trees) are handled in a simple, consistent fashion using datasets. They are accessed and managed using the *Project Explorer* as shown in the figure to the right.



A dataset is a set of values associated with each grid cell, or scatter point. A dataset can be steady state (one value per item) or transient (one value per item per time step). The following objects in WMS each have a scalar dataset list:

- TINs (elevations and solutions for flood plain delineation)
- 2D Grids (solutions for the GSSHA model)
- 2D Scatter Point Sets (could be a variety of things, but are used in WMS for storing hydraulic modeling water surface elevations that are used to delineate flood plains)

The commands for manipulating datasets are located in the *Data* menu of the respective modules, and as part of the *Project Explorer* where datasets are listed separately, or as part of solution folders.

Datasets are used for both pre- and post-processing of models. For example, a scalar dataset associated with a 2D grid can represent starting values of surface depth or values of hydraulic conductivity for a runoff modeling problem. Another dataset associated with the same grid may represent computed depth values. All datasets can be used to generate contours, color fringes, and animation sequences.

Active Dataset

The active dataset is specified by selecting it from the *Project Explorer*. Datasets may be stored in individual folders. In addition, if a transient dataset is highlighted, the time steps for the dataset are listed in the text box directly beneath the *Project Explorer* and one of the time steps is highlighted.

The values corresponding to the active dataset and time step are used whenever contour, or color fringe plots are generated. In addition, the entire range of time steps of the active dataset are used whenever animation film loops are generated. Whenever a new data set is created by importing from a file, interpolating, or using the data calculator, the dataset becomes the active dataset for the object.

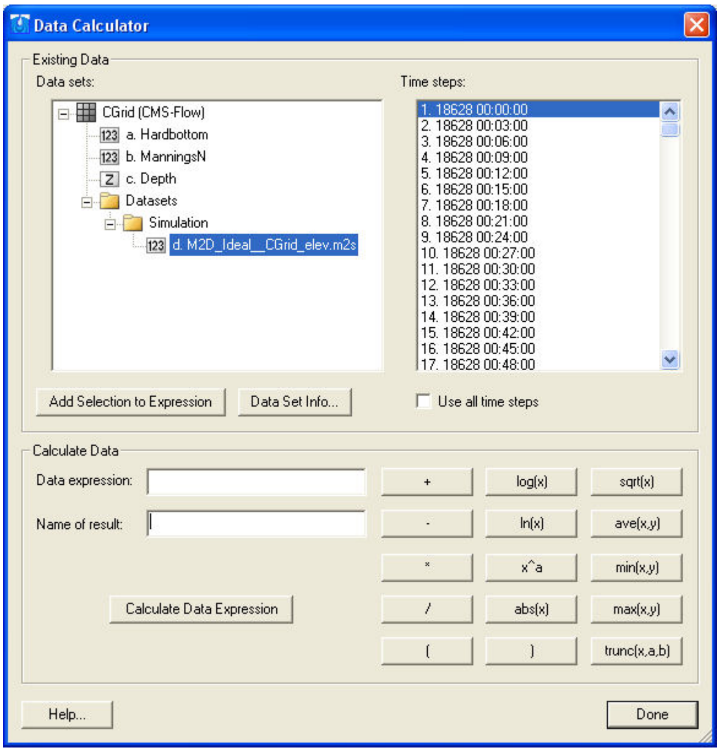
Data Calculator

- Performs mathematical calculations on scalar datasets
- Calculations can include any number of scalar datasets and user supplied numbers
- Useful for computing derived values such as Froude numbers
- Useful for comparing scalar datasets

More on Data Calculator

Dataset Info

The **Properties** command, available when right-clicking on a dataset in the *Project Explorer* will bring up a dialog listing some of the main characteristics of the active scalar dataset. These include statistics such as maximum, minimum and range as well as mean and standard deviation. A histogram of the dataset is also generated and plotted in the dialog.



Data Calculator

Deleting Datasets

Data sets can be deleted by selecting the dataset in the *Project Explorer* and selecting the **Delete** key on your keyboard. This deletes the binary copy of the dataset on disk. If the original dataset file was already in binary form, the file is not deleted.

Datasets that are part of solutions cannot be deleted unless the entire solution is deleted.

All datasets associated with an object are automatically deleted whenever the object is deleted or whenever the number of cells or vertices in the grid or scatter dataset is changed due to an editing command.

Elevations

Whenever a grid or scatter point set is created or read from a file, a scalar dataset is created containing the elevations of the cells or data points. Thus, there is always at least one dataset associated with each grid/TIN. This dataset cannot be deleted.

Mapping Elevations

For 2D grids it is often useful to change the values used for the elevations of the objects. For example, suppose a set of data values has been interpolated to a grid. The values can be displayed using contours. Another way to display the values is to map the dataset to the mesh elevations. This option further emphasizes the variation in the data when the grid is displayed in oblique view.

Any dataset can be mapped to elevations using the **Map to Elevations** command in the *Data* menu. The original elevations are always saved as a dataset so that the original elevations can be restored at a later time.

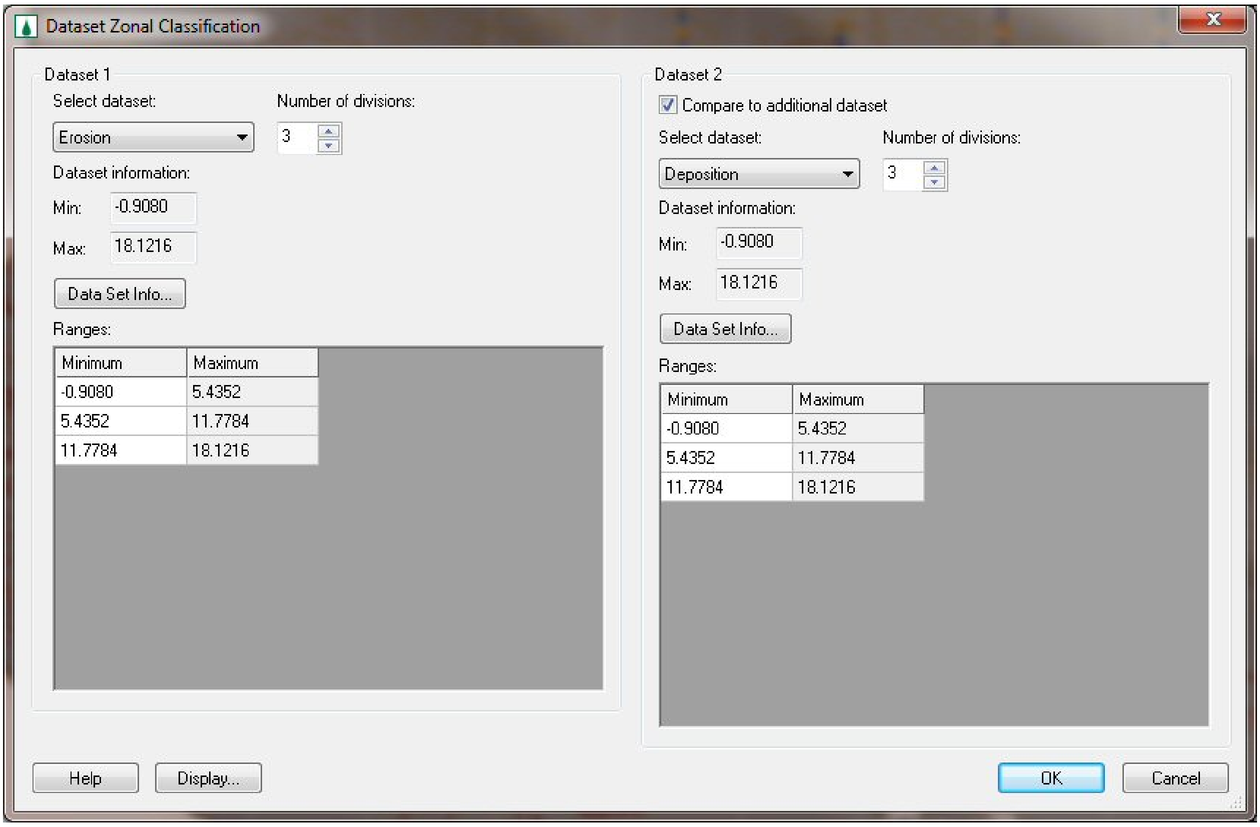
Solutions

Because some programs produce multiple datasets as part of a model run (GSSHA, stochastic modeling with flood plain delineation) it is convenient to be able to group all of the datasets together for reading, displaying, and managing. WMS uses the concept of a solution set which is collection of datasets for TINS or GRIDs and can also include hydrographs at single points. Solutions are identified in the *Project Explorer* as folders, with included datasets and provide another level of management for datasets.

Dataset Zonal Classification

The **dataset zonal classification** function classifies one or two datasets into a single integer dataset (also known as an index map in GSSHA models).

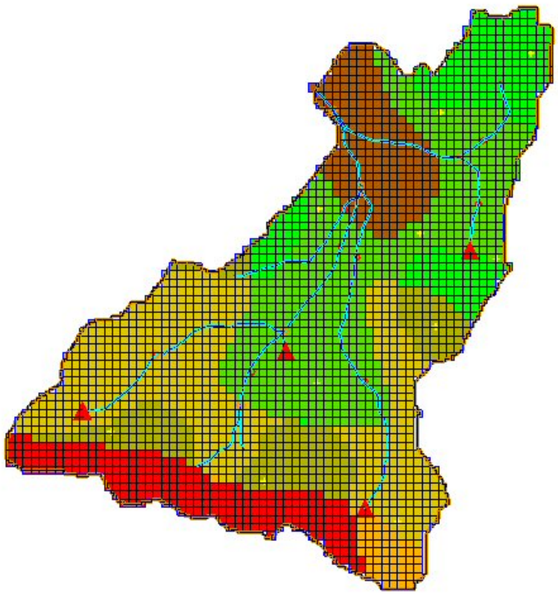
Sometimes, you need to classify the results of one or two datasets to create a single index map. For example, If you have one dataset showing sediment erosion and a second dataset showing sediment deposition, you might want to find areas with high sediment erosion and low deposition or vice versa. The **dataset zonal classification** tool helps you to find these areas. When you select the **dataset zonal classification** command from the *Data* menu, the following dialog appears:



You can modify the parameters in this dialog to generate the type of index map you want to generate. The *Select Dataset* option allows you to select a dataset on the 2D grid. Changing the *number of divisions* changes the number of minimum/maximum value pairs used to setup the dataset. After you select a dataset, you can select the **Dataset Info** button to show the dataset properties. You can also edit the minimum values for each range. You cannot set the minimum value for each range above the maximum for the current range or below the minimum for the previous range.

If you just want to classify values in a single dataset, you can turn off the *Compare to additional dataset* option (this is turned off by default) and define zonal classifications and colors for 2 or more ranges of numbers.

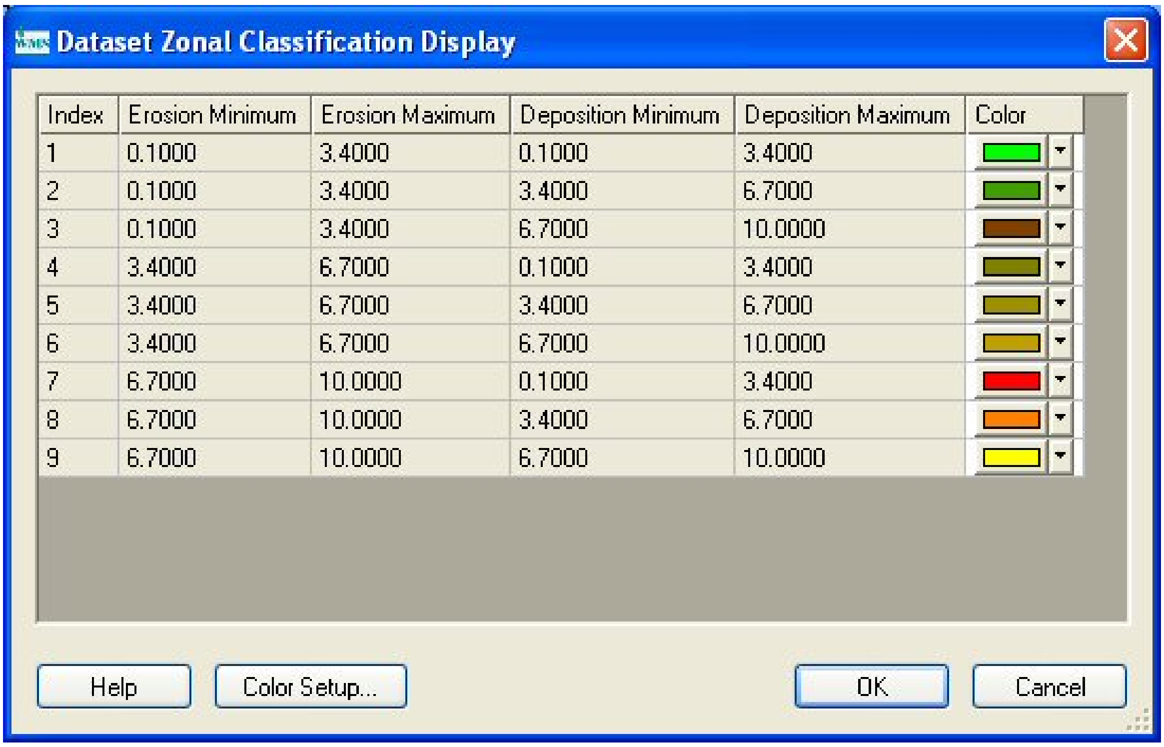
Clicking on the **OK** button generates a new index map with the name *Classified*, shown below. The colors are assigned to each grid cell in this dataset based on the colors specified in the *Dataset Zonal Classification Display* dialog.



Clicking on the **Display** button allows you to view the index map values and set the colors for each set of ranges on each of the two datasets.

Dataset Zonal Classification Display

The **Dataset Zonal Classification Display** dialog is used to define the index map colors for each index map ID used when combining two datasets.



Index	Erosion Minimum	Erosion Maximum	Deposition Minimum	Deposition Maximum	Color
1	0.1000	3.4000	0.1000	3.4000	
2	0.1000	3.4000	3.4000	6.7000	
3	0.1000	3.4000	6.7000	10.0000	
4	3.4000	6.7000	0.1000	3.4000	
5	3.4000	6.7000	3.4000	6.7000	
6	3.4000	6.7000	6.7000	10.0000	
7	6.7000	10.0000	0.1000	3.4000	
8	6.7000	10.0000	3.4000	6.7000	
9	6.7000	10.0000	6.7000	10.0000	

You can either change individual colors or the colors for the color index values and the surrounding colors will be interpolated based on the index value colors. To change the index value colors, select the **Color Setup** button and define the colors used for low and high values of combinations of the one or two datasets that are selected.

Related Topics

- File I/O
- Data Calculator

Deleting Data

The different ways to delete data in WMS makes it very easy to eliminate any portion or all of the data associated with a given model. One important thing to remember is that deleting in WMS does not cause the data to be permanently removed or erased from the hard drive, but rather eliminates it from core memory for a specific run of WMS. For example if you read a TIN in from a file and then delete the TIN it will no longer appear in WMS, but the file that was read in is not deleted from the hard disk.

The methods that can be used to delete data from WMS include:

- Deleting all data and starting over with the **New** command in the *File* menu.
- Deleting selected items (vertices, triangles, arcs, etc.) with the **Delete** command in the *Edit* menu.
- Deleting all data associated with a given module (i.e. all terrain data in the terrain data module) using the **Delete All** command in the *Edit* menu.
- Deleting object specific data such as all feature objects, drawing objects, or DXF data with the **Delete** command found in individual menus, or by right clicking on the object in the Project Explorer and choosing **Delete**.

Related Topics

- New
- Delete All
- Delete

Gages

The **Gages** command can be used to establish the position and rainfall accumulation for rainfall gages. Gages may be entered with or without a terrain model present, but if a terrain model is present then gage weights (using the Thiessen polygon method) for each basin are automatically computed when the **Compute Basin Data** command is chosen. When a terrain model is not present the gage weights for each basin must be entered manually from within the Precipitation dialog.

In order for gage information to be used during the simulation, the basin precipitation type must be set to gage in the *Precipitation* dialog.

Defining/Editing Gages

The *Gage* dialog consists of a list of defined gages as well as the fields necessary to define a new gage. When a gage is selected from the list, its values are copied into the appropriate fields for editing. When the New button is chosen, a new gage is created with default values. If the **Copy** button is chosen a new gage is created with the values of the currently selected gage. The **Delete** button can be used to remove the currently selected gage.

The Gage Position

If a terrain model is being used then the X and Y position in a consistent coordinate frame need to be entered in the appropriate edit fields. Once there are three or more defined gages, the Thiessen polygon network can be displayed. If a terrain model is not being used then the weights must be assigned manually and the position information is not important. In this case the Define rain gage location toggle should be turned off.

The rain gage coverage may also be used to define the position of the gages. Each feature point in the rain gage coverage is converted to a gage location and will automatically appear in the *HEC-1 Gages* dialog. See article Rain Gage for more information on the rain gage coverage.

The Gage Type

A gage may be a storm total and/or temporal distribution (recording) station type. Recording stations allow for a continuous (incremental or cumulative) rainfall accumulation to be entered. The storm total station only allows for a single rainfall value for the event. A station may be both types if the distribution corresponds to the storm total value entered. However, a more typical situation is to have several stations for which only a storm total is known and to which some type of standard distribution will be applied. To accomplish this, the storm total stations may be entered along with one "imaginary" gage that is used to define the distribution. When automatically computing weights, only storm total stations are used in the Thiessen network, and the distribution for each storm total station is found by locating the nearest distribution gage.

Gage Tools

Gages can also be created and deleted using the *Gage* tools. The gage tools appear in the dynamic portion of the *Tool Palette* of each of the modules which support gages. The tools are as follows:

 **The Create Gages Tool**

The **Create Gages** tool is used to interactively create gages in the *Graphics Window*. When this tool is active, a new gage is created by clicking in the *Graphics Window* at the desired location of the gage (the *Graphics Window* must be in plan view when creating gages). The xy coordinates of the gage are defined by the cursor position and the user is prompted for the z coordinate. The x, y, and z coordinates of a new gage can be edited using the *Edit Window*. In addition, once a gage has been defined with the **Create Gages** tool, the gage can be edited using the *Gages* dialog.

The Select Gages Tool

The **Select Gages** tool is used to select previously defined gages. A set of selected gages can be deleted by hitting the *DELETE* key or by selecting the **Delete** command from the *Edit* menu. The coordinates of a selected gage can be edited using the *Edit Window*. The location of a gage can also be edited by holding down the mouse button when a gage is selected and dragging the gage. This tool is also used to control what is plotted in the *Gage Plot Window*. Only the curves associated with selected gages are plotted.

Making Gage Plots

Once a set of gages has been defined, a plot can be generated in a *Hydrograph Window* representing the variation vs. time of any of the transient data sets associated with grids interpolated to the gages. The curves are plotted only for gages which have been selected using the **Select Gages** tool. This makes it possible to quickly change the combination of curves plotted.

To display a plot in the *Hydrograph Window*, right-click on the gage you want to plot and select either **Plot All** or **Plot Selected**. Selecting **Plot All** plots all the datasets for the selected gage(s). Selecting **Plot Selected** brings up a dialog that allows you to select which datasets you want to plot for the selected gages.

Visualizing Data

One of the most important steps in any modeling problem is calibration. During the calibration phase, an attempt is made to model a set of conditions which have been known to exist at a site and for which measured data (surface depth, infiltration) are available. The geometry, resolution, and input parameters of the model are adjusted until the output computed by the model is reasonably close to the measured data.

The calibration stage can be the most tedious and time-consuming portion of the modeling process. In order to make the calibration stage more efficient, a set of tools for managing gages has been provided in WMS. A "gage" is an xyz point defined by the user representing a location where field data has been collected (e.g., a gaging station) or simply a point of interest in the model. Once a set of gages has been defined, whenever a transient dataset is imported to WMS to a grid, the dataset is interpolated to each gage and a curve is drawn in the *Hydrograph* window representing the variation of the dataset with time at each gage. The plot can be customized to include any combination of gages and datasets. Field data can be imported from text files and plotted for comparison with computed curves.

Gages and gage plots are supported only in the 2D Grid module.

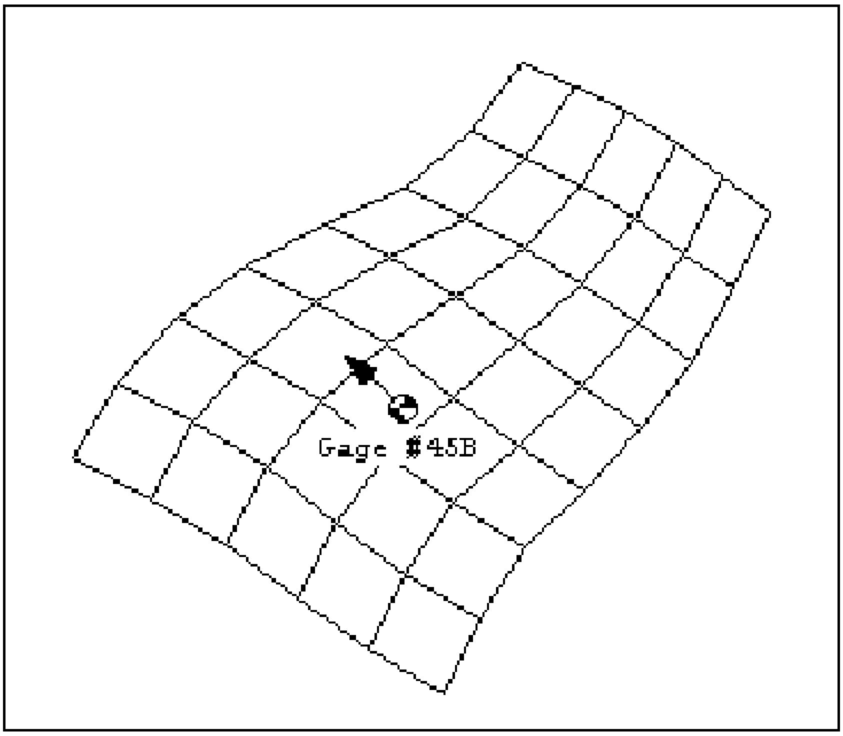
Gages Dialog

A set of gages can be created by selecting the **Gages** command from the *Data* menu. This command activates the *Gages* dialog. All existing gages are listed in the text box in the upper left corner of the dialog. One of these gages is highlighted at all times. The name, color, and location of the highlighted gage can be edited using the controls on the right hand portion of the dialog. The color and name are used in displaying the gage in the *Graphics Window*.

A new gage can be added to the list of gages by selecting the **Create New** button beneath the list of gages. A highlighted gage can be removed from the list by selecting the **Delete** button. A set of gages can be imported from a text file by selecting the Import button. A set of measured curves can be included in the gage file for comparison with computed curves. A set of gages created within WMS can be exported to a file for future use by selecting the **Export** button.

The *Interpolation Method* options in the lower right corner of the *Gages* dialog controls how datasets are interpolated to the gages for curve plotting. If the *Interp. from neighboring nodes/cells* option is chosen, the datasets are interpolated from the nodes or cells in the vicinity of the gage using a simple inverse distance weighted interpolation scheme.

Gages are plotted in the *Graphics Window* as shown below. The name is plotted just below the gage symbol. Each component of the gage can be turned on or off or resized using the Display items in the lower left corner of the *Gages* dialog.



Related Topics

- Datasets

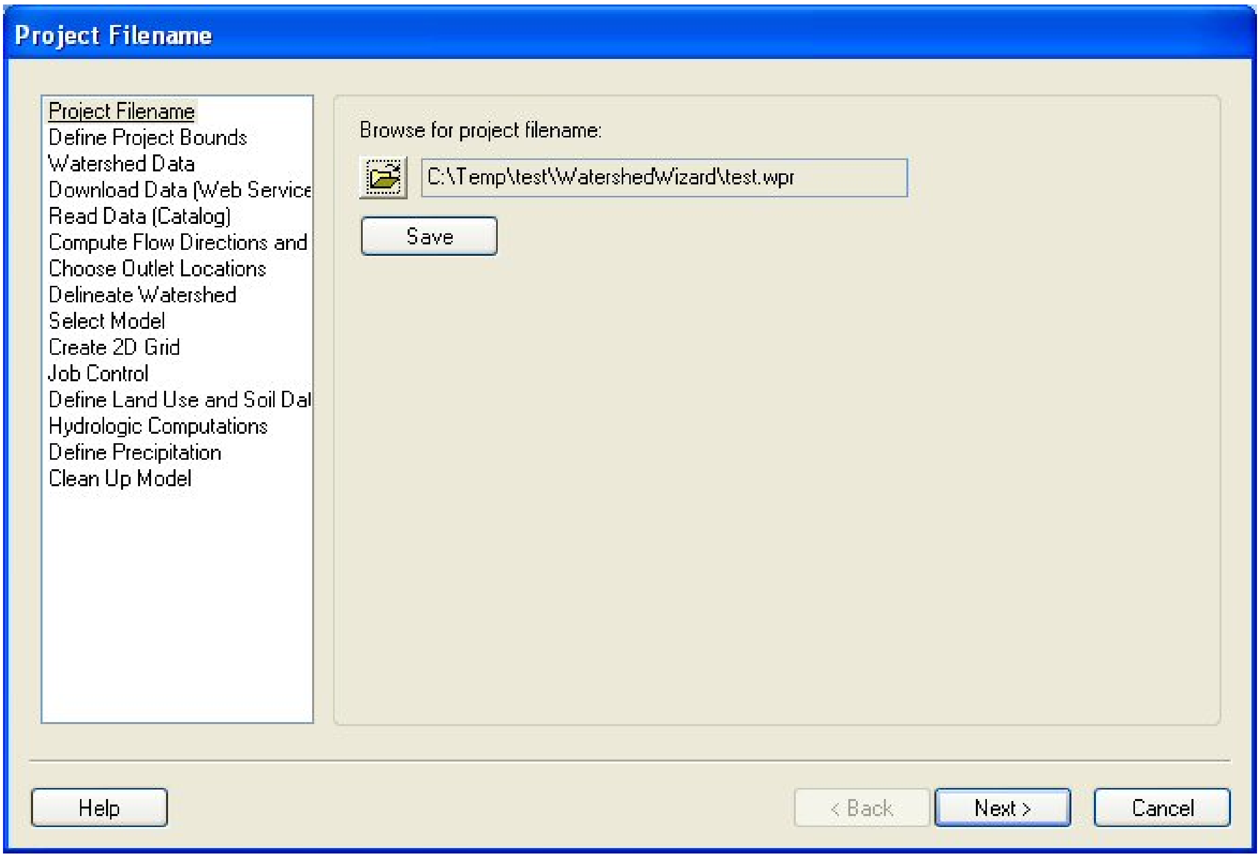
Hydrologic Modeling Wizard Overview

The WMS Hydrologic Modeling Wizard ( ) is a simple tool that walks you through all the steps involved in creating a hydrologic model. It can be accessed by selecting the hydrologic modeling wizard tool in the get data toolbar ( ).

Steps

The following steps are included in the hydrologic modeling wizard:

Project Filename

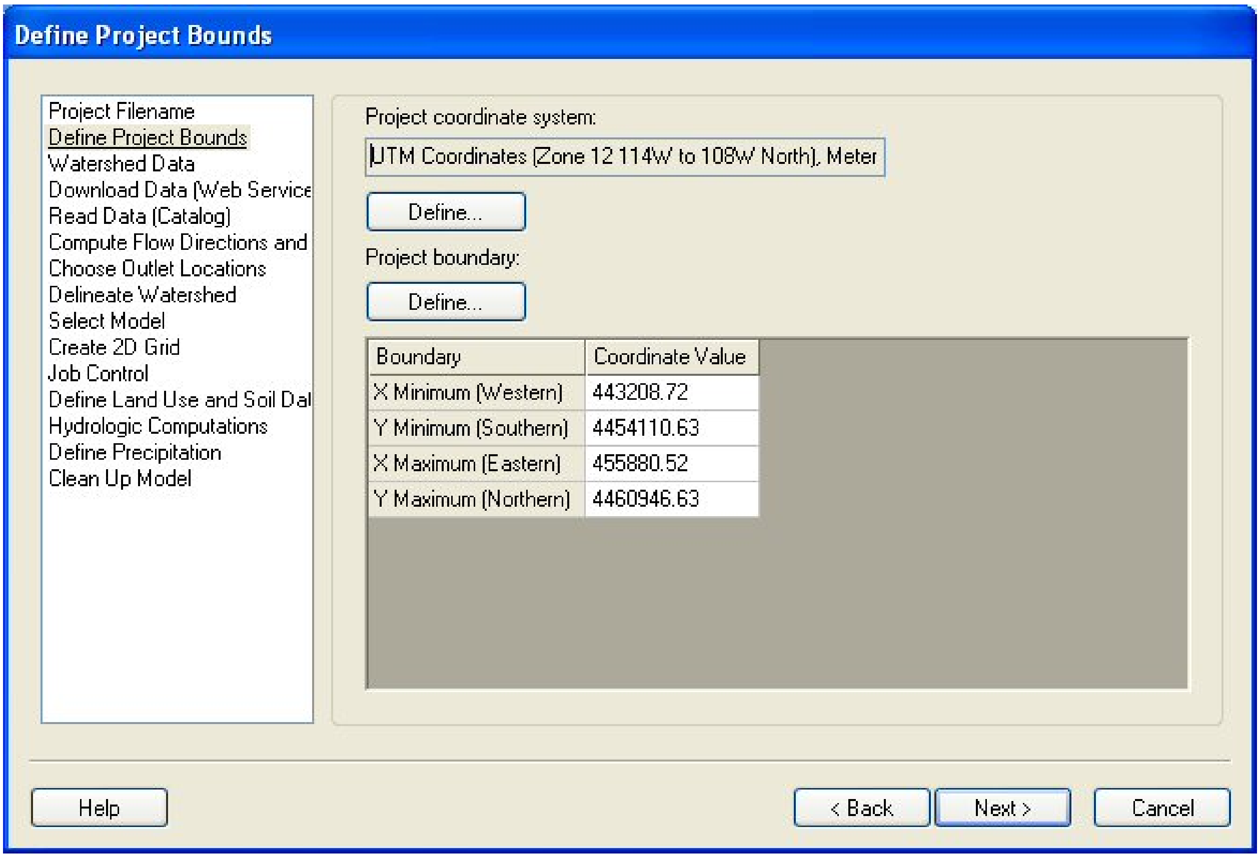


The project filename step is used for defining a project filename. This filename is used when saving files from the hydrologic modeling wizard.

Help

Save – This button saves the project in your current state.

Define Project Bounds



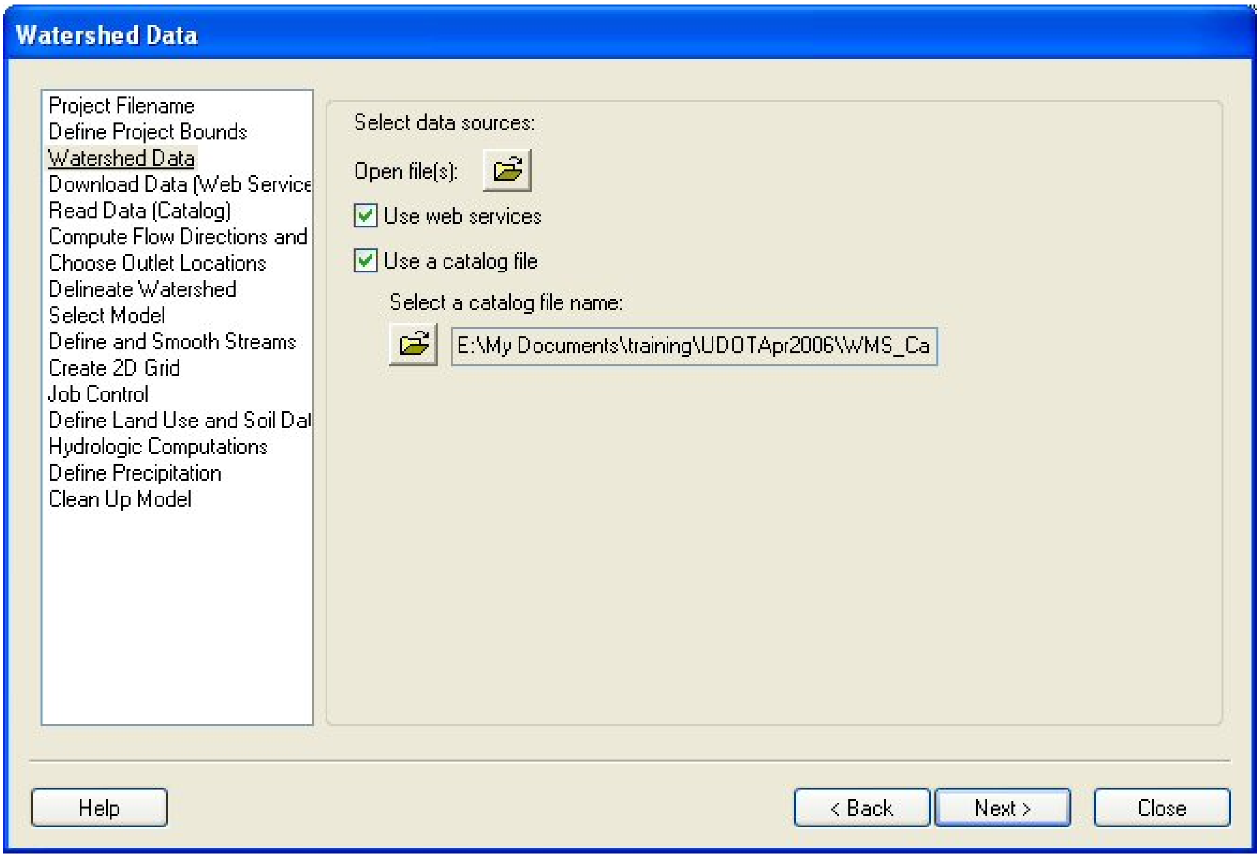
The define project bounds step is used for defining your project boundary.

Help

Top **Define** button – This button is used to define your project coordinate system, if it is known.

Bottom **Define** button – This button is used to define your project boundary in the Microsoft Virtual Earth web service client. Find the area you are interested in modeling and WMS will enter the minimum and maximum coordinates of the box you have defined in the Virtual Earth window.

Watershed Data



The watershed data step is used for defining how you will read data into WMS. You can obtain data using the WMS web service client, a catalog file, or by simply opening files.

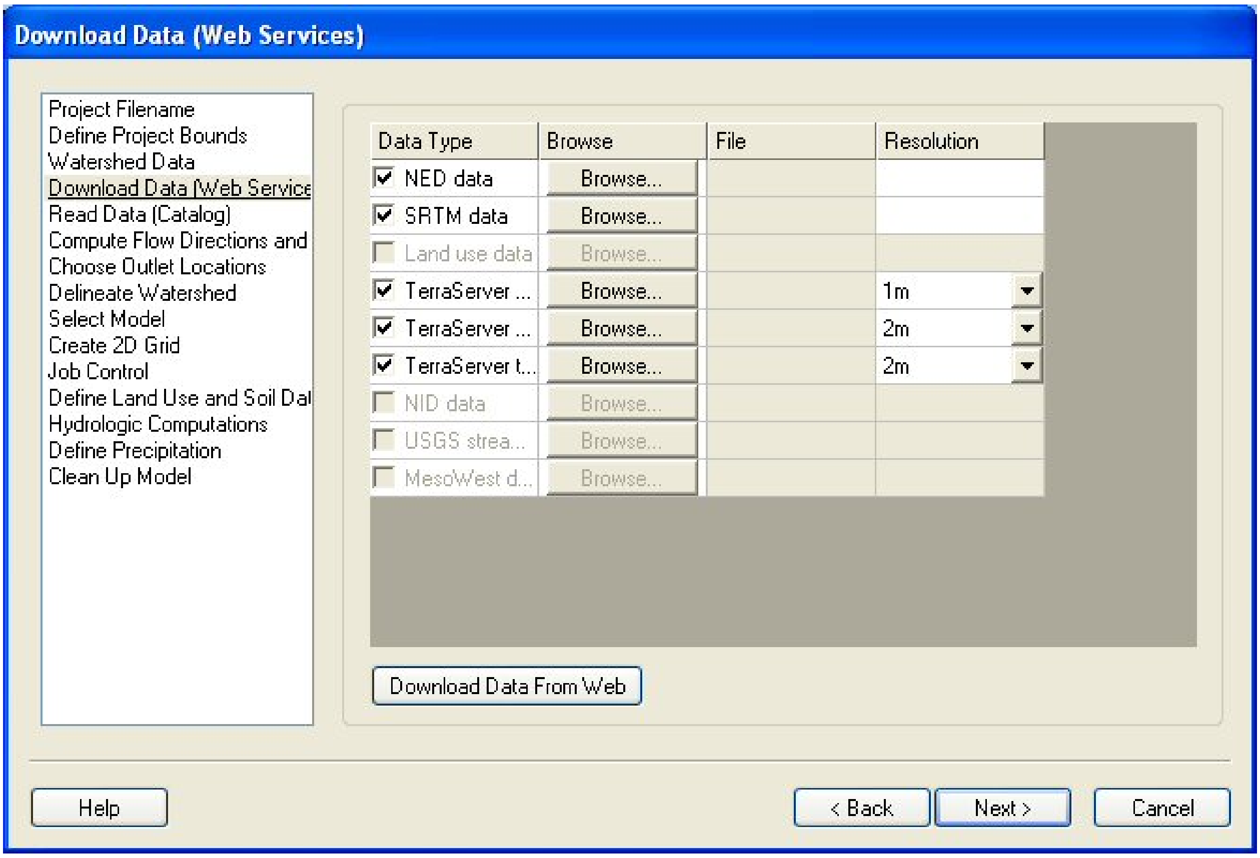
Help

Open file button – Use this button in the same way as the **File | Open** command in the *WMS* menus. After selecting this button, a file browser appears and WMS opens the files you select.

Web services – WMS will use the built-in web service client for obtaining data for watershed modeling.

Catalog file – WMS will use a catalog file for obtaining data for watershed modeling.

Download Data

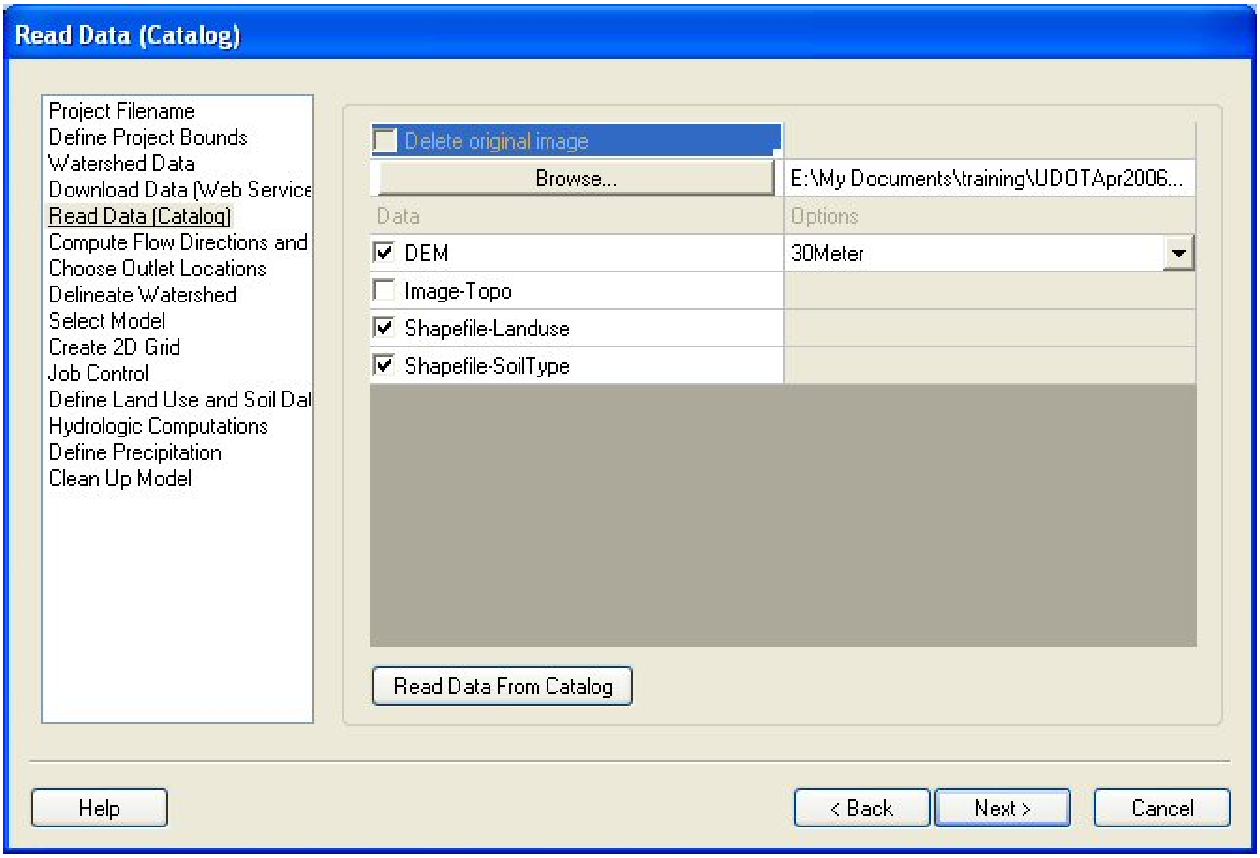


The download data (web service client) step is used to download data over the internet using the WMS web service client if a web service is available. NED and SRTM data may be downloaded from this dialog, but coverage may not exist or download times may be slow for your hydrologic modeling area. The NED and SRTM data are obtained from USGS databases. The Terraserver data are obtained from the Microsoft Research (MSR) Maps web service.

Help

Download Data From Web – This button downloads the selected datasets for your selected modeling area from the web service providers.

Read Data

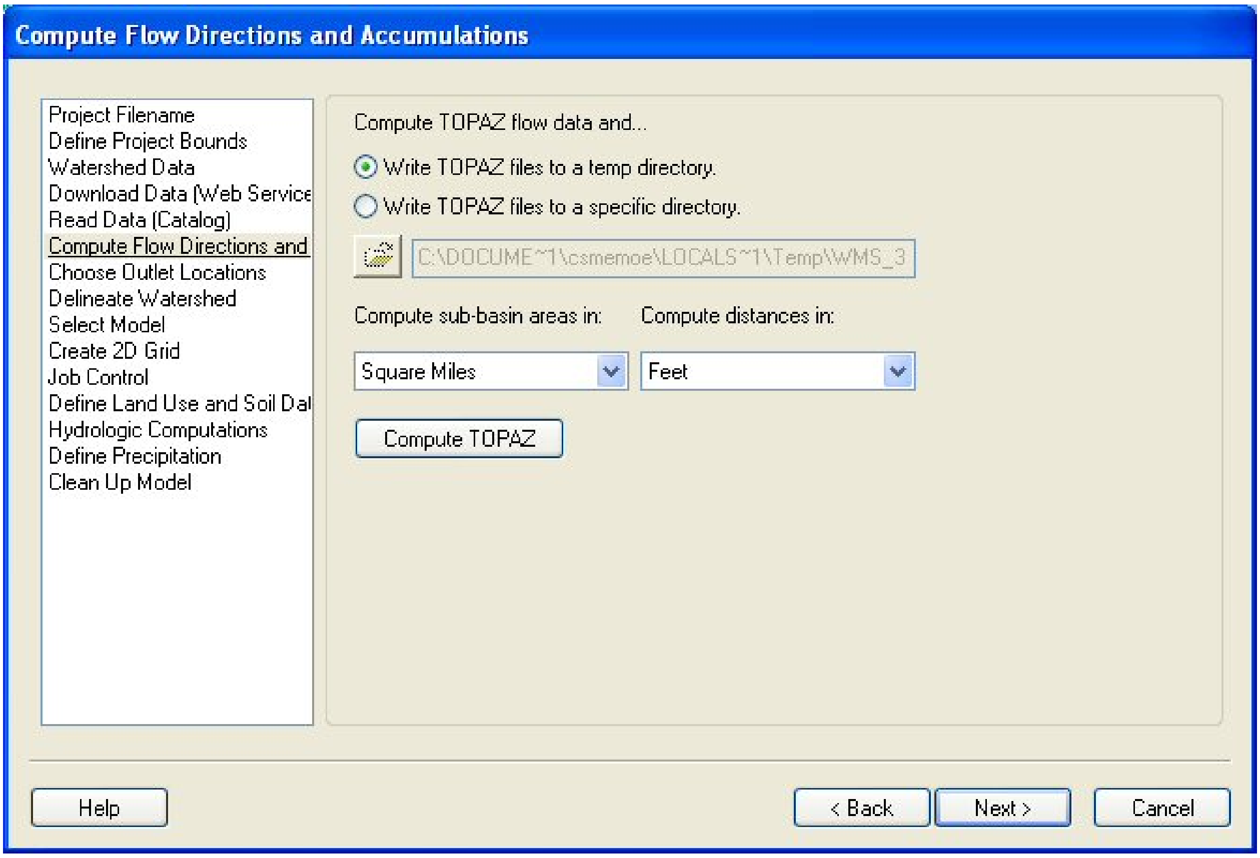


The read data from catalog step is used to select and read data from a WMS catalog.

Help

Read Data From Catalog – This button reads data from the catalog inside your selected project boundary.

Compute Flow Directions and Flow Accumulations



The Compute Flow Directions and Flow Accumulations step is used for running TOPAZ to compute flow directions and accumulations.

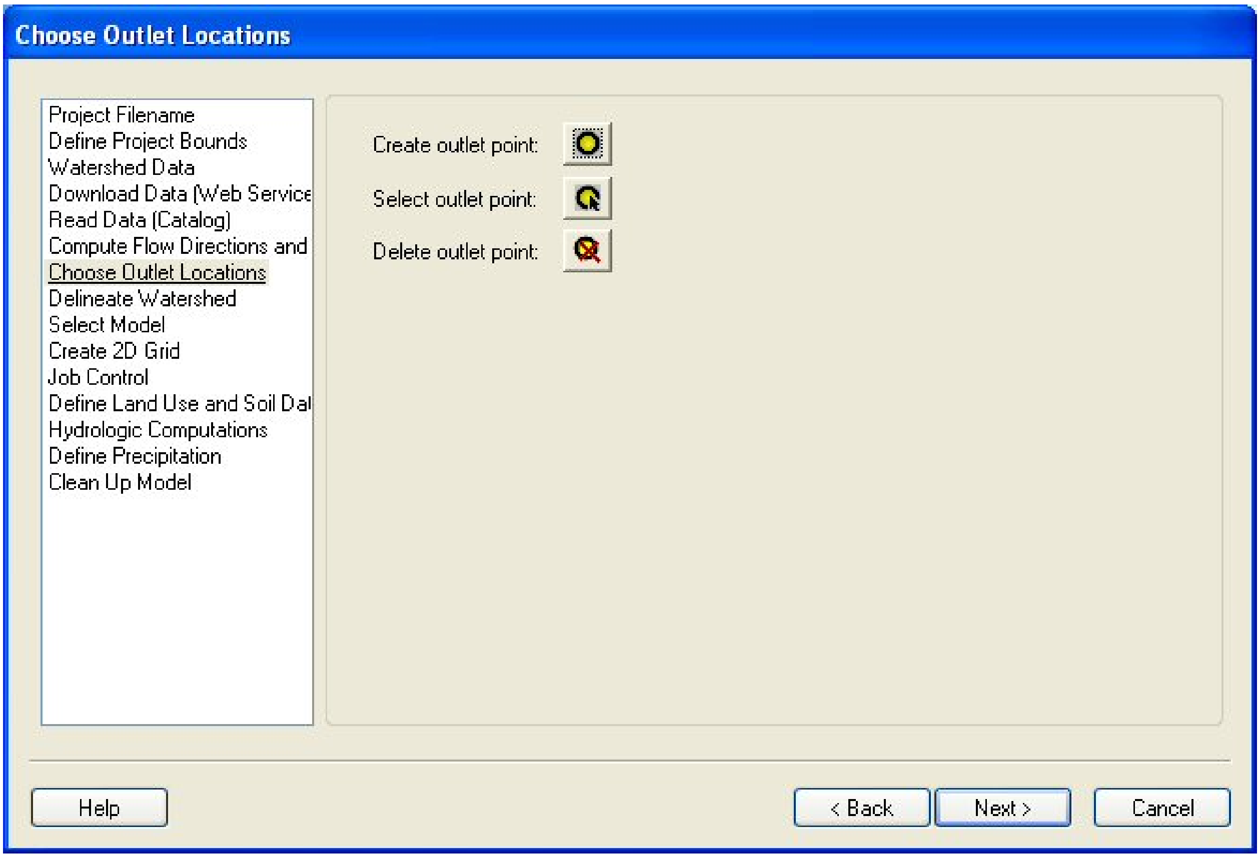
Help

Compute TOPAZ flow data and... – You can choose either to write your TOPAZ output to a specific directory or to the temp directory. Most frequently, you will just write your TOPAZ output to a temp directory.

Units – Use the units combo boxes to define your model's computation units.

Compute Topaz – This button runs TOPAZ using the settings you have selected.

Choose Outlet Locations



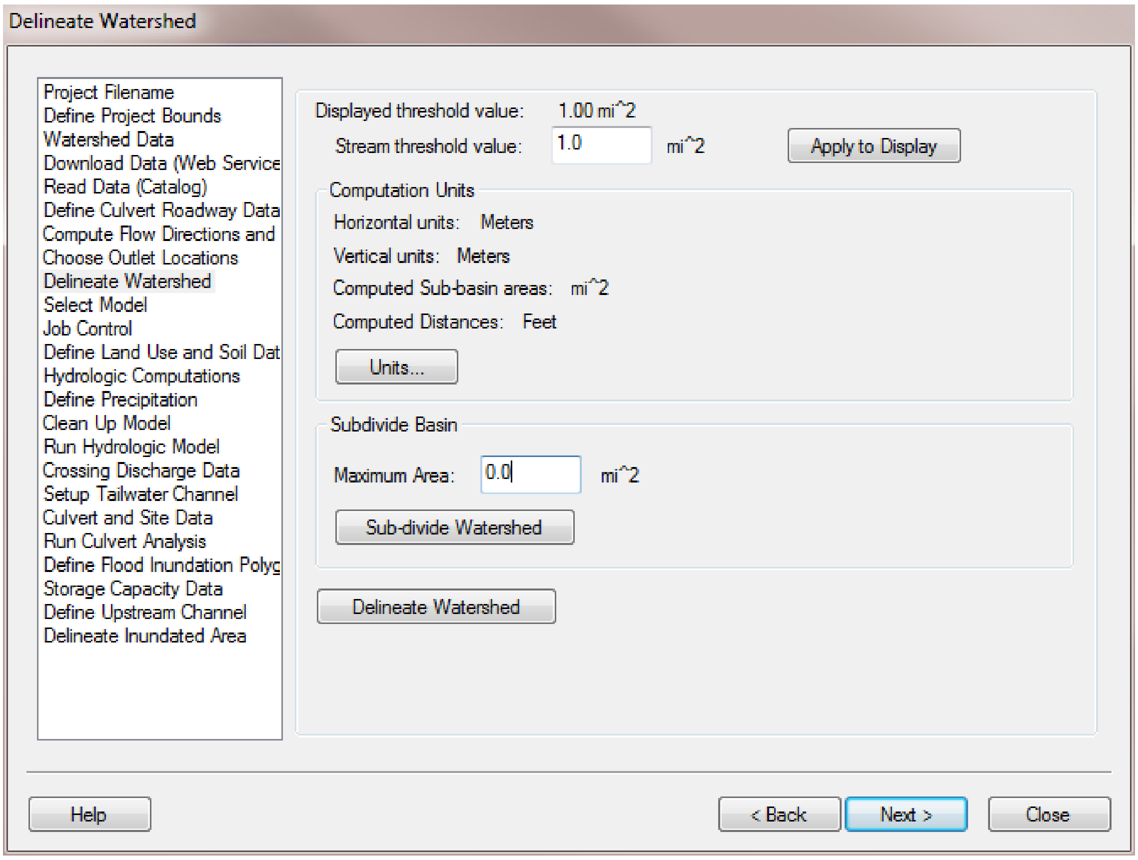
The Choose Outlet Locations step is used to define, move, or delete outlet locations.

Help

Create, **Select**, and **Delete** outlet point buttons – To create an outlet point, select the tool you want to use and then click on a flow accumulation cell. To select an outlet point, select the tool and click on a point or drag a point. To delete a point, select a tool and select an outlet point to delete.

In the HY-8 Modeling Wizard, this dialog has a **Define Outlets from Culvert Locations** button. This button defines an outlet at the point on a flow accumulation cell that is closest to the upstream end of the culvert arc.

Delineate Watershed



The Delineate Watershed step is used to delineate a watershed.

Help

Stream threshold value – This value is used to modify the stream density. Lower values will cause the streams to be more dense while higher values will create fewer streams in the completed model.

Apply to Display – Select this button to apply the stream threshold value entered to the display.

Create Tc Coverage – If this toggle is selected, a Time Computation coverage containing an arc with the longest flow path is created after delineating the watershed and sub-basins.

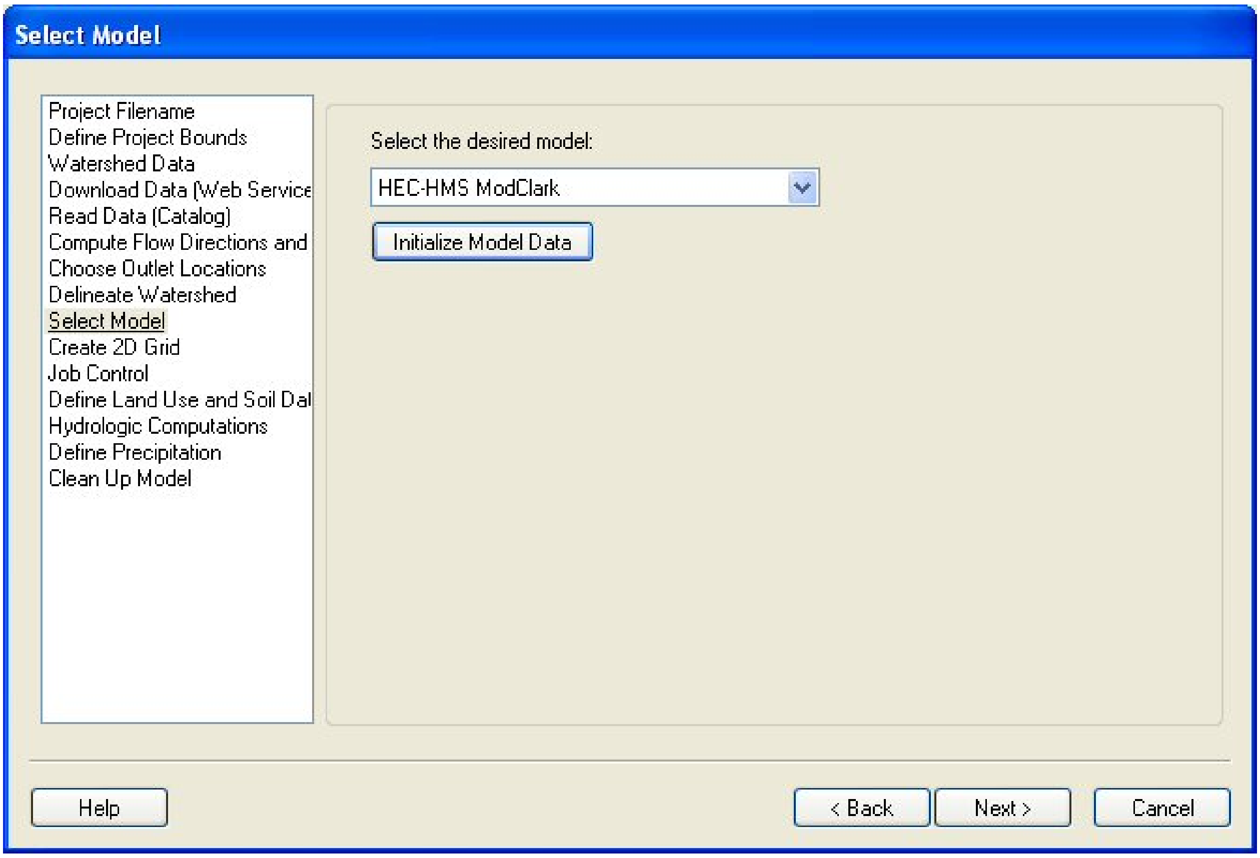
Units – This button is used to define your model and computation coordinates and units. You can turn on the option to create a Tc coverage and change which data is computed by selecting the *Drain Data Compute Opts...* button in the Units dialog.

Sub-divide Watershed – This button subdivides your watershed into sub-basins based on the maximum sub-basin area entered.

Delineate Watershed – This button delineates the watershed and computes each sub-basin's data based on the selected watershed delineation parameters.

After delineating a watershed, it is possible to manually edit the extents of the watershed.

Select Model

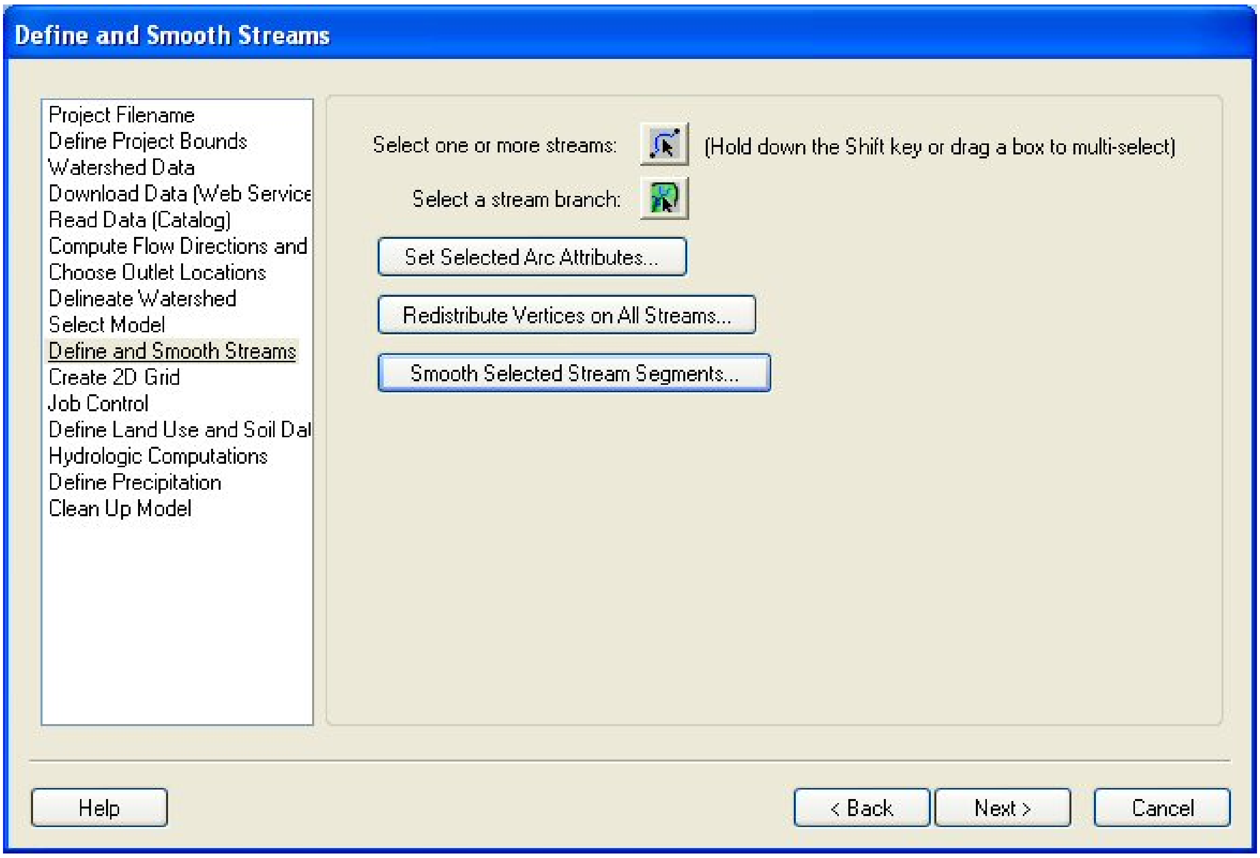


The Select Model step is used for defining which model you are running with your delineated watershed.

Help

Initialize Model Data – Click on this button to set your model to the selected option, initialize your model data to default values, and set the module to the correct module.

Define and Smooth Streams



When you are building a GSSHA model, you need to define the parameters for and smooth all the streams in your model before building the 2D Grid. When the 2D grid is generated, the elevations on cells intersecting stream arcs will be made to match the stream arc channel depth elevations (the stream arc channel depth elevation along the stream = the stream elevation + the channel depth elevation entered in the GSSHA Feature Arc Type dialog).

Help

Select one or more streams – This button selects the select feature arc tool. After this tool is selected, you can select one or more stream arcs and edit the attributes or smooth the selected stream arcs.

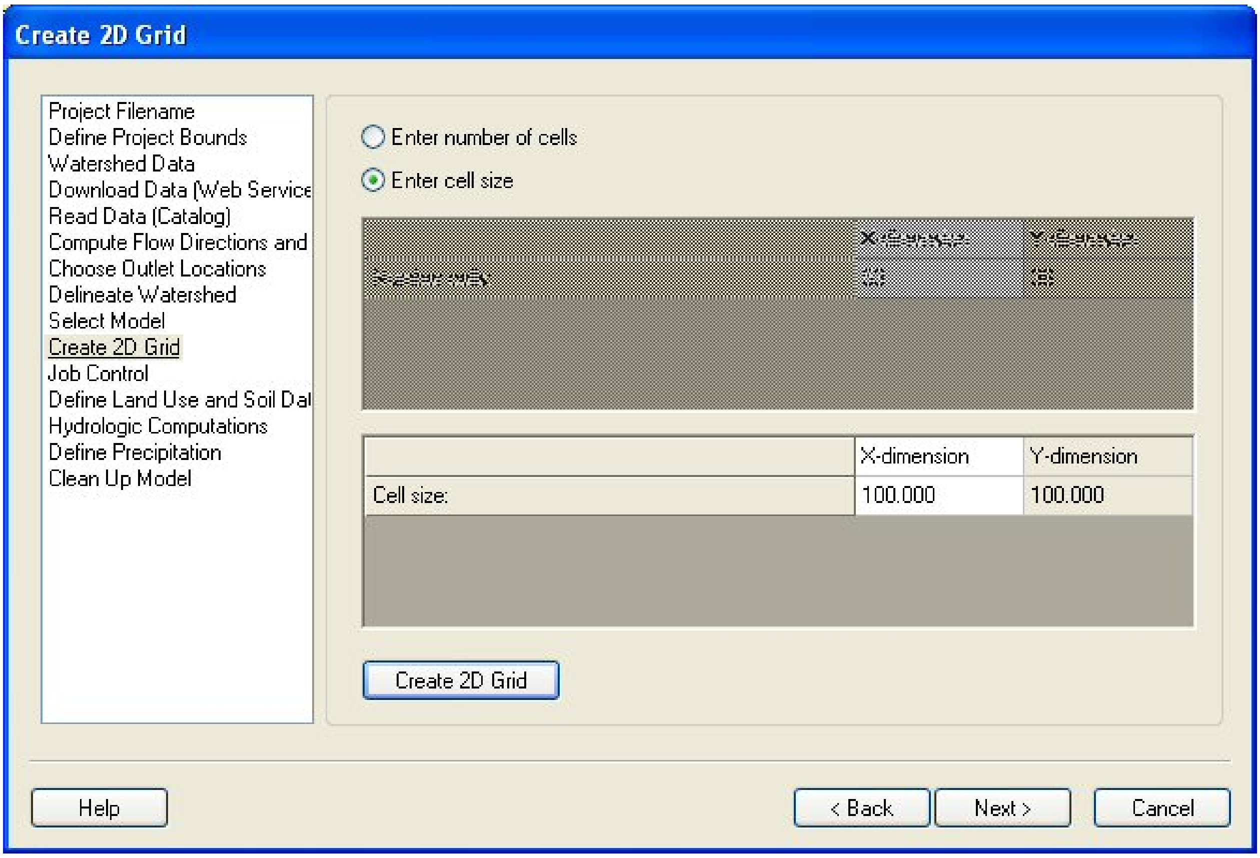
Select a stream branch – This button selects the select stream branch tool. This is a specialized tool that selects a stream arc and any arcs upstream from the selected arc. This tool is useful for selecting an entire stream network or a branch of a stream network.

Set Selected Arc Attributes – This button brings up the GSSHA Feature Arc Type dialog. This dialog allows you to set the attributes for all the selected arcs.

Redistribute Vertices on All Streams – Selecting this button will first switch your active coverage to be the GSSHA coverage. If no arcs are selected, all the stream arcs will be selected in the GSSHA coverage. If arcs are selected, no additional arcs will be selected. The Redistribute Vertices dialog will appear, allowing you to set a new spacing for the vertices on the selected arcs.

Smooth Selected Stream Segments – This button will bring up the Smooth GSSHA Streams dialog for the selected arc(s). Before selecting this button, you should select a set of non-branched stream arcs. The smooth streams dialog has several options for modifying and smoothing the streams in the selected branch.

Create 2D Grid

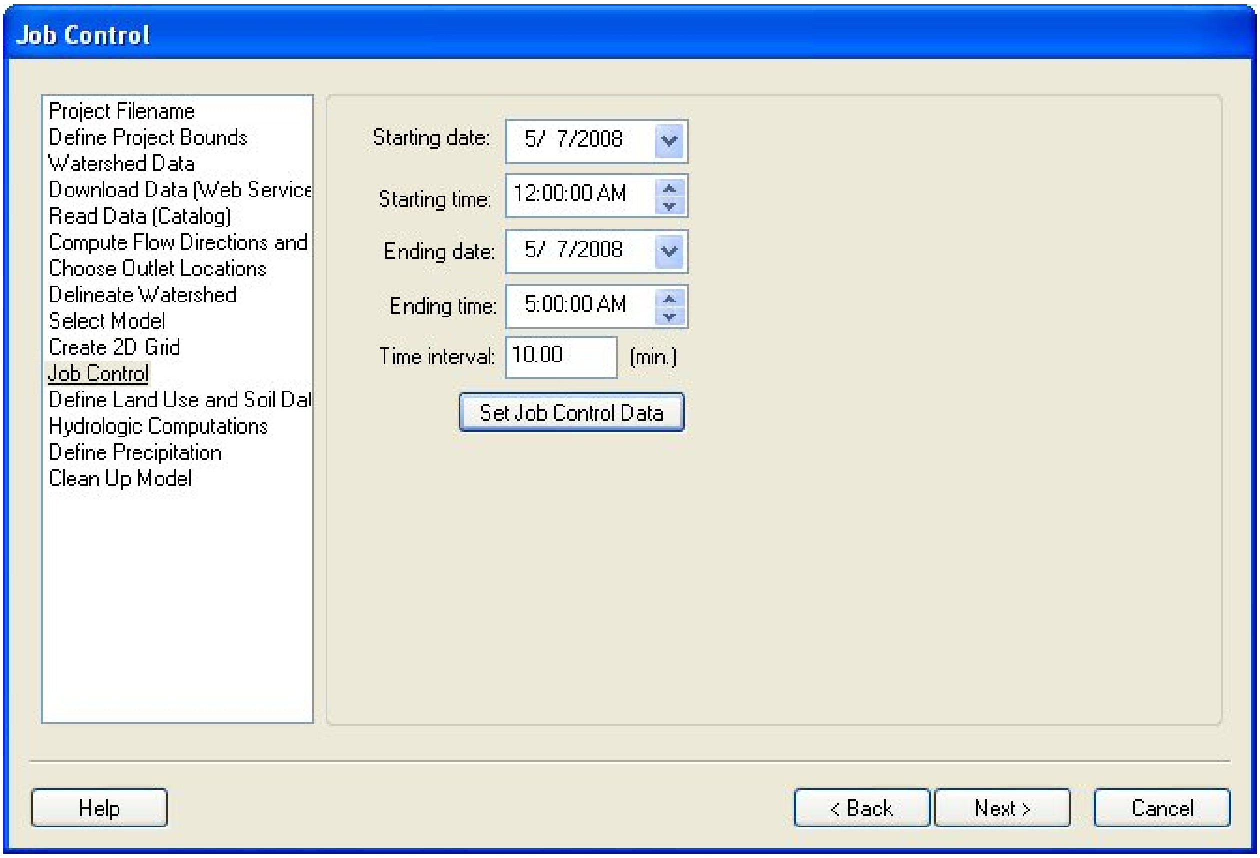


The Create 2D Grid step is used to create a 2D grid from your watershed boundary. Select a grid cell size or number of cells and select the Create 2D Grid button to create a 2D grid.

Help

Create 2D Grid – This button creates a 2D grid based on the cell size or number of cells entered. For the GSSHA or ModClark models to run correctly, the X and Y dimensions of the 2D grid cells must be equal.

HMW Job Control

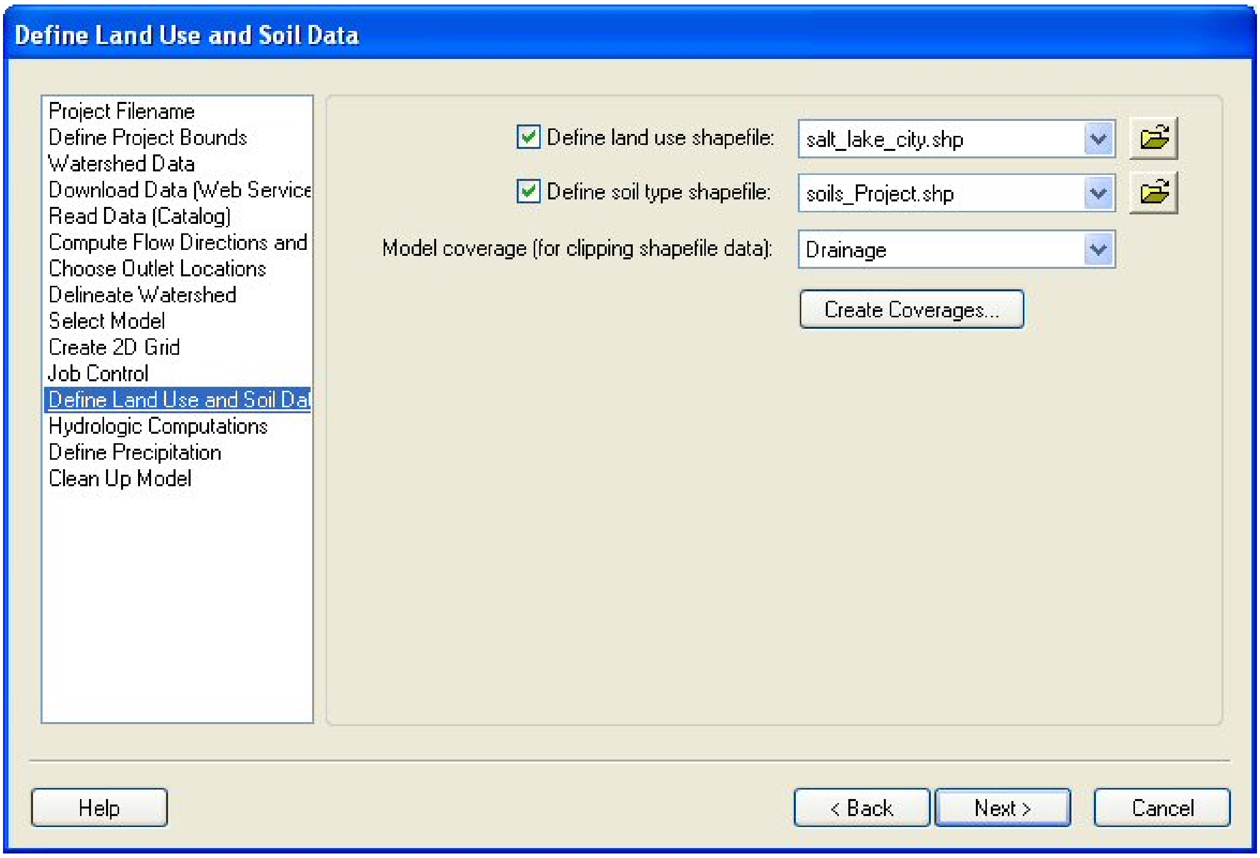


The job control step is used define the time parameters for running your model. Define the start and end time and date and select **Set Job Control Data** to set your job control parameters for the model you have selected.

Help

Set Job Control Data – Sets your job control parameters for the model you have selected.

Define Land Use and Soil Data

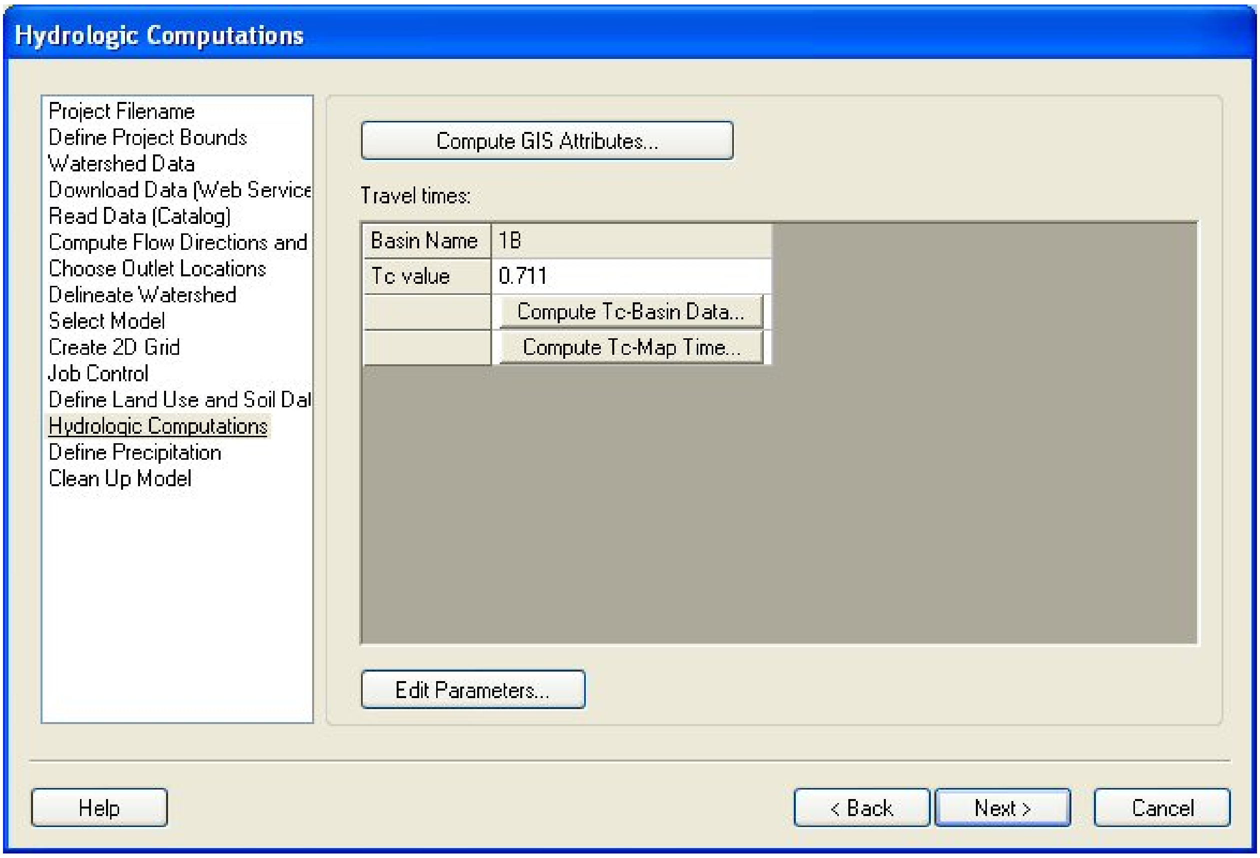


The Define Land Use and Soil Data step converts GIS Module shapefiles to data in the WMS Map module. WMS uses the boundary of your watershed to clip the shapefile data for the selected files.

Help

Create Coverages – Define your land use and soil type shapefiles and a model coverage and your land use/soil data will be transferred to coverages in the map module and clipped to your watershed boundary when you select this button.

Hydrologic Computations



The Hydrologic Computations step is used to compute hydrologic parameters for your watershed and sub-basins.

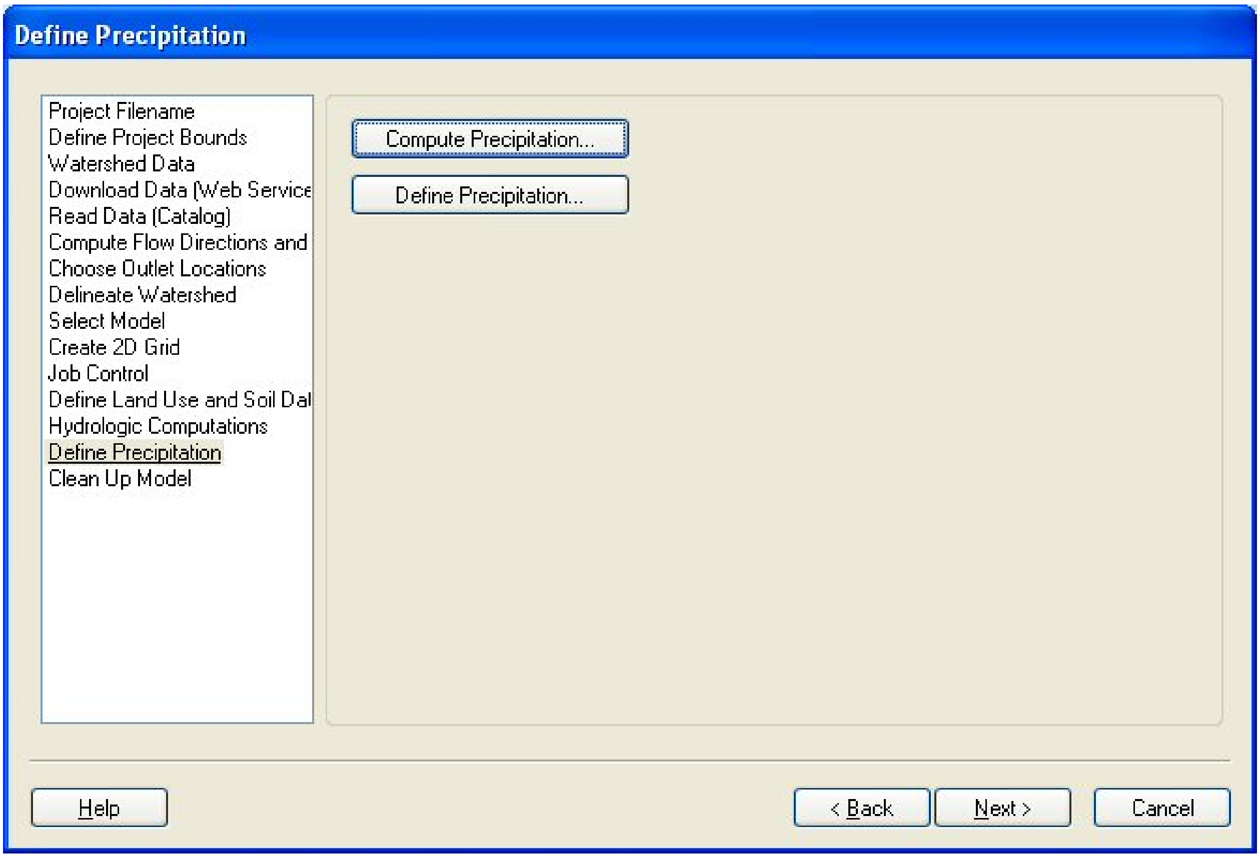
Help

Compute GIS Attributes – Depending on the model you are using, this button brings up a dialog allowing you to compute attributes from your land use and/or soil type data.

Travel times – This spreadsheet allows you to compute the Time of Concentration for all the sub-basins in your watershed using either the basin data or map data method.

Edit Parameters – This button brings up the edit parameters dialog for your model, allowing you to change modeling parameters for all the sub-basins in your watershed.

Define Precipitation



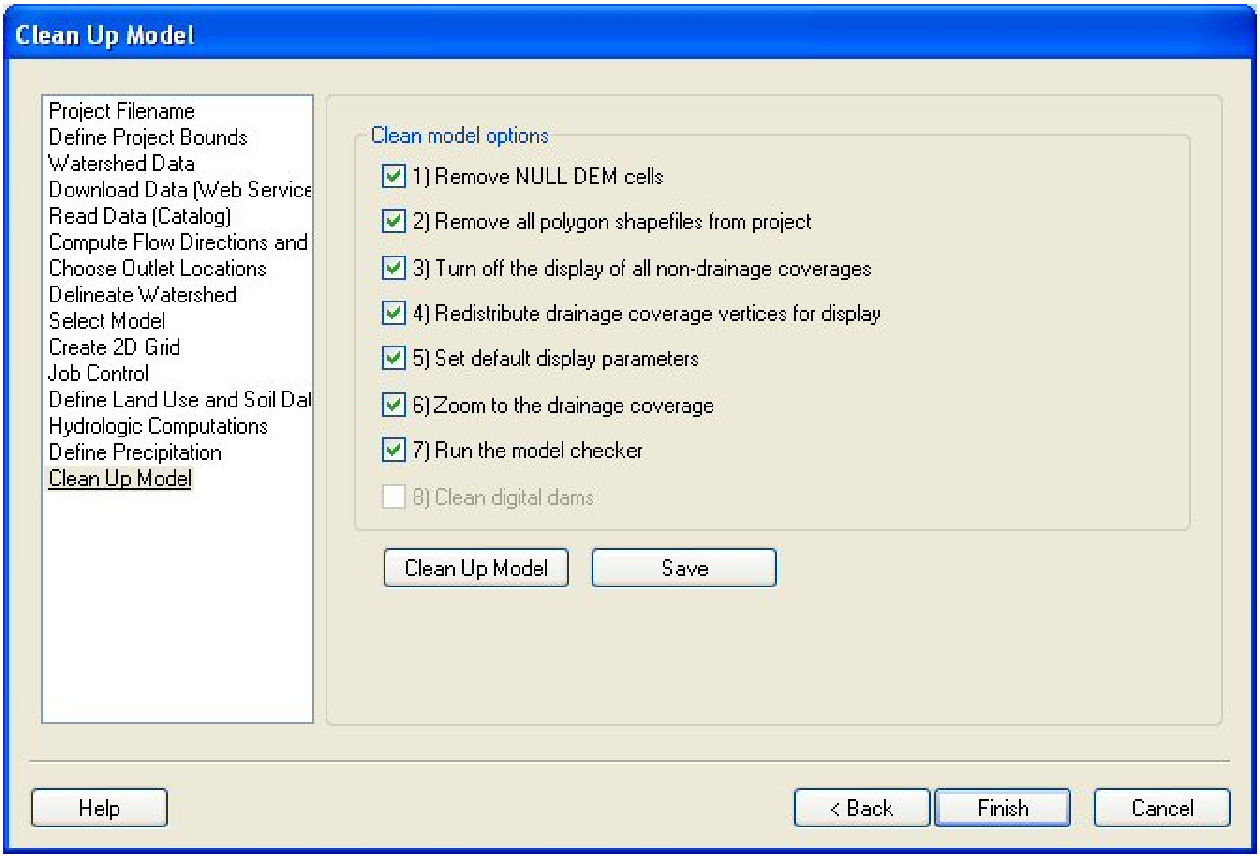
The Define Precipitation step is used to compute and define precipitation for your model.

Help

Compute Precipitation (certain models only) – This button allows you to compute the precipitation for your model from a NOAA Atlas 2 or any other type of rainfall grid.

Define Precipitation – This button allows you to define precipitation for your selected model.

Clean Up Model



The Clean Up Model step is used to clean up your model by doing tasks that are typically done when your model is finished.

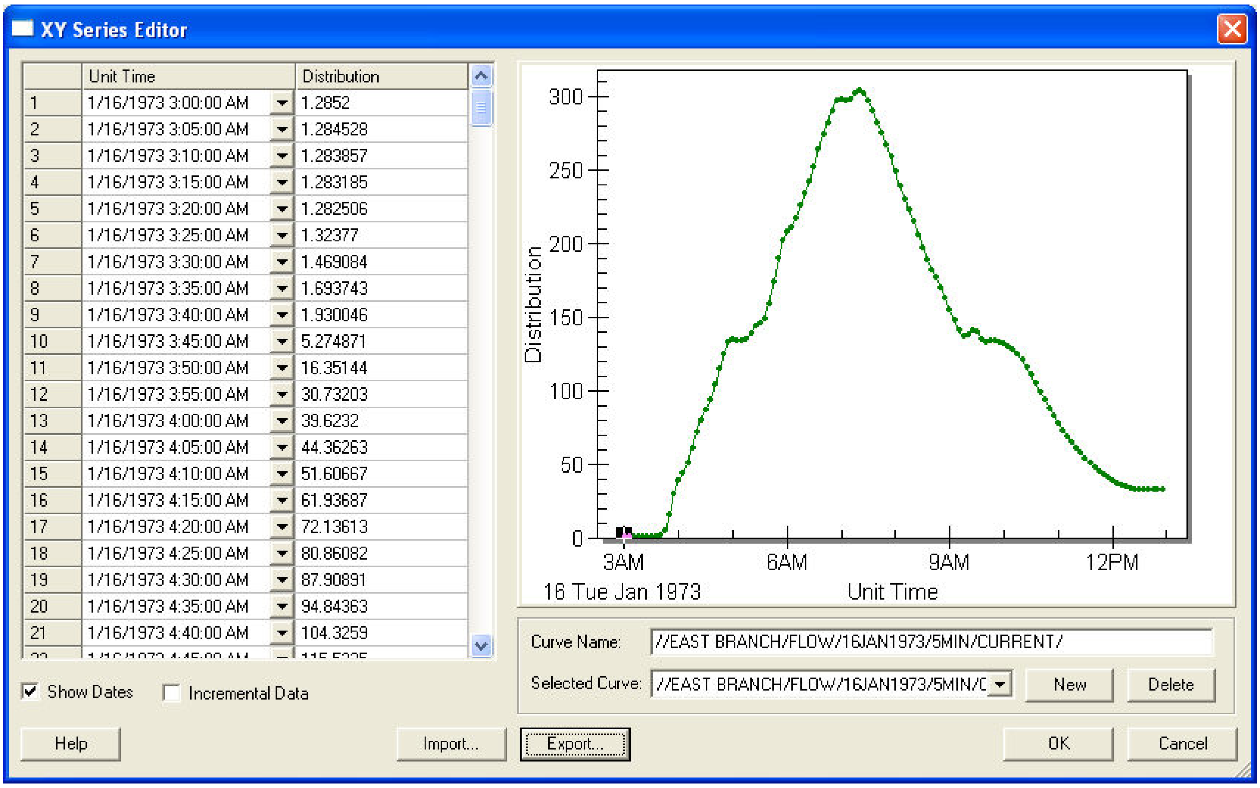
Help

Clean Up Model – This button runs only the selected tasks listed above the button.

Save – This button saves the project in your current state.

XY Series Editor

Use the *XY Series Editor* to edit, import, and export time series and XY series data. This data can include precipitation, discharge, and other parameters versus time or other data such as elevation-area and elevation-discharge curves.



The *Show Dates* option can be used to display the date and time at each point if the data is time series data. The incremental data option can be used if the data is incremental data instead of cumulative data, such as incremental precipitation data. The **Import** and **Export** buttons can be used to import or export the data in XY Series or DSS file formats.

Related Topics

- DSS Files
- DSS Interface
- XY Series Files

Land Use

In the **Compute GIS Attributes** dialog, the *Land Use* option determines whether a land use coverage or a land use grid will be used. The critical attribute for land use is an ID that can be related to a table of parameters for curve numbers, Green & Ampt parameters, HSPF segments, or any other hydrologic parameter that requires a land use ID for computation.

With a land use grid, WMS uses the ID associated with each cell in the land use grid to determine the land use value. When there is a land use coverage defined, WMS uses the ID associated with each polygon to determine the land use. The land use table attributes can be edited by right-clicking on the land use grid or by going to the attributes of a selected land use polygon in the land use coverage. When there is both a land use grid and a land use coverage, it can be selected whether there is a land use grid or a land use coverage in the drop down box.

Related Topics

- Compute GIS Attributes
- Soil Type
- Mapping Tables
- Land Use Coverage
- Obtaining Land Use Data from the Internet

Edit File

The **Edit File** command allows you to examine (or edit) any text file from within WMS. This command is particularly useful if errors occur while running a simulation using one of the hydrologic models supported by WMS. If a model does not run to a successful completion, errors can usually be found by examining the ASCII output file.

You will be prompted for the name of a text editor (Notepad is the default) and then the file is brought up in the specified editor. You can use the **Find Other** button to locate a different word processing *.exe file and the "Never ask this again" toggle can be set so that you are not prompted for the word processing program each time.

Related Topics

- Running an HEC-1 Analysis
- Running a TR20 Analysis
- Running HSPF

Annotations

Annotation Objects

The XMS application family provides a series of tools to annotate the data in an application for presentations, animations and screen shots.

These tools (annotation objects)are accessed through the Annotations Module and include:

- Images
- North Arrows
- Scale Bars
- Text
- Lines
- Ovals
- Rectangles

Screen vs World Space Layers

All annotation layers either contain objects referenced to world or screen coordinates. Objects referenced to world coordinates will change size and position on the screen with the underlying data. This is useful to identify specific locations in your model such as peir locations. Objects associated with screen coordinates do not move on the screen with the underlying data. This is useful for titles, legends such as north arrows and scale bars, and logos. Some types of annotations can only be created in screen space layers including North Arrows, Images, and scale bars.

If the first annotation object you create, could be part of a screen or world space layer SMS will ask which type of layer you wish to create and add the object to. You can create additional layers by right clicking on the *Annotation Data tree* item and selecting ***Create Screen Space Layer*** or ***Create World Space Layer***. Layers are differentiated by including an 'S' for screen space layers or 'W' for world space layers in their icons in the project explorer.

If multiple layers exist, any newly created annotation object will be placed in the "current" layer.

Annotation Object Attributes

The extents of annotation objects defined by a frame. The user defines this frame initially when creating the annotation object by left clicking at any point on the screen and dragging a rectangle with the mouse (left button still down). The display will show the frame while you drag with the mouse. (Points and lines defining degenerate frames are not allowed.) When the user creates a annotation, if the frame is too big for the window, it will be resized appropriately. Annotations can't be resized or moved even partially outside of the borders of the window. If the user resizes a annotation through a quick mouse drag and the cursor lands outside the window, the annotation will be redrawn to take up all the window space in that direction.

This frame bounds the region of the screen where the object will appear with the modeling data. The user interacts with the object by interacting with its frame and specifying its attributes or properties (see the section on selection below). The frame anchors the annotation object on the screen. This anchoring defines both the size and position of the object. The x-location, y-location, x-size and y-size are all defined independently as either a pixel value or percentage of the screen.

Horizontally, the user can position the left edge, the right edge or the center of the object. If the user positions the left edge, the object position is defined relative to the left edge of the screen. If the user positions the right edge, the object position is defined relative to the right edge of the screen. If the user positions the center of the object, the object position is defined relative to the horizontal center of the screen.

For example, the left side of the frame may be specified as 100 pixels from the left edge of the screen. Alternatively, the user may specify that the right edge of the frame should be 10% of screen width from the right edge. Finally, the user may specify that the center of the object is 100 pixels to the right of the center of the screen.

The vertical position and sizes of the object are similarly specified in the anchoring attribute of the object.

All annotation objects also have attributes. The specific attributes depend on the type of object. The attributes define color, line thickness, fill properties, associated images, etc.

Screen Space Images

A screen space image is simply a graphics icon mapped to the screen. A typical application would be to display a company, department, or municipality logo next to the numeric model being displayed in the graphics window.

Attributes of the screen space images include:

- General anchoring attributes
- The image file being displayed as part of the project
- Whether the image is being displayed as a scaled (distorted object), scaled based on its original aspect ratio, or locked at another aspect ratio.
- Transparency – The image properties dialog have a transparency checkbox. When checked it will cause the image to be redrawn with the most used color in the image. When that it is checked, it also causes the color checkbox and the tolerance edit field to become available. If the color checkbox is checked, it will activate the color button and the color button will have the latest chosen image color painted on it or the most used color in the image, if it has not been activated before. Clicking on the down arrow part of the color button causes a color popup to be displayed with swaths of the 40 most used colors in the image or all the colors in the image, if the image has less than 40 colors. Clicking on one of those colors will cause the image to be redrawn with that color made transparent in the image. The tolerance edit field allows for variation in the matching of the red, green and blue components. The tolerance field ranges in allowable values from 0.0 to 1.0. 0.0 means the red, green and blue components must exactly match. Values higher than 0.0 indicate the degree of variation from the given color. Clicking the transparency checkbox to the off state causes the image to be redrawn with no transparency.

Scale Bars

A scale bar occupies a fixed size of the screen to display the relative size of the objects in the simulation. The user defines the minimum width of the scale bar section (in pixels), along with a minimum and maximum height of the scale (also in pixels). The XMS application adds a "Units" label (meters in the image shown below) and labels for the model distance related to the scale divisions.

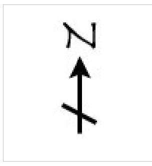
The XMS applications will compute a well conditioned number to use as the scale increment that fits in the specified scale bar extents.

Attributes of the scale bar include:

- General anchoring attributes.
- The minimum spacing between distance labels.
- The minimum division width (in pixels). The XMS application determines the number of divisions based on the minimum divisiion width and the width of the frame.

- The minimum and maximum height of the scale bar object.
- The font (style, color, size) used to label the scale bar.
- Whether or not the area behind the scale bar will be filled, and if so, with what color.

North Arrows



North arrow objects consist of automatically rotating screen space images. When an XMS application is installed, at least one default North Arrow image will be included in the application's home directory. Users may create or download as many north arrow icons as desired. These icons are displayed at the specified location (anchored with the standard options), but will rotate as the view direction changes so that the "up" direction of the icon always aligns with the "North" or positive "Y" direction.

Attributes of north arrows include:

- See screen space image attributes.

Text

Text can be created in world or screen space layers.

You can enter text by clicking in the graphics window with the create text tool active.

You can set the following attributes for text:

- Font – including size
- Color
- Background – fill behind with the background color or another color
- Border and border thickness

Lines/Arrows

You can create Lines/Arrows using the Create Line Tool. Lines/Arrows can be created in screen or world space layers.

The attributes available for Lines/Arrows include:

- Type – Dashed or solid w/ thickness
- Color
- Arrowheads – location (beginning, end, both) and size

Ovals

You can create ovals by dragging a box with Create Oval tool active. Ovals can be created in world or screen space layers.

The attributes that can be specified for ovals include:

- Line type – dashed/solid with width
- Line color
- Fill and color or No Fill

Rectangles

Rectangles can be created in screen or world space layers. Their attributes are the same as those available for ovals.

Selection

The first toolbar button is used to select and set attributes for annotation objects. This requires that objects exist to be selected. In this case when you press this tool and then left click in the annotation object, the object frame will be drawn around the annotation. In addition to the frame, the XMS application displays grab handles on the corners and edges of the frame. The user modifies the rectangular shape of the annotation by dragging one of the grab handles and changes the position of the object by dragging the annotation (click at any point in the object interior).

When this tool is active the right mouse button will pop up a menu of all dialogs for that particular type of annotation. This will include at least "Attributes" which will display a dialog of the objects attributes for editing.

Viewing Annotations At Specific Time Intervals

Available in SMS 11.1 and in development for GMS, annotations can be setup to be viewed at specific time intervals. To setup annotations so they only are displayed at specified time intervals do the following:

- Right click on the Annotation layer in the tree then select *Properties...*.
- This dialog will display the *Annotation Layer Properties* dialog.
- Check the *Apply time range* checkbox
- Modify the "begin" and "end" time controls to specify the range for when annotations are visible.
- Click *Ok*.

Now film loops and time steps will only display the annotation when its within the range specified.

Topologic Trees

A topological tree is one way to represent a watershed in the absence of terrain data. When a digital terrain model or feature objects are being used to automate basin delineation and build a particular sub basin configuration, a tree is simultaneously generated. The tree representation uses icons for confluences (outlet points) and basins, and should be clear to both experienced and inexperienced users of hydrologic models such as HEC-1 or TR-20.

The public domain version supports the building of a hydrologic model using a topologic tree as the only means of developing the model.

An equivalent tree representation is also shown and can be selected from the *Project Explorer*.

Building Trees

A topologic tree can be used to create a watershed configuration in the absence of digital terrain or feature object data (the only possibility for the public domain version of WMS). The tree representation can then be used to select basins and outlet points for entry of all basin and routing data, in the same way the digital terrain based watershed map can be used. Geometric parameters such as areas, lengths, and slopes, which can be automatically computed when a digital terrain model is used, must be entered interactively. However, all other features of the hydrologic modeling interfaces work in the same fashion.

All of the functions for building and/or editing trees (except diversions and reservoirs) cannot be used when a digital terrain model is used. In this case, all new basins and outlet points must be created from the digital terrain model.

Adding

Adding Basins

The **Add | Basin** command of the **Tree** menu defines a new drainage basin for the selected outlet point. Any number of sub basins can be associated with each outlet point. Typically, there are either one or two (a sub basin for each upstream branch), depending on whether or not the outlet point represents a branch in a stream network.

Adding Outlets

The **Add | Outlet** command of the **Tree** menu allows the user to enter a new outlet "upstream" from the currently selected outlet node. This represents a new confluence or sub-basin outlet which is upstream from the selected outlet. In this fashion, a network of outlet points can be built which actually represent the stream network of the watershed.

Adding Reservoirs

A reservoir can be created for an outlet by selecting the outlet and choosing the **Add | Reservoir** command from the **Tree** menu. By creating a reservoir at an outlet storage, routing for the outlet can be done followed by routing from the outlet of the reservoir to the next downstream outlet point.

Adding Diversions

Diversions are defined in a two step process: by defining the diversion at the outlet or basin where flow is to be diverted, and then assigning a previously defined diversion (retrieving a diversion) to another outlet. Diversions are defined by first selecting an outlet point or a drainage basin where the flow is diverted and then selecting the **Add | Diversion** command from the **Tree** menu.

After adding a diversion, a link will be drawn from the outlet or basin selected to the side of the Graphics Window, indicating that no retrieval outlet has yet been specified for that diversion.

The retrieval outlet point can be specified when defining diversions from one outlet to another by multi-selecting the retrieval outlet before issuing the Add Diversion command.

Adding Sources

The **Add | Source** command of the **Tree** menu allows the user to define a source at an "upstream" outlet. No basins or outlets may exist upstream from a source outlet as they are intended to represent the headwaters of a stream within a basin and the source is that flow entering the modeled watershed through the stream.

Adding Sinks

The **Add | Sinks** command of the **Tree** menu allows the user to define a sink at a "downstream" outlet. No outlets may exist downstream from a sink outlet as they are intended to represent the discharge of a stream within a basin and the sink is that flow leaving the modeled watershed through the stream.

Deleting

Deleting Basins

The **Delete | Basin** command of the **Tree** menu can be used to delete a selected drainage basin from a given outlet. The Select Drainage Basin tool should be active to select a basin icon.

Deleting Outlets

The **Delete | Outlet** command of the **Tree** menu allows the user to delete selected outlet points. The Select Vertices tool can be used to select outlet points which are to be deleted.

Deleting Reservoirs (Tree)

The **Delete | Reservoir** command of the **Tree** menu can be used to delete a selected reservoir from a given outlet. The Select Outlets tool should be active to select an outlet icon where a reservoir has been defined from the Tree menu.

Deleting Diversions

The **Delete | Diversion** command of the **Tree** menu can be used to delete a selected diversion. The Select Diversions tool should be active to select a diversion icon.

Deleting Sources

The **Delete | Sources** command of the **Tree** menu is used to delete a defined source from an outlet.

Deleting Sinks

The **Delete | Sinks** command of the **Tree** menu is used to delete a defined sink from an outlet.

Inserting Outlets

The **Insert Outlet** command of the **Tree** menu allows the user to enter a new outlet between two selected outlet points. This makes it possible to insert a confluence which may have been inadvertently left out or to simply subdivide a watershed into smaller sub basins. The Insert Outlet command can also be used to insert a new outlet at the base of a tree. In such cases then only the bottom most outlet should be selected before issuing the Insert Outlet command.

Merging Outlets

The **Merge Outlets** command of the **Tree** menu can be used if you wish to connect the downstream-most outlet of one watershed into an outlet of another watershed. This will effectively join the two watersheds into a single a watershed.

This can be used to combine information from two model (HEC-1) files into one. In order to do this you would open the first HEC-1 file into an editor, then you would insert the second file below (or above the first). You will want to eliminate the job control records and the zz record in the middle of the new file and save. When you open this new file into WMS it will contain two trees that can be merged together at the appropriate location.

Retrieving Diversions

To retrieve a diversion at an outlet point, select the outlet point where the diverted flow is to be retrieved and then select the **Retrieve Diversion** command from the **Tree** menu. If more than one diversion not yet retrieved exists, a list of defined diversions without retrieval outlet points (a diversion cannot be assigned to more than one retrieval outlet) will appear in the diversion selection dialog. Select the name of the desired diversion and choose OK.

After retrieving a diversion, the diversion link will now be drawn from the outflow outlet or basin to the retrieval outlet point. HEC-1 only allows inflow to be defined at outlet points.

Rebuilding the Display

When creating a topologic tree WMS tries to default the display according to the width and breadth of the tree, placing an icon in each quadrant. You may wish to move individual icons around in order to display the tree differently. However, if you wish at anytime to have WMS rebuild the default display you can choose the **Rebuild Tree Display** command of the *Tree* menu.

Related Topics

- Tool Palette

Hydrographs

Output for several of the hydrologic models includes a hydrograph. WMS allows hydrographs to be displayed and compared. Multiple hydrographs can be read in and viewed, listed, and exported. Plot options allow control over the way hydrographs are displayed.

A hydrograph can be computed from any of the supported hydrologic models. A hydrograph can also be defined/imported using the **New** command in the *Hydrograph* menu. This option allows you to copy a hydrograph from the clipboard, or open a text file using the text import wizard and define it for a selected basin or outlet node.

The hydrograph should be selected prior to bringing up the detention basin calculator.

Open Hydrograph Plot

The **Open Hydrograph Plot** command brings up a *Plot Window* for selected hydrograph(s). If you wish to view more than one hydrograph you should select multiple hydrograph icons from the graphics window before choosing this command. You may also bring up a new hydrograph window by double-clicking on the hydrograph icon.

The display settings of a hydrograph window, like all plot windows in WMS, are controlled by right clicking in the *Plot Window* and choosing from the available drop down menu commands.

Reading Hydrographs

Hydrographs can be read into WMS from a results files using the **Open** command in the *Hydrographs* menu. Hydrographs are generated for each basin and outlet point when running one of the hydrologic models. Two hydrographs may be generated for the outlet points, one representing the combination of "upstream" hydrographs, and one representing a hydrograph which has been routed to the next downstream outlet point.

If multiple storms or multiple ratios of a given storm are analyzed in the same run of HEC-1, then multiple hydrographs for each basin and outlet point will computed. If multiple storms have been defined using the JD card, the index numbers of hydrographs to be read must be specified. All indices or particular numbers can be specified. If multiple ratios of the same storm exist (defined on JR records), all ratios will be read. The display of a given ratio is controlled from within the *Hydrologic Modeling* tab in the *Display Options* dialog. Currently multi-plan storm hydrographs are not read.

Each storm index becomes a separate hydrograph set. Hydrograph sets are named by appending the index number to the file from which it is read. While it may be convenient to read all indices each time, it may produce an overwhelming number of hydrographs and make the display too cluttered for comparisons.

It is often useful to read a hydrograph not computed by HEC-1, or one of the other hydrologic models for calibrating purposes. This can be done by creating a hydrograph file using the WMS file format, but an easier method is to use the **New** command in the *Hydrograph* menu (or by right-clicking on a basin or outlet station from the *Project Explorer*) to either open a spreadsheet text file, or paste a copied hydrograph from a spreadsheet program using the Text Import Wizard. The basin or outlet where you wish to define the hydrograph (i.e. the point where you wish to compare a computed hydrograph with the measured hydrograph) should be selected prior to using this command.

Viewing Hydrographs

Hydrographs can be read into WMS and displayed in icon form at the appropriate basin or outlet.

For an HEC-1 analysis, a TAPE22 file can be read into WMS. Any number of hydrograph sets (TAPE22 files) may be read into WMS and displayed in a *Hydrograph Plot Window*. The name of the TAPE22 file is given as the solution file when running HEC-1 from the *HEC-1* menu.

For a TR-20 analysis the GRAPHICS file, which stores discharge hydrographs, can be read into WMS. This options is only available when running the version of TR-20 distributed with WMS. This file is specified as part of the "all other files" prefix when running TR-20 from within WMS, and always has the three letter extension of THY.

For the NSS and rational analysis programs hydrographs can be computed from dimensionless unit hydrographs applied to the computed peak discharges. These hydrographs are automatically stored and displayed.

A hydrograph can also be imported from a text file or copied from the clipboard for a basin or outlet and then used to calibrate computed hydrographs or used as input to the detention basin calculator.

Hydrographs are displayed in a *Hydrograph Plot Window* by using the **Select Hydrograph** tool and selecting the **Open Hydrograph Plot** command in the *Display* menu. Multiple hydrographs may be selected and overlaid in a plot window at the same time, and several different display options can be used while examining hydrographs by right-clicking in the plot window.

Displaying Hydrographs

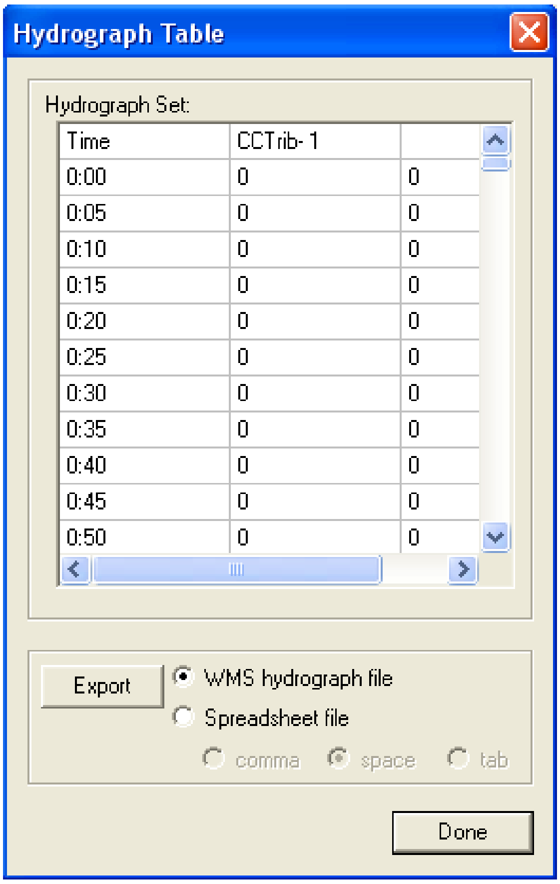
WMS creates a new Plot Window each time a hydrograph is selected for display. A hydrograph can be displayed in a plot window by double-clicking on the small hydrograph icon that appears after running a hydrologic simulation, reading a hydrograph file, or in some other way calculating the hydrograph. It can also be created by selecting the hydrograph icon and choosing **New Plot Window** from the *Display* menu. If you wish to have more than one hydrograph displayed in a plot window then select all hydrographs prior to choosing the **New Plot Window** command by holding down the *SHIFT* key (you can also double-click on the last of the hydrographs you select).

The display options of the plot window are handled from the standard *plot window* menu accessed by right clicking within the plot window.

A series of standard comparison plots can be generated using the **Plot Wizard** command in the *Display* menu.

Listing Hydrographs

Hydrographs values can be listed in a tabular format using the **List** command in the *Hydrograph* menu. All currently selected hydrographs will be listed in a table in this dialog. You can output the hydrograph to either a WMS formatted hydrograph file, or a standard spreadsheet formatted file (tab, comma, or space delimited) from within this dialog.



Hydrograph in a tabular format

Deleting Hydrographs

All hydrographs read into WMS can be deleted simultaneously using the **Delete All** command in the *Hydrographs* menu. The **Delete Previous** command can be used to remove the most recent set. Deleting the previous hydrograph set when it is no longer needed for display and comparison eliminates the confusion created by displaying too many hydrographs in the *Hydrograph Window*.

New Hydrographs

Sometimes it is useful to create a new hydrograph, not computed by one of the WMS models, for calibration or further analysis. This can be done using the **New** command in the *Hydrograph* menu after selecting the basin/outlet where you wish to associate the new hydrograph. When selecting this command you can choose to import an existing text file or paste from the Windows clipboard a hydrograph that has previously been cut/copied from a spreadsheet or text file. In either case the data will be processed using the *Text Import Wizard* and a new hydrograph created for the selected basin/outlet.

The text file or copied data should include a column for time and a column for flow. A couple of limitations associated with creating a new hydrograph in this fashion are that the times must be in minutes and the time step must be constant. After copying from the clipboard, or opening the text file the data will be placed in the *Text Import Wizard* and you will be able to identify header rows, set the starting row, and map the columns to their respective data fields.

Exporting Hydrographs

Selected hydrograph(s) may be exported to either the native WMS file format or to a space, comma, or tab delimited text file that can be read into a spreadsheet program like Microsoft Excel.

Related Topics

- Detention Basin Calculator

Hydrograph Files

When running HEC-1, WMS reads the TAPE22 hydrograph results file. However, it is convenient to read measured hydrographs for comparison with computed hydrographs, therefore a simplified hydrograph file format can be used. The format is shown in Figure 1, with a sample file given in Figure 2.

HYDROGRAPH
name starttime interval numordinates
flow-1 flow-2 flow-3 ... flow-10
...
...
...
flow-n

Figure 1. Hydrograph File Format.

HYDROGRAPH
B1 0 30 25
0.0 0.4 0.9 1.5 2.2 3.0 3.9 4.9 6.0 7.2
9.5 12.8 16.0 15.2 14.1 10.2 9.0 7.5 5.8 4.0
3.3 2.2 1.0 0.5 0.1

Figure 2. Sample Hydrograph File.

Card Type	**HYDROGRAPH**
Description	Defines a set of hydrographs.
Required	YES
Format	HYDROGRAPH name starttime interval numordinates flow-1 flow-2 flow-3 ... flow-10 ... ... flow-n

Sample	HYDROGRAPH B1 730 45 12 8.0 14.0 25.0 40.0 65.0 63.0 55.0 50.0 42.0 30.0 15.0 0.0 B2 730 45 12 10.0 17.0 29.0 45.0 71.0 64.0 53.0 41.0 28.0 15.0 4.0 0.0		
Field	*Variable*	*Value*	*Description*
1	name	str	Station (basin or outlet) name (KK in HEC-1).
2	starttime	+	Military time (0-2400) at the start of the event.
3	interval	+	Interval in minutes between hydrograph ordinates.
4	numordinates	+	The number of hydrograph ordinates.
5-n	flow	+	Hydrograph ordinates.Each of fields 1-5+n are repeated for each station in the file.

The hydrograph file is considered to be free-field when read by WMS so any number of spaces can be used to separated the different parameters. Figure 2 shows a sample file where there are 25 ordinates with a time interval of 30 minutes.

3.6. Images

Images

Images are one of the four basic object types that is supported in the Map module. An image is typically a scanned map or aerial photo in TIFF or JPEG format. Images can be imported to WMS and displayed in the background to aid in the placement of objects as they are being constructed or simply to enhance a plot. Images can also be draped or "texture mapped" onto a TIN or 2D grid.

WMS supports the following image types:

- TIFF
- JPEG
- MrSID
- ECW

Importing an Image

The first step in using a new digital image for either background display or for texture mapping is to import the image. This can be done by either selecting the **Import** command in the *Images* menu or using the **Open** command in the *File* menu with the correct extension and opening an image file (TIFF, JPEG, or MRSID). Image files will be either georeferenced (contain coordinate information embedded) or not.

Multiple Images

WMS now allows more than one image at a time, however you will want to insure that each image is in the same coordinate system and that they are adjacent or it will do little good to have more than one image.

Geo-Referenced TIFF Files

Without embedded coordinate information about the image, it must be registered to real world coordinates when first imported. Some image files have the coordinate information embedded as part of the image and are referred to as georeferenced images. When a georeferenced image is imported to WMS, the image is automatically registered.

Additionally, some images contain companion world files (separate text files with the coordinate information) that can be used to georeference the image. WMS automatically recognizes some world files and when they are the images are automatically georeferenced using the included information. If the world file is not recognized automatically it can still be imported from the registration dialog.

Image Display Options

Once an image is imported, the **Display Options** command in the *Display* menu can be used to control how the image is displayed. The **Image Display Options** button of the *Map* tab brings up the *Image Display Options* dialog. The following display options are available:

Draw on XY plane behind all objects

If this option is selected, the image is drawn in the background prior to drawing any other objects. This mode is used to aid in the creation of new objects or to simply enhance a plot. The image is only displayed in plan view.

Texture map to surface when shaded

If this option is selected, the image is "draped" or texture mapped over the designated surface (TIN or 2D Grid). The image must be registered such that the surface lies within the domain of the image. The surface is texture mapped when the image is shaded using the **Shade** command.

Screen Capture

Displays created by WMS can be captured to a TIFF image file using the **Screen Capture** command. These images can then be used in WMS as backgrounds, or can be used as images in other applications or report documents.

When the **Screen Capture** command is issued the image currently in the *Graphics window* is converted internally to a TIFF image. Since the real coordinates of the screen corners are already known the image is automatically registered as it is captured. These images and registration points can be exported and read back into WMS at a later time.

Image Crop Collar

Many of the images available for use in WMS are the standard USGS map series. These maps have been scanned as is and contain the information on the collar (border) of the maps. This information is okay and in fact you may want to see it, but often it is convenient to remove the collar (especially when you are tiling multiple images together).

The **Crop Collar** command, available by right-clicking on an image in the *Project Explorer*, can be used to automatically remove the collar from the image for display.

Exporting Images

The **Export** command, available when right-clicking on the image in the *Project Explorer* is used to save a registered image. This command is most useful after screen capturing or cropping an image, or multiple images so that you can save the new area as an image file.

Related Topics

- Map Module
- Registering an Image

Image Preferences

The *Image Preferences* tab of the *Preferences* dialog is used to set procedures for using images. This tab has two primary sections:

- Image Pyramids – WMS 8.0 employs a new method for opening images that has to do with the level of detail that is available for different levels of viewing. In particular, when pyramids are built, multiple files of varying resolutions are saved by WMS so that when an area is displayed (by zooming), an appropriate resolution image is available to best represent the area. While this provides clearer displays, the process takes time. When image pyramids are not built, only one resolution image is created. Thus, while viewing an area near the original resolution, the display looks good; yet when zoomed out to a more overall extent view, the image may appear choppy or grainy.
- TIFF→JPEG Conversion – In order to show multiple image types in the graphics window simultaneously, image files are automatically converted to JPEG format, by default. This process can take significant time; therefore, the option exists to not convert files. Furthermore, users can specify where the converted JPEG is saved.

Related Topics

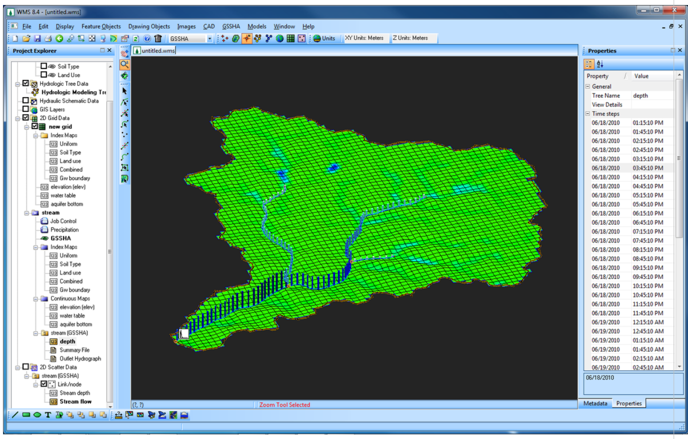
- Overview

Image Gallery

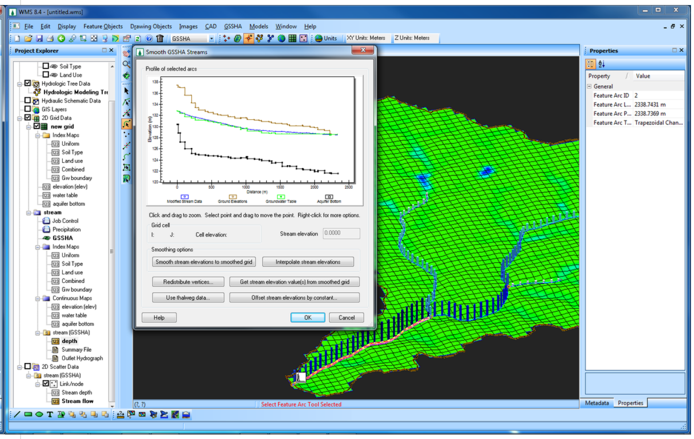
Various images generated by WMS. Feel free to post your cool WMS-generated images here but only if you're OK with them being free and in the public domain. Please follow these guidelines when posting images.

Black background

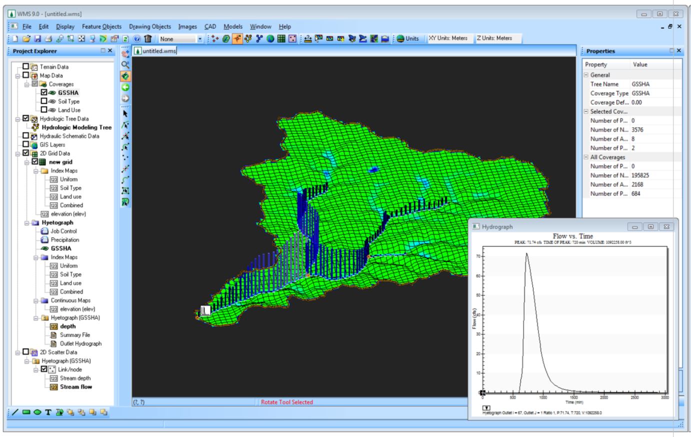
All these images have the background color set to black.



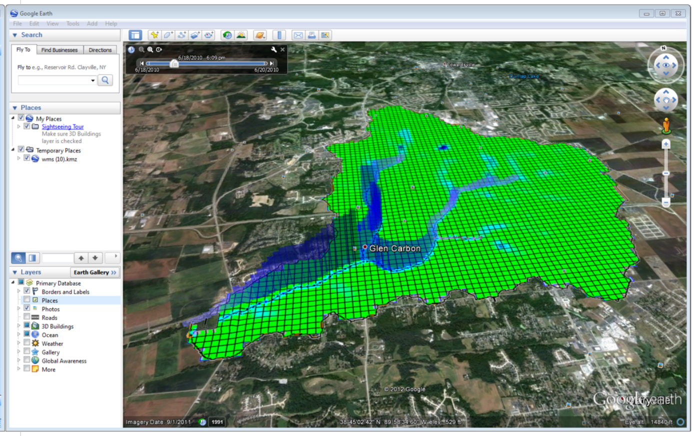
GSSHA Groundwater Model with stream depths.



GSSHA Groundwater Model with stream smoothing dialog.



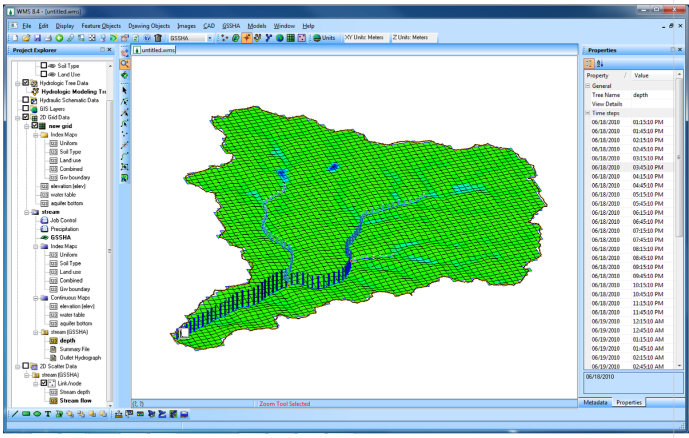
GSSHA Runoff simulation, with hydrograph.



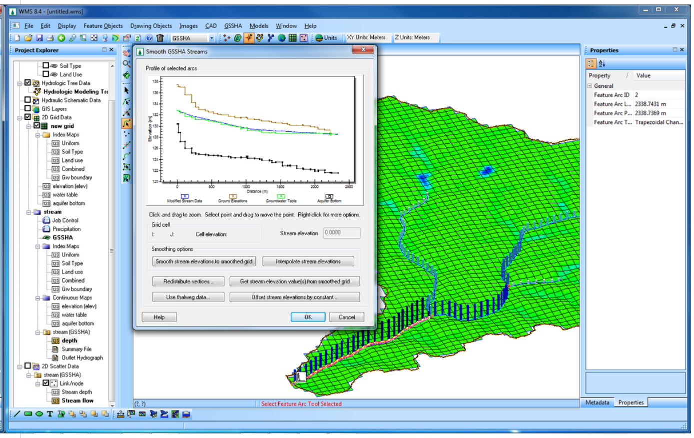
WMS 9.0, exporting simulation into Google Earth.

White background

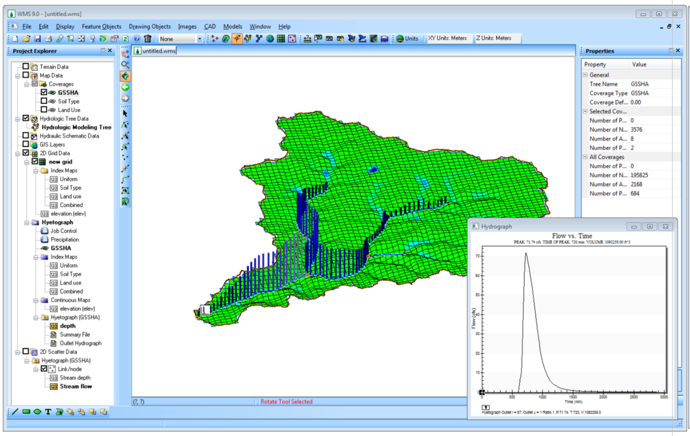
Same as above but with the background color set to white.



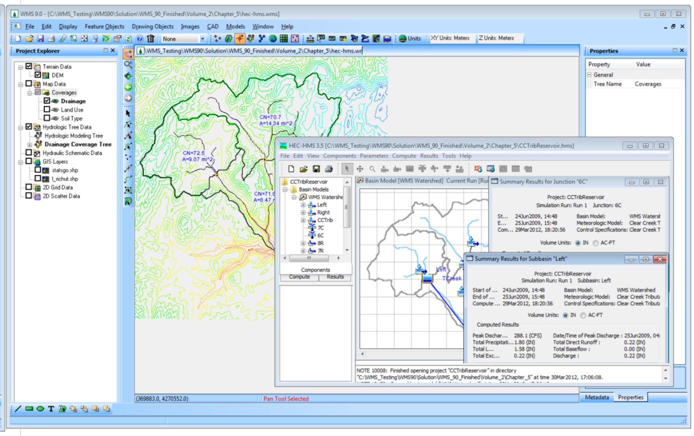
GSSHA Groundwater Model with stream depths.



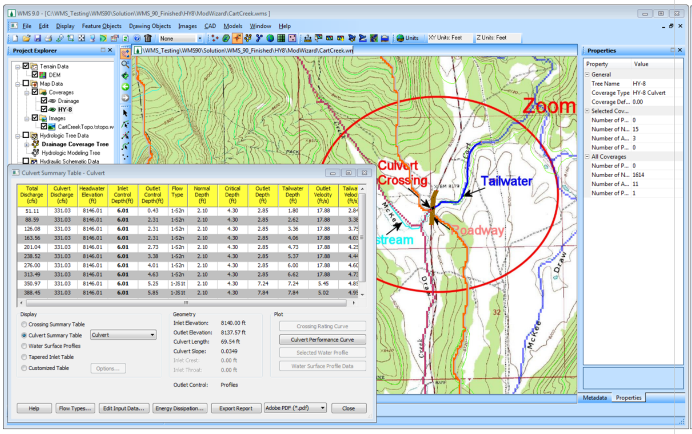
GSSHA Groundwater Model with stream smoothing dialog.



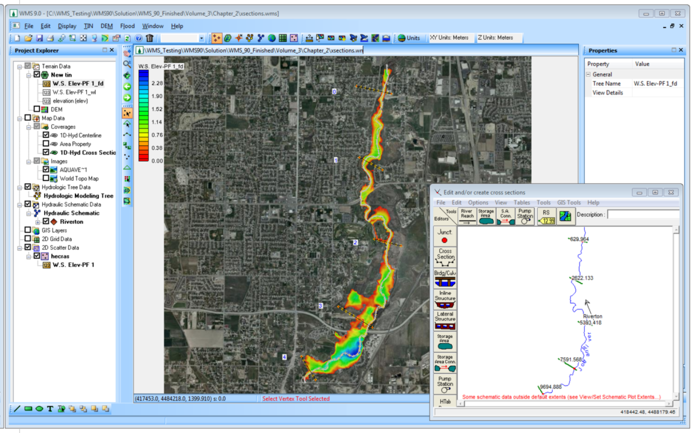
GSSHA Model with with hydrograph.



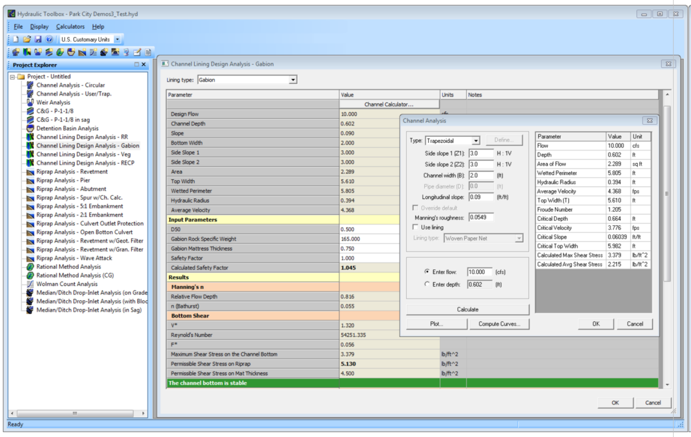
WMS 9.0 and HEC-HMS.



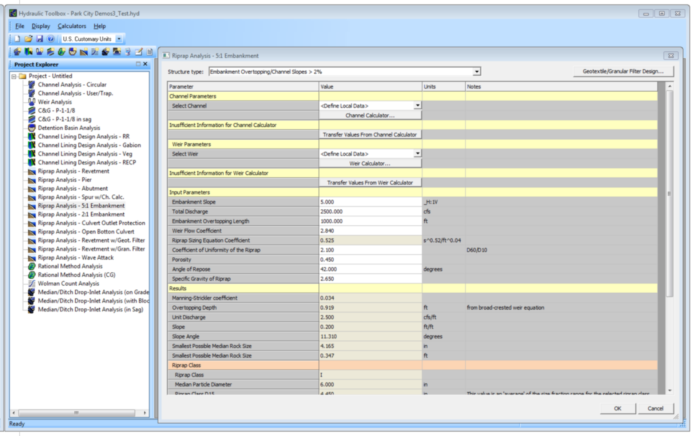
WMS 9.0 and HY-8.



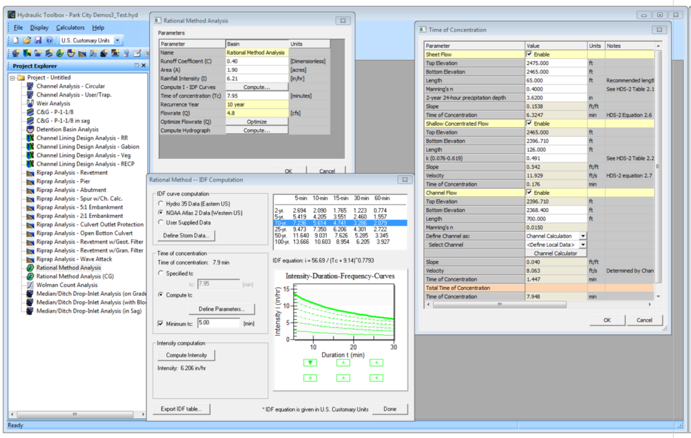
WMS 9.0 and HEC-RAS.



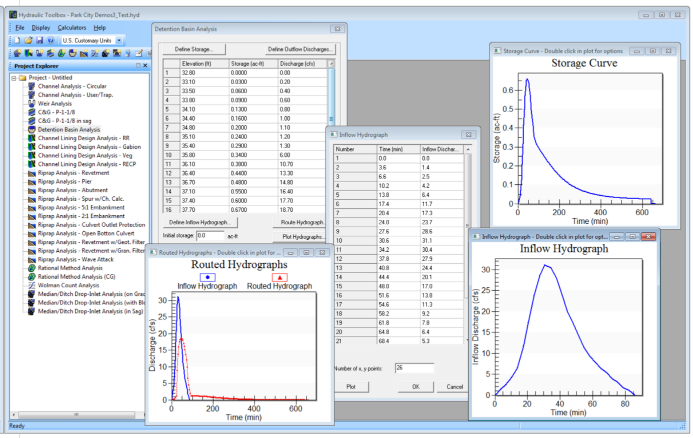
Hydraulic Toolbox, Channel Lining Design Analysis - Gabion.



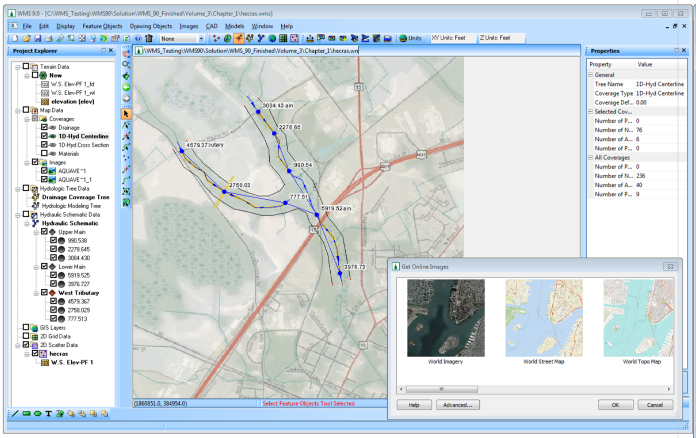
Hydraulic Toolbox, Riprap Analysis - 5:1 Embankment.



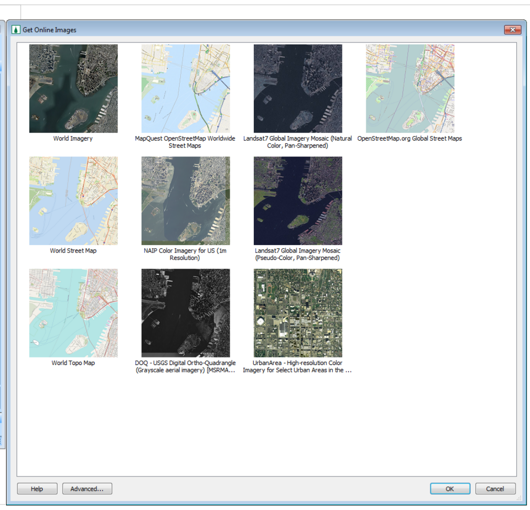
Hydraulic Toolbox, Rational Method Analysis.



Hydraulic Toolbox, Detention Basin Analysis.



WMS 9.0 using online images.



WMS 9.0 new Get Online Images window.

Registering an Image

If an image file is not geo-referenced then the user must define the coordinate system of the image. The register dialog allows the user to specify the coordinate system for the image. When an image is opened, if the image is not self-referenced, XMS attempts to find world file with the same name as the image (*.wld or *.jpgw extension). If neither of these is found, the register dialog opens.

What is Image Registration?

Before an image can be displayed, the image must be "registered" or geo-referenced. Registering an image involves identifying points on the image corresponding to locations with known real world (XY) coordinates. Once these points are identified, they are used to scale and translate the image to the proper location when it is drawn with the other objects in the Graphics Window. If an image is not registered properly, any objects which are created using the background image as a guide will have the wrong coordinates.

Register Image Dialog

An image is registered using the Register Image dialog. The main feature of the Register Image dialog is a large window in which the image is displayed. Two or three points (shown by "+" symbols) are also displayed in the window. These points are used to identify locations with known real world coordinates. The real world coordinates (X,Y) and image coordinates (U,V) of the registration points are listed in edit fields below the image. The points are moved to the desired locations on the image by dragging the points using the tools described below. Once the points are located, the real world coordinates can be entered in the corresponding edit fields. The dialog contains the following options:

- **2 point or 3 point registration –** Two point registration rotates and uniformly scales an image. Three point registration allows for non-uniform scaling to account for some parallax.
- **Import World File –** Used to import a TIFF world file (*.tfw). A TIFF world file has the information needed to set the (X,Y) and (U,V) coordinates in order to place the image in the correct world coordinates.
- **Image name –** Used to associate a name with the file. This name will appear in the project explorer.

Register Image Dialog Tools

The following tools can be used to help position the registration points:

Tool	Tool Name	Description
	Select Point Tool	The Select Point tool is used to select and drag register points to a location on the map for which real coordinates are known so that they can be entered in the corresponding XY edit fields.
	Zoom Tool	In some cases, it is useful to magnify a portion of the image so that a registration point can be placed with more accuracy. The Zoom Tool is used to zoom in a portion of the image.
	Pan Tool	After zooming in on a portion of the image, the Pan Tool is used to pan the image vertically or horizontally.
	Frame Macro	The Frame Macro is used to automatically center the entire image within the drawing window of the dialog after panning and zooming in on a specific location.

Import World File

The **Import World File** button can be used to automatically define the registration data. A world file is a special file associated with a previously registered image that is exported from ArcView® [1] or Arc/Info® [2]. The file contains registration data that can be used to register the image.

Saving/Reading Image Registration Data

When a project file is saved, a link to the image is saved in the project file, along with the current image registration information so that the image is re-registered to the same coordinates every time the project is opened. The original image file and world file (if one exists) are not altered.

Convert Point Coordinate System

The x, y coordinates of each register point must be specified. If the user has the (x,y) coordinates in a different coordinate system than their project, the coordinates will need to be converted.

GMS Point Conversion

The Convert Point button in the image registration dialog will allow the user to convert the coordinates.

SMS Point Conversion

The Single Point Conversion command in the *Edit* menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

An alternative approach is to convert the coordinate system after importing by right clicking on the image in the Project Explorer and choosing ***Coordinate Conversion*** from the right click menu.

WMS Point Conversion

The Single Point Conversion command in the **Edit** menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

References

[1] http://www.esri.com/software/arcview/
[2] http://www.esri.com/software/arcgis/arcinfo/

3.7. File Support

Text Import Wizard

WMS can read many files generated by other software in their native format. Refer to Non-native WMS Files for a list. For files that are not included in the list, WMS provides the *Text Import Wizard*.

The *Text Import Wizard* enables users to import many different types of data into WMS. The *Text Import Wizard* is initialized by selecting a *.txt file in the **Open** command from the *File* menu, it can also be initiated by pasting data from a text file or spreadsheet into WMS using the Paste command from the *Edit* menu.

The wizard has two steps.

Step 1 – Delimiting Columns

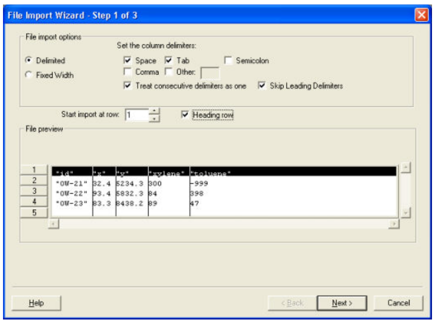
Step 2 – Assigning Column Types

Step 1 – File Outline

The first step in the wizard allows users to delimit the data into columns. Two options exist to delimit the data.

- Delimited
- Fixed Width

For the *Delimited* option, typical delimiters are included as well as an option for users to specify a delimiter. Columns can also be specified with a fixed width by clicking on the ruler bar or the window with the data. Break lines can be dragged, and they can be deleted by double-clicking on the break line or dragging them off the screen. The user can specify the starting row the data will be imported. If your data has a row of headings, you can indicate such and WMS will use the headings in the next step to determine what kind of data each column represents.

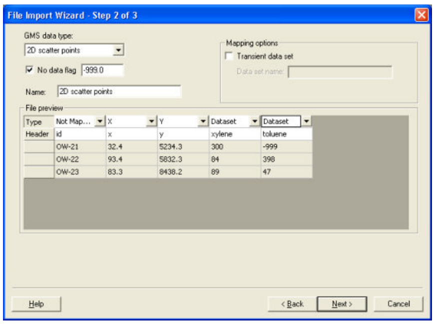


Step 2 (File Format) of the Text Import Wizard.

Step 2 – Assigning Column Types

The first 20 lines of the file are displayed in a spreadsheet according to the file outline specified in step 1. This step lets you pick what kind of data you are importing (see the supported formats below). A "no data flag" can be specified for the file. This is a number that, when encountered in the file, tells WMS to mark the value as "NULL" or "no data". If the file type is a scatter point, the file can be marked as transient. If the file is transient, only one dataset can be created and all dataset columns become time steps of the dataset.

The data in the columns are identified by selecting the type in the combo box at the top of each column in the spreadsheet. If a row of headings exists, WMS will automatically select the proper type if it recognizes the heading. Otherwise they are labeled "Not Mapped" by default. The available column types changes depending on the WMS data type selected. Certain column types must be mapped for each file format before the user can progress to the next step in the wizard. The name of each column is changed by editing the Header cell.



Step 2 (File Format) of the Text Import Wizard.

Supported Formats

The following types of data can be imported into WMS via the Text Import Wizard:

- 2D Scatter Sets

Fields

A description of the fields (columns) that WMS recognizes when importing text files is provided in the tables below.

2D Scatter Points			
Field	Type	Required	Comments
X	Number	yes	
Y	Number	yes	
Label	Text	no	
Material ID	Number	no	
Data set	Number	no	One transient, or multiple steady state datasets.
Example			
"id" "x" "y" "rain" "humidity" "OW-21" 32.4 5234.3 300 -999 "OW-22" 93.4 5832.3 84 398 "OW-23" 83.3 8438.2 89 47			

Related Links

- Open
- WMS Supported File Formats

ARC/INFO ASCII Grid Files

WMS can import ARC/INFO® ASCII grid for use as a background DEM. Since it is a simple file format, other digital elevation data can be formatted in the same way and then imported into WMS using the **Import Grid** command in the *DEMs* menu.

The CASC2D model may also import and used ARC/INFO® grid files when defining map parameters. The file format is shown in Figure 1 and an example file in Figure 2.

ncols ncol	/* Number of columns in the grid */
nrows nrow	/* Number of rows in the grid */
xllcorner x	/* Lower left x coordinate of grid */
yllcorner y	/* Lower left x coordinate of grid */
cellsize size	/* Grid cell size */
NODATRA_value NODATA	/* value of an empty grid cell */
z_{11} z_{12} z_{13} ... z_{1ncols}	/* values of row 1 */
z_{21} z_{22} z_{23} ... z_{2ncols}	/* values of row 2 */
.	
.	
.	
z_{nrows1} z_{nrows2} z_{nrows3} ... $z_{nrowsncols}$	/* values of last row*/

Figure 1. ARC/INFO® ASCII Grid File Format.

ncols 128
nrows 136
xllcorner 422415
yllcorner 4515405
cellsize 30
NODATA_value -9999
1287 1286 1286 1288 ...
1288 1288 -9999 1289 ...
.
.
1282 -9999 1283 1284 ...

Figure 2. Sample ARC/INFO® ASCII Grid File.

The card types used in the ARC/INFO® grid file format are self explanatory.

ASCII Dataset Files

Datasets can be stored to either ASCII or binary files. Multiple datasets can be stored in a single file and both scalar and vector datasets can be saved to the same file. The file format is identical for 2D and 3D datasets. The ASCII dataset format is shown in Figure 1. A sample data set file is shown in Figure 2.

For scalar dataset files, one value is listed per vertex, cell, node, or scatter point. For vector dataset files, one set of xyz vector components is listed per vertex, cell, node, or scatter point. If necessary, a set of status flags can be included in the file. If the status flag is false (0), the corresponding item (node, cell, etc.) is inactive. If status flags are not included in the file, it is assumed that all items are active.

DATASET	/* File type identifier */
OBJTYPE type	/* Type of object dataset is associated with */
BEGSCL	/* Beginning of scalar dataset */
OBJID id	/* Object id */
ND numdata	/* Number of data values */
NC numcells	/* Number of cells or elements */
NAME "name"	/* Dataset name */
TS istat time	/* Time step of the following data. */
$stat_1$	/* Status flags */
$stat_2$	
.	
.	
$stat_{numcells}$	
val_1	/* Scalar data values */
val_2	
.	
.	
$val_{numdata}$	/* Repeat TS card for each time step */
ENDDS	/* End of data set */
BEGVEC	/* Beginning of vector dataset */
VECTYPE type	/* Vector at node/gridnode or element/cell */
OBJID id	/* Object id */
ND numdata	/* Number of data values */
NC numcells	/* Number of cells or elements */
NAME "name"	/* Data set name */
TS istat time	/* Time step of the following data. */
$stat_1$	/* Status flags */
$stat_2$	
.	
.	
$stat_{numcells}$	

$v_{x1}\ v_{y1}\ v_{z1}$	
$v_{x2}\ v_{y2}\ v_{z2}$	
.	
.	
$v_{numdata}\ v_{numdata}\ v_{numdata}$	
/* Repeat TS card for each time step */	
ENDDS	/* End of data set */
/* Repeat BEGSCL and BEGVEC sequences for each dataset */	

Figure 1. ASCII Dataset File Format.

DATASET
OBJTYPE grid2d
BEGSCL
OBJID 27211
ND 8
NC 8
NAME "trichloroethylene"
TS 1 1.00000000e+00
0
0
0
1
1
1
1
0
0.00000000e+00
0.00000000e+00
0.00000000e+00
3.24000000e+00
4.39000000e+00
2.96000000e+00
7.48000000e+00
0.00000000e+00
ENDDS
BEGVEC
VECTYPE 0
OBJID 27211
ND 8
NC 8

NAME "velocity"
TS 1 5.00000000e+00
0
0
0
1
1
1
1
0
1.60000000e+01 1.60000000e+01 3.20000000e+01
6.40000000e+01 6.40000000e+01 1.28000000e+02
1.44000000e+02 1.44000000e+02 2.88000000e+02
1.96000000e+02 1.96000000e+02 3.92000000e+02
2.25000000e+02 2.25000000e+02 4.50000000e+02
9.21600000e+03 9.21600000e+03 1.84320000e+04
9.60400000e+03 9.60400000e+03 1.92080000e+04
9.80100000e+03 9.80100000e+03 1.96020000e+04
ENDDS

Figure 2. Sample ASCII Data Set File.

The card types used in the scalar dataset file format are as follows:

Card Type	**DATASET**
Description	File type identifier. Must be on first line of file. No fields.
Required	

Card Type	**OBJTYPE**		
Decription	Identifies the type of objects that the datasets in the file are associated with.		
Required	YES. If card does not exist, the file can only be read through the Data Browser. The datasets would then be assigned to the objects corresponding to the active module.		
Format	OBJTYPE type		
Sample	OBJTYPE tin		
Field	*Variable*	*Value*	*Description*
1	type	tin mesh2d grid2d scat2d mesh3d grid3d scat3d	TINs 2D meshes 2D grids 2D scatter points 3D meshes 3D grids 3D scatter points

Card Type	**BEGSCL**
Description	Scalar data set file identifier. Marks beginning of scalar dataset. No fields.
Required	YES

Card Type	**BEGVEC**
Description	Vector dataset file identifier. Marks beginning of vector dataset. No fields.
Required	YES

Card Type	**VECTYPE**			
Card ID	150			
Decription	Identifies the type of vector data that will be read and where to apply it.			
Required	This card is only required if the vector data is associated with elements/cells. If this card is not present, it is assumed that the data are associated with nodes/gridnodes.			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	0 1	The vectors will be applied to the nodes/gridnodes. The vectors will be applied to the elements/cells.

Card Type	**ND**		
Decription	The number of data values that will be listed per time step. This number should correspond to the total number of vertices, nodes, cells centers (cell-centered grid), cell corners (mesh-centered grid), maximum node id (meshes) or scatter points.		
Required	YES.		
Format	ND numdata		
Sample	ND 10098		
Field	*Variable*	*Value*	*Description*
1	numdata	+	The number of elements or cells.

Card Type	**NAME**		
Decription	The name of the dataset.		
Required	YES.		
Format	NAME "name"		
Sample	NAME "Total head"		
Field	*Variable*	*Value*	*Description*
1	"name"	str	The name of the dataset in double quotes.

Card Type	**TS**		
Decription	Marks the beginning of a new time step, indicates if stat flags are given, and defines the time step value, status flags, and scalar data values for each item.		
Required	YES.		
Format	TS istat time stat1 stat2 . . stat numcells val1 val2 . . valnumdata		
Sample	TS 1 12.5 0 1 1 1 34.5 74.3 58.4 72.9		
Field	*Variable*	*Value*	*Description*
1	istat	0	Use status flags from previous time step. For first time step, this indicates that all cells are active. 1 Status flags will be listed.
2	time	+	The time step value. If only one time step exists, time is not required
2 - (n+1)	stat	0,1	The status of each item. If active, stat=1. If inactive stat=0. Omitted if i=0 on STAT card.
(n+2) - (2n +1)	val	"+/-"	The scalar data values of each item.

Binary Dataset Files

Datasets can be stored to either ASCII or binary files. Compared to ASCII files, binary files require less memory and can be imported to WMS more quickly. The disadvantages of binary files are that they are not as portable and they cannot be viewed with a text editor.

The binary dataset file format is shown in Figure 1. The binary format is patterned after the ASCII format in that the data are grouped into "cards". However, the cards are identified by a number rather than a card title.

Card	Item	Size	Description
	version	4 byte integer	The WMS binary dataset file format version. value = 3000.
100	objecttype	4 byte integer	Identifies the type of objects that the data sets in the file are associated with. Options are as follows: 1 TINs 2 Boreholes 3 2D meshes 4 2D grids 5 2D scatter points = 6 3D meshes 7 3D grids 8 3D scatter points
110	SFLT	4 byte integer	The number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).
120	SFLG	4 byte integer	The number of bytes that will be used in the remainder of the file for status flags.
130 or 140	BEGSCL or BEGVEC		Marks the beginning of a set of cards defining a scalar or vector dataset.
150	VECTYPE	4 byte integer	(0 or 1) In the case of vector data set files, indicates whether the vectors will be applied at the nodes/gridnodes or the elements/cells.
160	OBJID	4 byte integer	The id of the associated object. Value is ignored for grids and meshes.
170	NUMDATA	4 byte integer	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered grid) or scatter points.
180	NUMCELLS	4 byte integer	This number should correspond to the number of elements (meshes) or the number of cells (mesh-centered grids). Value is ignored for other object types.
190	NAME	40 bytes	The name of the dataset. Use one character per byte. Mark the end of the string with the '\0' character.
200	TS		Marks the beginning of a time step.
	ISTAT	SFLG integer	(0 or 1) Indicates whether or not status flags will be included in the file.
	TIME	SFLT real	Time corresponding to the time step.
	statflag1	SFLG integer	Status flag (0 or 1) for node 1

	statflag2	SFLG integer	Status flag (0 or 1) for node 2
	....		
	val1	SFLT real	Scalar value for item 1
	val2	SFLT real	Scalar value for item 2
	....		
			Repeat card 200 for each timestep in the dataset.
210	ENDDS		Signal the end of a set of cards defining a dataset.

Figure 1. The Binary Scalar or Vector Dataset File Format.

The cards in the binary dataset file are as follows:

Card Type	**VERSION**
Card ID	3000
Description	File type identifier. No fields.
Required	YES

Card Type	**OBJTYPE**			
Card ID	100			
Description	Identifies the type of objects that the data sets in the file are associated with.			
Required	YES. If card does not exist, the file can only be read through the Data Browser. The data sets would then be assigned to the objects corresponding to the active module.			
Field	*Variable*	*Size*	*Value*	*Description*
1	id	4 byte int	1 2 3 4 5 6 7 8	TINs Boreholes 2D meshes 2D grids 2D scatter points 3D meshes 3D grids 3D scatter points

Card Type	**SFLT**			
Card ID	110			
Description	Identifies the number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	sizefloat	4 byte int	4, 8, or 16	Number of bytes

Card Type	**SFLG**			
Card ID	120			
Description	Identifies the number of bytes that will be used in the remainder of the file for status flags (1, 2, or 4).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	sizeflag	4 byte int	1, 2, or 4	Number of bytes

Card Type	**BEGSCL**
Card ID	130
Description	Marks the beginning of a set of cards defining a scalar dataset.
Required	YES

Card Type	**BEGVEC**
Card ID	140
Description	Marks the beginning of a set of cards defining a vector dataset.
Required	YES

Card Type	**VECTYPE**			
Card ID	150			
Description	Identifies the type of vector data that will be read and where to apply it.			
Required	This card is only required if the vector data is associated with elements/cells. If this card is not present, it is assumed that the data are associated with nodes/gridnodes.			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	0 1	The vectors will be applied to the nodes/gridnodes. The vectors will be applied to the elements/cells.

Card Type	**OBJID**			
Card ID	160			
Description	The id of the associated object.			
Required	This card is required in the case of TINs, 2D scatter points, and 3D scatter points. With each of these objects, multiple objects may be defined at once. Hence the id is necessary to relate the dataset to the proper object.			
Field	*Variable*	*Size*	*Value*	*Description*
1	id	4 byte int	+	The id of the object.

Card Type	**NUMDATA**			
Card ID	170			
Description	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered grid), maximum node id (meshes) or scatter points.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	numdata	4 byte int	+	The number of items. At each time step, numdata are listed.

Card Type	**NUMCELLS**			
Card ID	180			
Description	This number should correspond to the element id (meshes) or the number of cells (grids).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	numcells	4 byte int	+	The number of elements or cells.

Card Type	**NAME**			
Card ID	190			
Description	The name of the data set.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	name	40 bytes	str	The name of the dataset. Use one character per byte. Mark the end of the string with the '\0' character.

Card Type	**TS**			
Card ID	200			
Description	Defines the set of scalar values associated with a time step. Should be repeated for each time step.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	istat	SFLG int	0 1	Use status flags from previous time step. For the first time step, this value indicates that all cells are active. Status flags will be listed.
2	time	SFLT int	+	The time step value. This number is ignored if there is only one time step.

	stat	SFLG int	0 1	Inactive Active One status flag should be listed for each cell or element. These flags are included only when istat = 1.
	val	SFLT real	±	The scalar values

Card Type	**ENDDS**
Card ID	210
Description	Signals the end of a set of cards defining a dataset
Required	YES

DSS Files

WMS 8.0 and later provide support for importing, exporting, viewing, and editing HEC Data Storage System (DSS) files inside of WMS. HEC-HMS and HEC-RAS both use DSS files for importing and exporting time series data.

What is HEC-DSS?

DSS is a file format that is used for storing time series data (such as precipitation and discharge over time) and other types of data (such as unit hydrographs, elevation-area curves, and elevation-discharge curves). DSS files store one or more blocks, or records, in a single file with the ".dss" extension. Each record in the file has header information that identifies the units, start date, and/or start time of the information in the record. Each record is identified by a unique identifier called the "pathname". DSS files are binary files with no published format, and WMS uses DLL's provided by HEC to import and export data from these files.

If you understand the "pathnames" in DSS files, you will better understand how to use DSS files. DSS references data sets, or records, by their pathnames. A pathname may consist of up to 391 characters and is, by convention, separated into six parts, which may be up to 64 characters each. Pathnames are automatically translated into all upper case characters. They are separated into six parts (delimited by slashes "/") labeled "A" through "F", as follows:

For regular-interval time series data, the part naming convention is:

Part	Description
A	Project, river, or basin name
B	Location
C	Data parameter
D	Starting date of block, in a 9 character military format
E	Time interval
F	Additional user-defined descriptive information

A typical regular-interval time series might be:

/RED RIVER/BEND

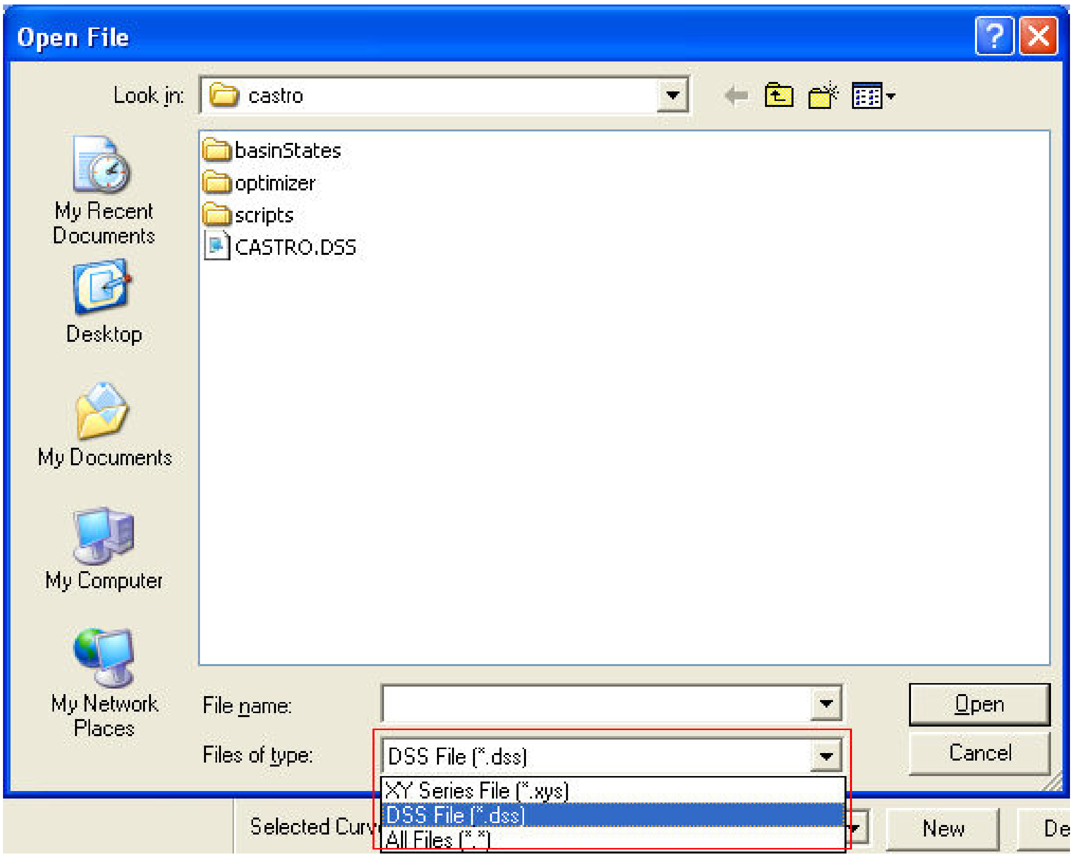
MARINA/FLOW/01JAN

Related Topics

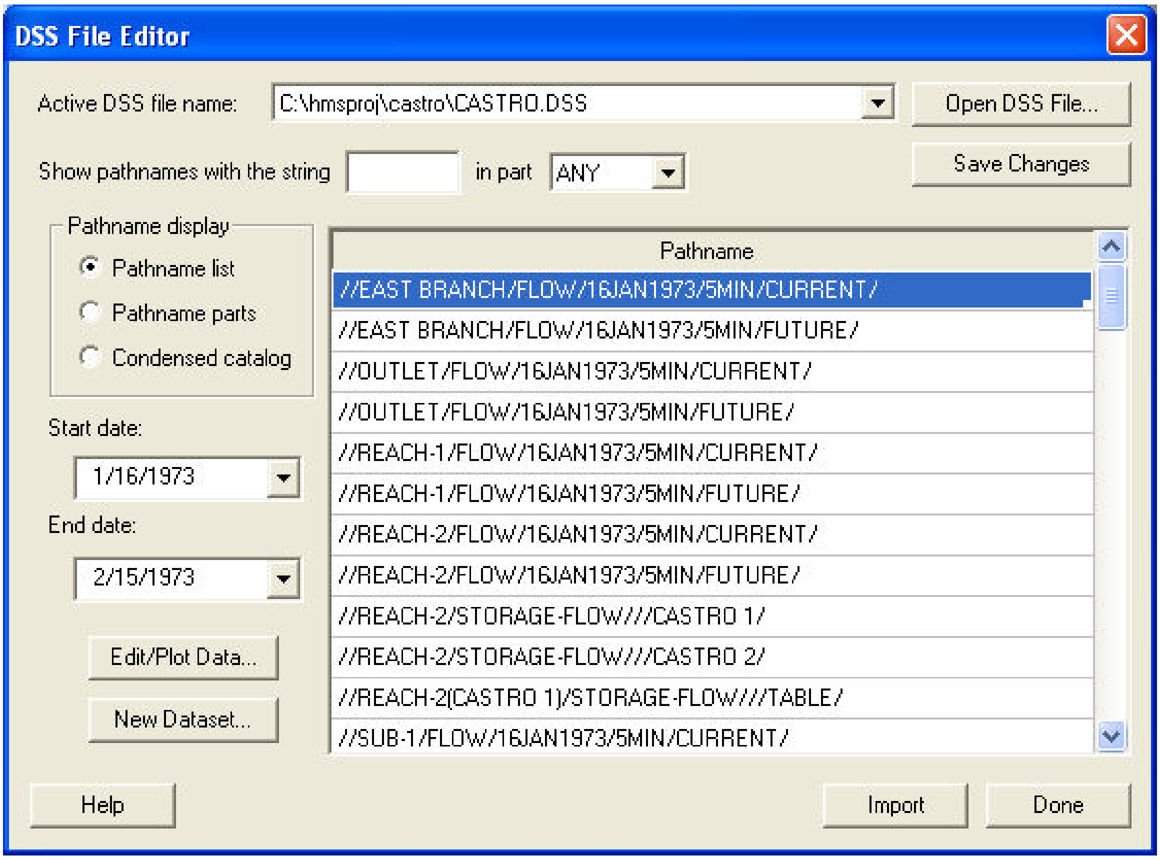
- DSS File I/O
- WMS Non-native Files
- DSS Interface

DSS File I/O

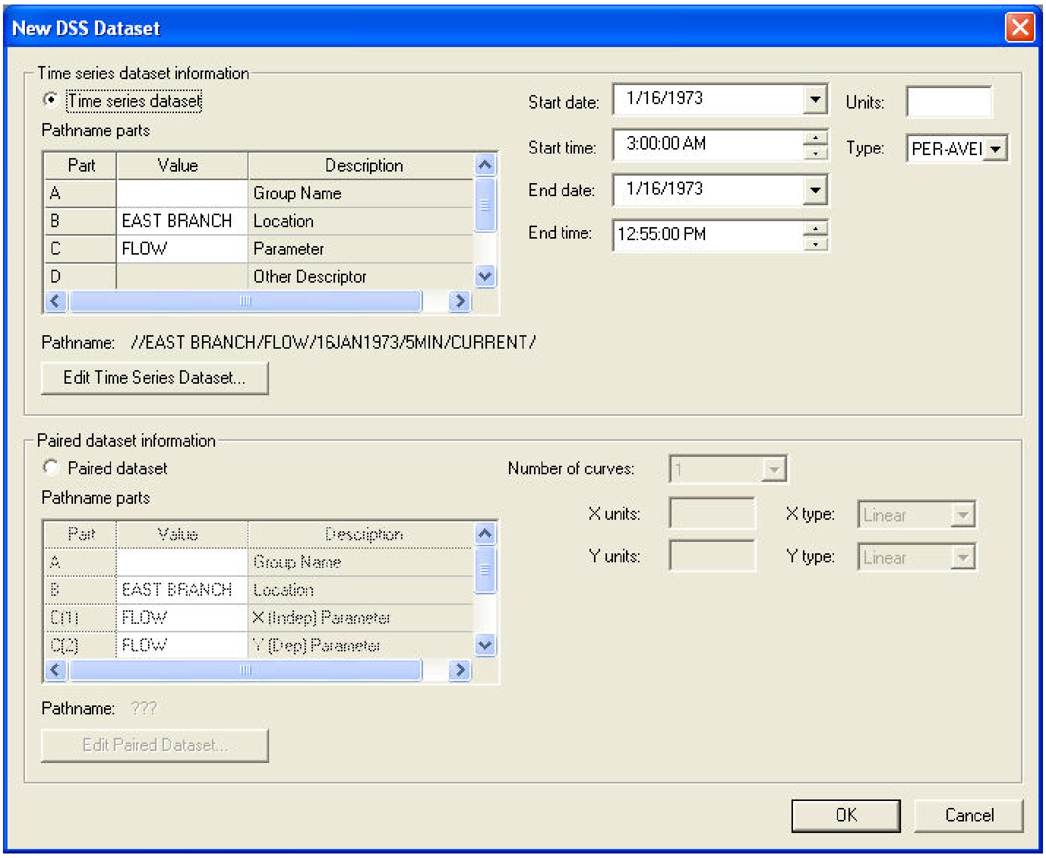
DSS files can be imported and exported from the XY Series Editor (using the Import/Export buttons) or from the Open command in the File menu. Selecting the Import command in the XY Series Editor will bring up the Open File dialog, which will have an option for reading a DSS file:



Selecting this option and opening a DSS file will bring the DSS file into the DSS file editor. Selecting a pathname and selecting the Import button in the DSS file editor will bring the data associated with the selected pathname and the selected start and end dates into the XY Series Editor.



Selecting the Export button from the XY Series Editor will allow you to export the selected xy series to a DSS file using the "New DSS Dataset" dialog:



Related Topics

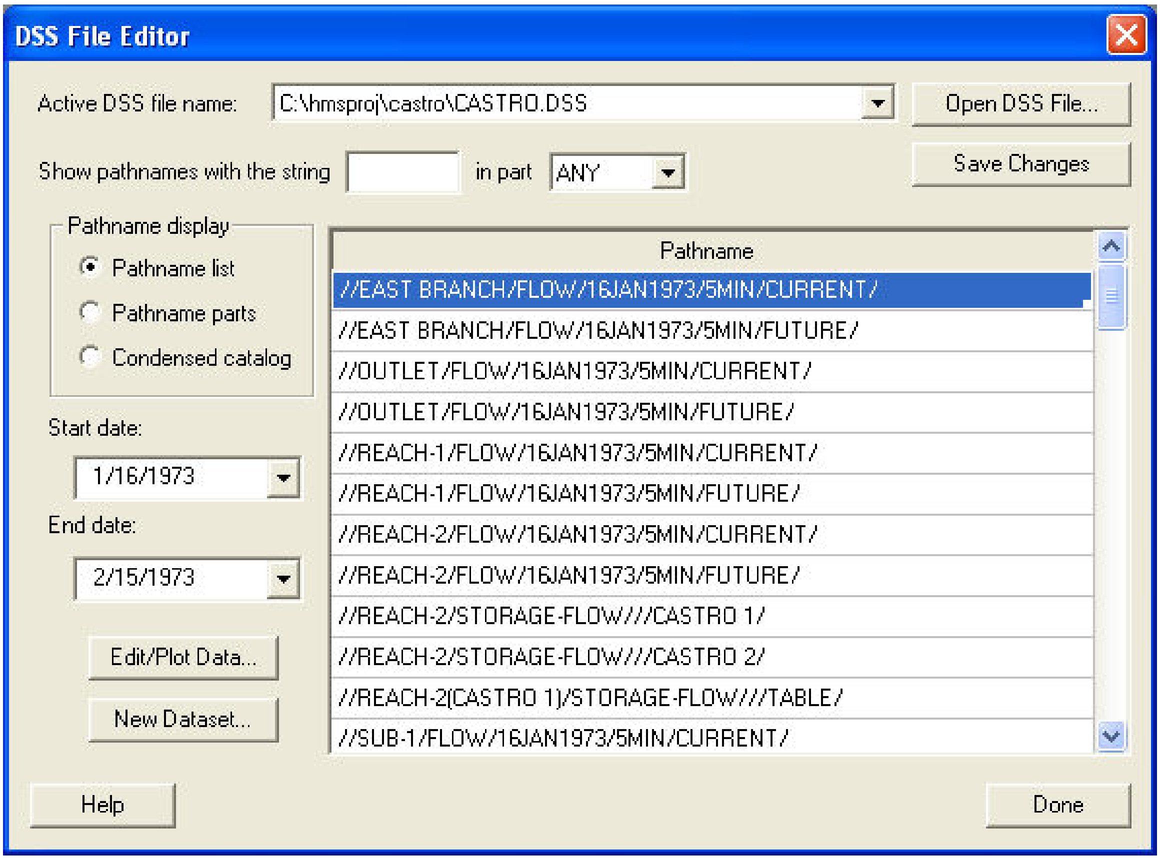
- DSS Files
- XY Series Files

DSS Files-WMS Interface

WMS has an interface to enable importing, exporting, viewing, and editing DSS files inside of WMS.

DSS File Editor

The DSS File Editor is used to view and edit a DSS file. This dialog comes up when you read a DSS file from the XY Series Editor or from the **Open** command in the **File** menu:



The following gives a description of the functionality of the DSS File Editor.

"Active DSS file name" combo box

This combo box displays a list of DSS file names that were selected using the “Open DSS File” button.

"Open DSS File" button

This command brings up the file browser dialog to open the DSS file. If a DSS file is selected, that DSS file (with its path) shows up in the “Active DSS file name” combo box and it becomes the active DSS file. A listing of the pathnames in this file is shown in the pathname spreadsheet.

"Save Changes" button

This button saves any changes made to the active DSS file.

"Show pathnames with the string" options

Remember that a single DSS file can have several pathnames, and that each pathname is broken up into six parts—A, B, C, D, E, and F. Since there can be hundreds of pathnames in a DSS file, it is useful to be able to search for text in the pathnames to find what you want. That’s what these options are for.

The edit field starts out with nothing in it. As the user types something in it, only those pathnames containing what has been typed are displayed. The search is case insensitive.

"Pathname display" group box

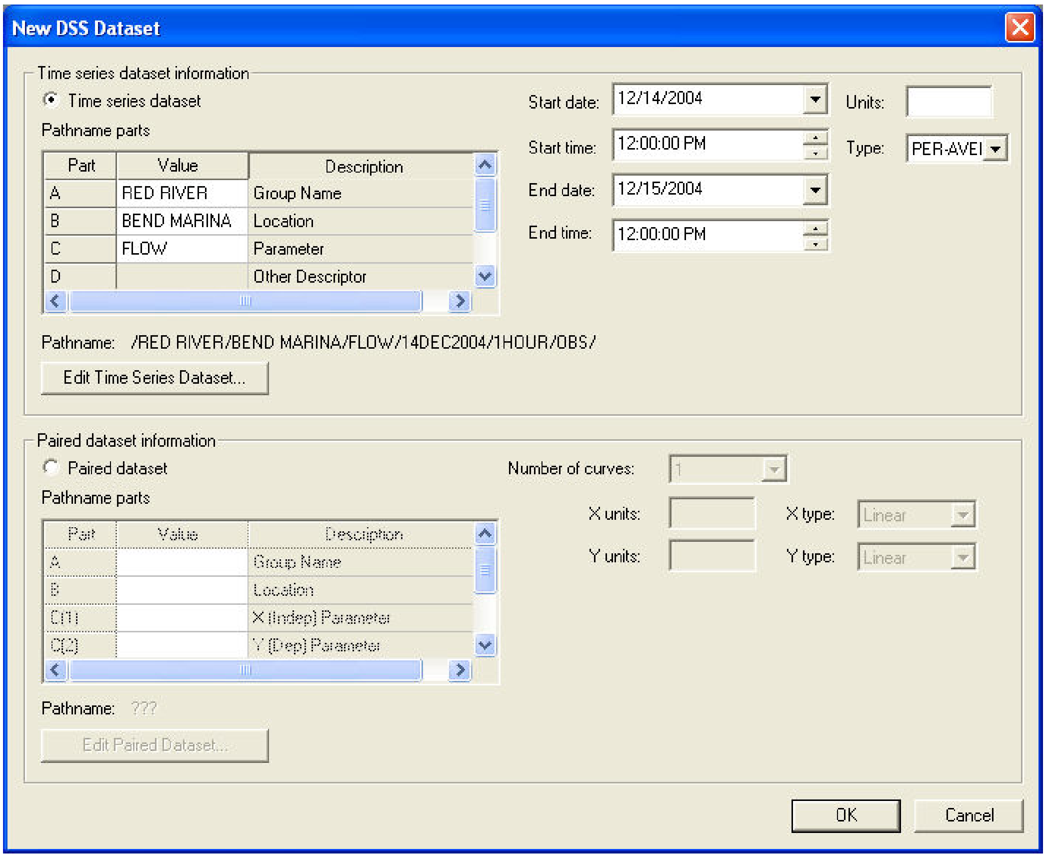
This radio group has options for different ways of displaying the pathname. The default will be the “Pathname list” option. This just displays the pathnames normally, with the “/” in between different parts. If the “Pathname parts” option is selected, WMS displays each part of the pathnames separately. The “Condensed catalog” option condenses the dates for different pathnames into a single pathname.

"Edit/Plot Data" button

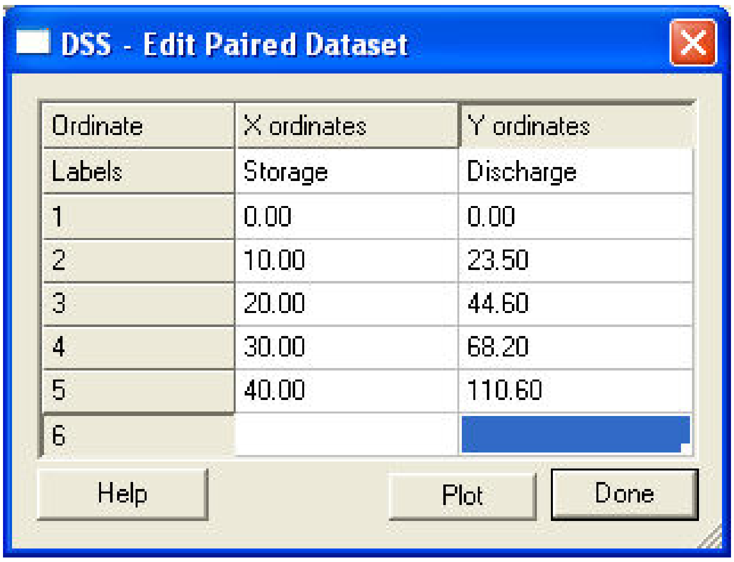
This button brings up the data from the DSS file for the selected pathname in the XY Series Editor, allowing you to edit and plot the data.

"New Dataset" button

This button brings up the New DSS Dataset dialog, allowing you to enter the pathname and other information for a new dataset:



Selecting the “Edit Time Series Dataset” button brings up the XY Series editor, allowing you to edit the time series dataset. Selecting the “Edit Paired Dataset” button brings up a spreadsheet (see below). The first editable column on this spreadsheet contains X values and the second, third, and other columns will contain Y data (Y-1, Y-2, Y-3,) up to the number of curves specified in the combo box in the New DSS Dataset dialog.



"Pathname" Spreadsheet

The "pathname" spreadsheet (in the DSS File Editor dialog) lists the pathnames of all the records in the active DSS file, based on the Pathname display options.

Related Topics

- DSS Files
- XY Series Files

File Extension

Here is a table of file extensions and their functions:

Extention	File Name	Description
.1	1 FILE	
.94	94 FILE	
.OCX	ACTIVE X CONTROL	
.PDF	Adobe Acrobat Document	
.EXE	APPLICATION	
.DLL	APPLICATION EXTENSION	
.ASC	ASCII Grid Files	WMS can import ARC/INFO® ASCII grid for use as a background DEM. Since it is a simple file format, other digital elevation data can be formatted in the same way and then imported into WMS using the Import Grid command in the DEMs menu.
.BASIN	BASIN FILE	
.BSN	BSN FILE	
.INI	CONFIGURATION SETTINGS	
.CSE	CSE FILE	
.DAT	DAT FILE	
.DBF	DBF FILE	
.DDF	USGS DEM	
.DEM	USGS DEM	
.DLG	Digital Line Graph	The DLG to Feature Arcs option allows a Digital Line Graph file to be imported and points connected into a series of arcs. USGS DLG files, like DEMs, can be downloaded via the Internet and are available for many parts of the US.
.DSS	HEC Data Storage System files	This format is used for importing and exporting time series and XY series data to and from HEC-RAS, HEC-HMS, and other HEC software.
.DTED	DTED Grid	
.DTM	This file format is no longer supported. If you are interested in using this file format, contact the WMS developers.	
.DWG	CAD Files	CAD files may also be imported and then converted to feature objects, TINs, or simply used to enhance the display of a project.
.DXF	CAD Files	
.ERR	ERR FILE	

.EX_	EX_ FILE	
.F01	F01 FILE	
.FAC	FAC FILE	
.FBC	FBC FILE	
.FDR	FDR FILE	
.FLT	FLT FILE	
.G01	GO1 FILE	
.GDM	GDM FILE	
.GGD	GRASS ASCII grid	
.GMT	GMT FILE	
.HC1	HC1 FILE	
.HDR	NED GRIDFLOAT or BIL Header	This is the format that should be used from the USGS's National Elevation Data set (NED). The default at this site is Arc/Info grid, but you should modify your request to the grid float format. The BIL format can also be used.
.HLP	HELP FILE	
.HTM	HTML DOCUMENT	
.ID	ID FILE	
.IMG	IMG FILE	
.INP	INP FILE	
.ISR	ISR FILE	
.ISU	ISU FILE	
.JGW	JGW FILE	
.JPG, JPEG	JPEG Image	Jpeg image format file.
.JPGW	JPGW FILE	
.JPW	JPW FILE	
.LAS	LAS FILE	
.MAP	LINKER ADDRESS MAP	
.LNP	LNP FILE	
.LOS	LOS FILE	
.LSF	LSF FILE	
.MAN	MAN FILE	
.MAT	MAT FILE	
.MDB	MDB FILE	
.PKG	MICROSOFT DEVELOPER EXTENSION	
.DOC	MISCOSOFT WORD DOCUMENT	
.COM	MS-DOS APPLICATION	

.BAT	MS-DOS BATCH FILE	
.NET	NET FILE	ARC/INFO TIN - This particular format saves node (vertex) coordinates, a list of edges and a list of triangles, making it possible to restore the TIN topology in WMS exactly as it was in ARC/INFO
.O01	O01 FILE	
.LIB	OBJECT LIBRARY	
.OCR	OCR LIBRARY	
.OUT	OUT FILE	
.P01	P01 FILE	
.PDN	PDN FILE	
.PRJ	PROJECT FILE	
.PTP	PTP FILE	
.R01	R01 FILE	
.RAS	RAS FILE	
.SBN	SBN FILE	
.SBX	SBX FILE	
.SDAT	SDAT FILE	
.SHP	Shapefiles	
.SHR	SHR FILE	
.SHX	SHX FILE	
.IDX	SQL SERVER REPLICATION SNAPSHOT INDEX SCRIPT	
.STO	STO FILE	
.SUP	WMS Superfile	Besides the tools in WMS for editing data imported from shapefiles, a special extension for ArcView has been created which allows you to create the outlets, streams, and basins themes, reorder streams, rename attributes, and then export to a WMS/ArcView super file which will allow you to open all three themes by importing the super file. This extension is placed in the wmshydro directory (under the main wms directory) when installing WMS. It is also available on the WMS website and has a separate document describing its usage. In order to be activated in ArcView you must move the extension file (.avx) to the Ext32 directory found under the ArcView® installation directories.
.TBL	TBL FILE	
.TDAT	TDAT FILE	
.TDN	TDN FILE	
.TXT	TEXT DOCUMENT	
.THY	THY FILE	
.TIF	TIF Image Data	A TIFF file can be imported and registered so that it appears as a "backdrop" in WMS or mapped to TIN data when shading.
.TIN	TIN FILE	
.TRE	TRE FILE	
.WDM	.WDM FILE	
.WPR	WMS PROJECT FILE	
.XY	XY FILE	

.XYZ	XYZ Data	Choosing this option allows you to read in a space-delimited file containing x, y, and z coordinate values. The Triangulate command in the TINs menu can then be used to create a TIN from the xyz data. The file must be space delimited in order for WMS to import it.

File I/O

Previously defined datasets can be input to WMS by selecting the **Open** menu command when right-clicking in the *Project Explorer* on the TIN/scatter set. This will bring up the file browser or a dialog with a list of file type options. The file types that can be imported to WMS as datasets are as follows:

- WMS ASCII Dataset Files
- WMS Binary Dataset Files
- ARC/INFO® ASCII Grid Files
- GRASS Grid Files

Additional formats will be added as new computational models are supported.

Once one of the file type options has been chosen, a file browser dialog appears and the user must select a file corresponding to the type selected.

Datasets can be exported from WMS to files by selecting the **Export** button in the *Data Browser*. Datasets can be saved as either binary or ASCII dataset files. Scatter point files can also be saved from the 2D scatter point module.

When a dataset is imported to WMS, a copy of the dataset is written to a temporary file on disk in binary form. If the imported dataset is already in the form of a WMS binary dataset file, a copy of the file is not made. When part of the dataset is needed it is loaded from the hard disk into internal memory. Only one time step of one scalar dataset is read into internal memory at any given time. This method of file manipulation reduces the amount of RAM required, but it requires extra hard disk space. It also requires that write permission is active in the WMS working directory.

When a new dataset is created through interpolation or using the data calculator, a temporary binary file is created for the dataset.

Related Topics

- Open
- ASCII Dataset Files
- Binary Dataset Files

Image Files

Image files are used in conjunction with TIFF files that have been previously imported to WMS and registered. They include the name of the TIFF file, the registration points, and the bounds of the clipping window. The format of the image file is shown in Figure 1 and a sample image file is shown in Figure 2.

IMAGE	/*File type identifier */
TIFF “filename”	/* Indicates the name of the tiff file used */
IMREGPTS	
PT1 u1 v1 x1 y1	
PT2 u2 v2 x2 y2	
PT3 u3 v3 x3 y3	
CLIPPOINT	
x1 x2	
y1 y2	

Figure 1. The Image File Format.

IMAGE
TIFF "jonescyn.tif"
IMREGPTS
PT1 0 756 422424.030700 4519460.893988
PT2 0 0 422424.030700 4515391.182075
PT3 715 0 426273.322285 4515391.182075
CLIPPOINT
422424.030700 426273.322285
4515391.182075 4519460.893988

Figure 2. Sample Image File.

The card types used in the Image file format are as follows:

Card Type	**IMAGE**
Description	File type identifier. Must be on first line of file. No fields.
Required	*YES*

Card Type	**TIFF**		
Description	Defines the name of the TIFF file to be displayed as an image.		
Required	YES		
Format	TIFF "filename"		
Sample	TIFF "jonescyn.tif"		
Field	*Variable*	*Value*	*Description*
1	filename	str	The name of the TIFF file.

Card Type	**PT1, PT2, PT3**		
Description	The three registration points used to define locations on a given image.		
Required	YES		
Format	PT1 tx1 ty1 wx1 wy1 PT2 tx2 ty2 wx2 wy2 PT3 tx3 ty3 wx3 wy3		
Sample	PT1 117 797 0.000000 10000.000000 PT2 117 88 0.000000 0.000000 PT3 1053 88 13220.0 0.000000		
Field	*Variable*	*Value*	*Description*
1-2	tx ty	±	Texture map coordinates.
3-4	wx wy	±	World coordinates.

Card Type	**CLIPPOINTS**		
Description	Defines the coordinates of the area in the TIFF file to be displayed as the image. (The area clipped and displayed from the TIFF file.)		
Required	YES		
Format	CLIPPOINTS xmin xmax ymin ymax		
Sample	CLIPPOINTS -628.990382 14338.471657 -857.665608 8354.617436		
Field	*Variable*	*Value*	*Description*
1-2	xmin xmax	±	Min and max values in the x direction.
3-4	ymin ymax	±	Min and max values in the y direction.

Import GIS File

WMS communicates the processed geometric data to HEC-RAS through the same GIS file that is used by the ArcView® Geo-RAS program. This file does not contain a complete model definition for a HEC-RAS project, but it does contain the river reach, cross section, and other important geometric components.

When importing the GIS file (this could be a file created by the Geo-RAS extension for ArcView®) a 1D-Hydraulic Centerline coverage, and 1D Cross Section Coverage will be created along with a new database where the cross section geometry is stored. As part of the **Import GIS File** command, you will be asked for a file name to save the cross section database to.

Related Topics

- Hydraulic Modeling
- Export GIS File
- Managing Cross Sections

Export GIS File

WMS communicates the processed geometric data to HEC-RAS through the same GIS file that is used by the ArcView® Geo-RAS program. This file does not contain a complete model definition for a HEC-RAS project, but it does contain the river reach, cross section, and other important geometric components. After the file has been exported, WMS will bring the exported hydraulic model into HEC-RAS as a new project.

1. Open your project in HEC-RAS
2. Select "File | Export GIS Data...". Select the data you would like to export, specify the location and name of the file in the top field (Export File:), and then select "Export Data". This exports the data so that WMS can recognize it. Click here for more information.
3. Close HEC-RAS completely.
4. Open WMS
5. Switch to the River Module, and change the model to HEC-RAS.
6. Select "HEC-RAS | Import GIS File"
7. Navigate to the folder where you exported the GIS data to in step 2 (it is saved to be in the same folder as your data). In the "Open" window, change the "Files of type:" field at the bottom to "All Files (*.*)"
8. Select the "sdf" file and select Open.
9. Specify where to save the cross section data and select save.

Related Topics

- Hydraulic Modeling
- Import GIS File

GRASS Grid (GSSHA Maps)

WMS can import GRASS ASCII grid for use as a background DEM. Since it is a simple file format, other digital elevation data can be formatted in the same way and then imported into WMS using the **Open** command in the *File* menu.

The GSSHA model also uses GRASS ASCII grid files format for all of the map parameters. The GRASS ASCII format is shown in Figure 1, and an example of file is shown in Figure 2.

north: n	/* Northern boundary grid coordinate */
south: s	/* Southern boundary grid coordinate */
east: e	/* Eastern boundary grid coordinate */
west: w	/* Western boundary grid coordinate */
rows: nrows	/* Number of rows in the grid */
cols: ncols	/* Number of columns in the grid */
z11 z12 z13 ... z1ncols	/* values of row 1 */
z21 z22 z23 ... z2ncols	/* values of row 2 */
.	
.	
.	
znrows1 znrows2 znrows3 ... znrowsncols	/* values of last row*/

Figure 1. GRASS ASCII Grid File.

north: 3451250
south: 3438850
east: 298960
west: 290860
rows: 10
cols: 5
0 1 1 0 1
1 1 0 0 1
.
.
0 0 1 1 0

Figure 2. Sample GRASS ASCII Grid File.

The card types used in the GRASS grid file format are self explanatory.

Land Use Files

When using a land use coverage or grid to map model parameters such as curve number, percent impervious, etc. a corresponding mapping file must be either created manually or imported into WMS. WMS can also export a mapping file created manually so that the same definitions can be remapped in a future model.

Depending on the application, one of three different land use tables will be required:

1. Mapping of land use to CN for hydrologic soil types A, B, C, and D.
2. Mapping of Green & Ampt parameters for the Maricopa County method.
3. Mapping of CASC2D attributes.

Descriptions and examples of each are given below.

Land use tables with corresponding curve numbers for different hydrologic soil groups vary from one text to another, or from one agency to another. For this reason, WMS supports user-defined tables. Users can create tables with the currently required data. The format of the table is shown in Figure 1, and a sample file in Figure 2.

ID1, "Land use description 1", CNA1, CNB1, CNC1, CND1
ID2, "Land use description 2", CNA2, CNB2, CNC2, CND2
ID3, "Land use description 3", CNA3, CNB3, CNC3, CND3
.
.
.
IDn, "Land use description n", CNAn, CNBn, CNCn, CNDn

Figure 1. Curve Number Land Use File Format.

1, "Fully developed urban areas, Poor Condition", 68, 79, 86, 89
2, "Paved parking lots, roofs, driveways, etc.", 98, 98, 98, 98
3, "Residential 1/8 acre lots (65% impervious)", 77, 85, 90, 92
4, "Residential 1/3 acre lots (30% impervious)", 57, 72, 81, 86

Figure 2. Sample Curve Number Land Use File.

No specific card types are required in this file but a description of each of the six fields required for each land use definition is given below. Each field must be separated by a comma, and the description string must be enclosed by double quotes.

Field	*Variable*	*Value*	*Description*
1	id	+	ID number of land use description.
2	description	str	Land use description.
3-6	curve#	+	SCS Curve Number (CN) for hydrologic soil groups A,B,C,D.

WMS can be used to map Green & Ampt infiltration parameters for HEC-1 using the Maricopa County methods. The file format and an example are given in Figure 3 and Figure 4.

ID1, "Land use description 1", IAB1, RTIMP1, PCTVEG1
ID2, "Land use description 2", IAB2, RTIMP2, PCTVEG2
ID3, "Land use description 3", IAB3, RTIMP3, PCTVEG3
.
.
.
IDn, "Land use description n", IABn, RTIMPn, PCTVEGn

Figure 3. Green & Ampt Land Use File.

20, "Mountainous shrub and brush", 0.3, 15.0, 50.0
19, "Mountainous forest", 0.25, 30.0, 50.0
29, "Mountainous grassland", 0.15, 55.0, 60.0
7, "Roadway", 0.3, 15.0, 50.0

Figure 4. Sample Green & Ampt Land Use File.

CASC2D map parameters may also be mapped to grid cells using a coverage or grid with an accompanying mapping file. The format of this file is given in Figure 5. The first part of the file serves as a dictionary. You may select any number of the “mappable” attributes listed, but values in the lower part of the table must appear in the same order as they are listed in the dictionary. An example table file is shown in Figure 6.

SOILSTABLE	/* File identifier */
HYDRAULIC_CONDUCTIVITY	/* First field identifier */
CAPILLARY	/* Second field identifier */
POROSITY	/* Third field identifier */
PORE_INDEX	/* Fourth field identifier */
RESIDUAL_SATURATION	/* Fifth field identifier */
MOISTURE_CONTENT	/* Sixth field identifier */
SURFACE_ROUGHNESS	/* Seventh field identifier */
INTERCEPTION_COEFF	/* Eighth field identifier */
STORAGE_CAPACITY	/* Ninth field identifier */
INITIAL_DEPTH	/* Tenth field identifier */
RETENTION	/* Eleventh field identifier */
AREA_REDUCTION	/* Twelfth field identifier */
ALBEDO	/* Thirteenth field identifier */
WILTING_POINT	/* Fourteenth field identifier */
VHEIGHT	/* Fifteenth field identifier */
TCOEFF	/* Sixteenth field identifier */
CANOPY	/* Seventeenth field identifier */
SOIL_ERODABILITY	/* Eighteenth field identifier */
CROP_MANAGEMENT	/* Nineteenth field identifier */
CONSERVATION_PRACTICE	/* Twentieth field identifier */

SAND_MAP	/* Twenty-first field identifier */
SILT_MAP	/* Twenty-second field identifier */
NUMSOILS n	/* Number of soils types in this file */
id1 "description1" hc1 cp1 po1 pi1 rs1 mc1 sr1 ic1 sc1 id1 rt1 ar1 ab1 wp1 vh1 tc1 ca1 se1 cm1 co1 sm1 si1	
id2 "description2" hc2 cp2 po2 pi2 rs2 mc2 sr2 ic2 sc2 id2 rt2 ar2 ab2 wp2 vh2 tc2 ca2 se2 cm2 co2 sm2 si2	
.	
.	
.	
idn "descriptionn" hcn cpn pon pin rsn mcn srn icn scn idn rtn arn abn wpn vhn tcn can sen cmn con smn sin	

Figure 5. CASC2D Attribute Mapping File Format.

SOILSTABLE
HYDRAULIC_CONDUCTIVITY
CAPILLARY
POROSITY
PORE_INDEX
RESIDUAL_SATURATION
MOISTURE_CONTENT
SURFACE_ROUGHNESS
NUMSOILS 4
1 "Loamy" 0.001000 0.010000 0.100000 0.250000 0.100000 1.000000 0.200000
2 "Sandy" 0.002000 0.010000 0.100000 0.250000 0.100000 1.000000 0.100000
3 "Silty" 0.003000 0.010000 0.100000 0.250000 0.100000 1.000000 0.050000
4 "Clay" 0.004000 0.010000 0.100000 0.250000 0.100000 1.000000 0.030000

Figure 6. Sample CASC2D Attribute Mapping File.

Read Stage File

Water levels are used in WMS as scattered data sets and can be imported as scattered data set (x, y, z). Files can be in a simple delimited format as shown here within a text file or spreadsheet and imported through the Text Import Wizard.

"id" "x" "y" "stg1" "stg2" 1 110 0 45 47 2 110 20 48 49 3 110 40 51 53

Data can also be formatted in a scatter point file, or in the same stage file format used in earlier versions of WMS (version 6.0 and earlier).

Since water levels are imported as a scatter data set and stored completely separate from the TIN, it is not necessary to match water level locations with TIN vertices (but it is okay if they are). This also removes the necessity of creating stream in the TIN. This flexibility provides the opportunity to incorporate water levels along rivers as well as over the floodplains that are either observed or simulated from a river hydraulic model.

One of the problems, though, with this new format is that there is no longer a way to "interpolate" stage values between known fixed stages as could be done in previous versions. If you wish to bring forward your data from earlier versions of WMS then you should read the stage file into the older version, interpolate the stage between fixed stage points, and save the stage file (including both the fixed and interpolated stages). Version 7.0 of WMS will then read this stage file and create scattered data points for each of the locations in the old format stage file.

Related Topics

- Overview of Flood Plain Delineation
- Delineate Flood Plain
- Text Import Wizard

Read a Stage File

Water levels are used in WMS as scattered data sets and can be imported as scattered data set (x, y, z). Files can be in a simple delimited format as shown here within a text file or spreadsheet and imported through the Text Import Wizard.

"id"	"x"	"y"	"stg1"	"stg2"
1	110	0	45	47
2	110	20	48	49
3	110	40	51	53

Data can also be formatted in a scatter point file, or in the same stage file format used in earlier versions of WMS (version 6.0 and earlier).

Since water levels are imported as a scatter data set and stored completely separate from the TIN, it is not necessary to match water level locations with TIN vertices (but it is okay if they are). This also removes the necessity of creating stream in the TIN. This flexibility provides the opportunity to incorporate water levels along rivers as well as over the floodplains that are either observed or simulated from a river hydraulic model.

One of the problems, though, with this new format is that there is no longer a way to "interpolate" stage values between known fixed stages as could be done in previous versions. If you wish to bring forward your data from earlier versions of WMS then you should read the stage file into the older version, interpolate the stage between fixed stage points, and save the stage file (including both the fixed and interpolated stages). Version 7.0 of WMS will then read this stage file and create scattered data points for each of the locations in the old format stage file.

Related Topics

- Overview of Flood Plain Delineation
- Delineate Flood Plain
- Text Import Wizard

Shapefiles

One of the most common methods for creating feature objects is to import a shapefile. The concept of a shapefile was established by Environmental Systems Research Institute (ESRI) in their ArcView® program and it has become the defacto standard for sharing GIS vector data (points, lines, and polygons).

A shape file is actually comprised of three or more files. The primary file is the *.shp and it contains the geometric information (coordinates and if necessary connectivity of the points, lines, polygons). The .dbf file is a standard database file and stores the attributes of the feature objects. Finally, there will be a *.shx file which is an indexing file. There may be a few other files that accompany the shapefile and so you should always move them around together if you are copying or moving them to a new directory.

Only one "theme" or type of feature can exist in a shapefile. For example you cannot store points and polygons in the shapefile, or streams and basin boundaries and so you may be required to import multiple files to make up the drainage coverage in WMS.

WMS includes all of the tools necessary to import shapefiles and convert the geometric and attribute information into feature objects. This can be done by directly opening the shapefile and converting to feature objects in the active coverage or by loading the shapefile in the GIS module.

Related Topics

- Importing Shapefiles
- GIS Module

WMS Native Files

File formats for most of the files native to (determined by) WMS can be accessed from the list below. Files which are used by analysis codes such as HEC-1 are not documented here since they are described in the documentation for the codes.

Most of the files used by WMS have a card type format. With this format, the different components of the file are grouped into logical groups called "cards." The first component of each card is a short name which serves as the identifier. The remaining fields on the line contain the information associated with the card. In some cases, such as lists, a card can use multiple lines.

There are many other files that are non-native to WMS that can be opened. A list of these file types and their default extensions can be found in the WMS Non-native Files help page.

The following are file formats that WMS uses:

- 2D Grid Files
- 2D Scatter Point Files
- ARC/INFO® ASCII Grid Files
- ASCII Data Set Files
- Binary Data Set Files
- DEM Files
- GRASS Grid
- Hydrograph Files
- Image Files
- Land Use Files
- Soil Type Runoff Coefficient Files

- TIN Files
- WMS Project Files
- XY Series Files

Related Links

- WMS Supported Formats
- WMS Non-native Files
- Opening Files
- Importing Text Files

WMS Non-native Files

The following table identifies the different kinds of files that can be imported (now opened) into WMS, along with the typical three letter extension used by each. Note that there is additional information on importing DEMs and ArcView® Shapefiles.

In addition to the file types listed below, several other types of data can be imported via the Text Import Wizard.

File Type	File Ext	Description
CAD	*.dxf or *.dwg	CAD files may also be imported and then converted to feature objects, TINs, or simply used to enhance the display of a project. Click here for more information on importing DXF Files.
TIFF	*.tif	A TIFF file can be imported and registered so that it appears as a "backdrop" in WMS or mapped to TIN data when shading. Click here for more details on importing TIFF image data.
DLG	*.dlg	The *DLG to Feature Arcs* option allows a *Digital Line Graph* file to be imported and points connected into a series of arcs. USGS DLG files, like DEMs, can be downloaded via the Internet (see the introduction to the DEMs module) and are available for many parts of the US.
Land Use	*.tbl	
RC Soil Table	*.tbl	
Feature Object Polygons Shapefile	*.shp	
Feature Object Arcs Shapefile	*.shp	
Feature Object Points Shapefile	*.shp	
ArcView®/WMS Superfile	*.sup	Besides the tools in WMS for editing data imported from shapefiles, a special extension for ArcView® 3.3 and earlier has been created [1] which allows you to create the outlets, streams, and basins themes, reorder streams, rename attributes, and then export to a WMS/ArcView® super file which will allow you to open all three themes by importing the super file. This extension may be available on the WMS website [2] and has a separate document describing its usage. In order to be activated in ArcView® you must move the extension file (.avx) to the Ext32 directory found under the ArcView® installation directories.
Arc/Info Grid	*.asc	

GRASS Grid	*.ggd	
Arc/Info Tin	*.net	An ARC/INFO TIN can be imported if it has been exported from ARC/INFO using the **UNGENERATETIN** command with the *NET* option. This particular format saves node (vertex) coordinates, a list of edges and a list of triangles, making it possible to restore the TIN topology in WMS exactly as it was in ARC/INFO. Once imported it can be used to perform all of the watershed characterization operations available in WMS.
Greenampt Soil	*.tbl	
Greenampt Landuse	*.tbl	
HSPF Landuse	*.tbl	
USGS DEM	*.dem *.ddf	The USGS and other government and proprietary agencies have distributed both the 1:250,000 and 1:24,000 scale USGS digital elevation files in a USGS-defined format for a number of years. The 1:250,000 DEMs available for download from the USGS web site remains in this format. However the 1:24000 DEMs are now distribute on this site in the SDTS format. Other common DEM file formats include ARC/INFO ASCII Grid, DTED, and GRASS. All of these dialogs are imported in WMS in essentially the same way as described below, but you will need to know which format the DEM is in prior to reading the DEMs for use in WMS (particularly in the case of the commonly used USGS 1:24000 DEMs that are available in both the USGS and SDTS formats). The DEM file type is specified in the Import File dialog and include all five of the previously mentioned formats (the USGS DEM option refers to both the older single file format and the newer SDTS format). Once you specify the format type the *Import DEMs* dialog controls file selection as well as other import options. For example, multiple files of the same format may be "tiled" together when importing, but you cannot mix and match between two or more different formats. NOTE: ARC/INFO grids that are to be imported must be saved as ASCII files from ARC/INFO® in either feet or meter units and not latitude-longitude.
DTED Grid	*.dted *.*	
Arc/Info Watershed Grid	*.asc	
[[WMS:GRASS Grid (GSSHA Maps)|GRASS Watershed Grid	*.ggdd	
[[WMS:TIN Files|XYZ Data->TIN Verticies	*.xyz	Choosing this option allows you to read in a space-delimited file containing x, y, and z coordinate values. The Triangulate command in the *TINs* menu can then be used to create a TIN from the xyz data. The file must be space delimited in order for WMS to import it.
DEM Attribute	*.*	
XMRG/NexRAD	*.*	This option was developed by personnel at the Waterways Experiment Station (WES) for importing “in-house” formatted files of NEXRAD precipitation. If you have access to data from WES contact us for what needs to be done to import this into WMS. The WMS developers would like to be able to support more standard formatted files of NEXRAD data. If you have additional information please contact the developers for help in implementation.
Intergraph DTM	*.dtm	This file format is no longer supported. If you are interested in using this file format, contact the WMS developers.
JPEG Image	*.jpg *.jpeg	Jpeg image format file.
[[WMS:DSS Files|DSS	*.dss	This is the HEC-DSS (Hydrologic Engineering Center-Data Storage System) file format. This format is used for importing and exporting time series and XY series data to and from HEC-RAS, HEC-HMS, and other HEC software.
NED GRIDFLOAT or BIL Header	*.hdr	This is the format that should be used from the USGS's National Elevation Dataset (NED). The default at this site is Arc/Info grid, but you should modify your request to the grid float format. The BIL format can also be used.

Related Topics

- WMS Supported File Formats
- WMS Native File Formats
- Opening Files

References

[1] http://wms.aquaveo.com/wmsavext.exe
[2] http://ems-i.com/

Soil Type Runoff Coefficient Files

Like land use, soil type coverages/grids can be used to map model parameters. Tables with corresponding runoff coefficients, Green & Ampt parameters, or CASC2D values for different can be mapped. The CASC2D tables are the same as defined in the land use section. Runoff coefficient table definitions and examples are given in Figure 1 and Figure 2.

ID1, "Soil type description 1", RC1
ID2, "Soil type description 2", RC2
ID3, "Soil type description 3", RC3
.
.
.
IDn, "Soil type description n", RCn

Figure 1. Soil Type Runoff Coefficient File Format.

1, "Highly impervious", 1.0
2, "Slightly pervious", .9
3, "Moderately pervious", .75
4, "Highly pervious", .5

Figure 2. Sample Soil Type Runoff Coefficient File.

Green & Ampt mapping file definitions and an example are given in Figure 3 and Figure 4.

ID1, "Soil type description 1", XKSAT1, RTIMP1, PCTEFFECTIVE1
ID2, "Soil type description 2", XKSAT2, RTIMP2, PCTEFFECTIVE2
ID3, "Soil type description 3", XKSAT3, RTIMP3, PCTEFFECTIVE3
.
.
.
IDn, "Soil type description n", XKSATn, RTIMPn, PCTEFFECTIVEn

Figure 3. Green & Ampt Soil Table File Format.

19, "Gunsight-Cipriano complex, 1-7% Slope", 0.63, 50.0, 0.0
18, "Greyeagle-Suncity Variant complex, 1-7% Slope", 0.23, 50.0, 0.0
17, "Gachado-Lomitas-Rock outcrop complex, 7-55% Slope", 0.16, 50.0, 20.0
72, "Lehmans-Rock outcrop complex, 8-65% Slope", 0.09, 50.0, 30.0

Figure 4. Sample Green & Ampt Soil Table File.

WMS Supported File Formats

WMS supports several file formats and all are opened using the **File | Open** command. Some of the file formats are native to WMS whereas others are formats defined by other programs or agencies (such as ArcView® or AutoCAD or the USGS). A list of native file formats and Non-native file formats can be viewed.

Previous versions of WMS used an Import command for non-native file formats while the **Open** command was reserved for native file formats. In this version all files are opened using the **Open** command, using the file extension drop-down list in the Open dialog to specify the type of file being opened. The list of file extensions supported is long and so WMS will try to show the types of files used in a specific module by default, but any file can be opened from any module, even if the file extension is not listed.

Related Topics

- Opening Files
- Native WMS Files
- Non-native WMS Files
- Importing Shapefiles

WMS Project Files

A WMS project file is a file which contains a list of other files. If a project file is selected using the **Open** command in the *File* menu, each of the files listed in the project file are opened and read in. This makes it possible to quickly read in several files without having to identify each file individually in the file browser. Most of the files are native-WMS file types, however other files such as DXF, and HEC-1 files can be included in order to maintain all files pertaining to a given project together.

The project file format, with all of the possible options, is shown in s without having to identify each file individually in the file browser. Most of the files are files created by WMS, however other files such as DXF, and HEC-1 files can be included in order to maintain all files pertaining to a given project together.

The project file format, with many of the possible options, is shown in Figure 1. The first line in the file is a PROJECT card that identifies the file as a project file. Each of the other cards has a file type identifier followed by a file name. With few exceptions, the file name should not contain the directory path. Any suffix may be used with the file name, but the default extension names are shown. In general you will want to save your project file in a separate directory since all of the constituent files will also be saved in this directory.

PROJECT	/* Project file type identifier */
DEFAULTS FILE.INI	/* Default settings file */
TIN FILE.TIN	/* TIN file */
DEM FILE.GRD	/* DEM file */
MAP FILE.MAP	/* MAP file */
GRID2D FILE.2DG	/* 2D Grid file */
SCATTER FILE.XY	/* Scatter point file */
HEC1 FILE.HC1	/* HEC-1 file */
TR20 FILE.DAT	/* TR-20 file */
DATA FILE.SCL	/* Data set file */
1DMODEL "1DMODEL"	/* Current 1D Hydraulic Model */
STOCHASTIC FILE.STO	/* Stochastic parameters file */
IMAGE FILE.IMG	/* Image registration file */
DXF C:\WMS\DATA\FILE.DXF	/* DXF file */

Figure 1. Project File Format.

Related Topics

- WMS Supported Formats
- File Open
- File Save

XY Series Files

The XY Series Editor is used in several places in WMS. The XY Series Editor is a general purpose editor for entering curves or pairs of lists of data. The XY Series Editor allows a curve to be imported from a file, created and edited graphically, or created and edited using two columns of edit fields in a spreadsheet-like interface.

XY series files can be used to prepare a set of curves for import to the XY Series Editor. XY series files are also used to export curves generated within the Editor for future use.

The format of the XY Series File is shown in Figure 1, and a sample file is shown in Figure 2. Curves are defined in an XY Series File using one of three types of cards: XY1, XY2, or XY3. With the XY1 card, both the x and y values are listed for each point on the curve. There is no limit to the spacing or interval used between subsequent x values. The XY2 card is identical to the XY1 card except that the number of points and the x values are assumed to be static and cannot be altered by the user. With the XY3 card, the x values are defined by a beginning x value, an initial increment in x, and a percent change in x per increment. Only the y values are explicitly listed.

XY1 id n dx dy rep begc name	/* XY Series vers. #1 */
x_1 y_1	/* XY values */
x_2 y_2	/* XY values */
.	
.	
x_n y_n	
XY2 id n dx dy rep begc name	/* XY Series vers. #2 */
x_1 y_1	/* XY values */
x_2 y_2	
.	
.	
x_n y_n	
XY3 id n x1 incx pcx dx dy rep begc name	/* XY Series vers. #2 */
y_1	/* Y values */
y_2	
.	
.	
y_n	

Figure 1. The XY Series File Format.

XY3 1 241 0 6 0 0 0 0 0 typeI-24hour
0.00000
0.00174
.
.
1.0000

Figure 2. The Sample XY Series File.

The card types used in the XY series file format are as follows:

Card Type	**XY1**		
Description	Defines a curve with a list of XY values. Any number of points and any x spacing between points may be used.		
Required	NO		
Format	XY1 id n dx dy rep begc name x_1 y_1 x_2 y_2 . . x_n y_n		
Sample	XY1 1 5 0 0 0 0 head 0.0 0.0 1.0 2.0 2.5 7.0 3.0 8.0 4.5 9.5		
'Field	*Variable*	Value	*Description*
1	id	+	The id of the XY series.
2	n	+	The number of point in the series.
3	dx	0,1	A flag defining whether the x values listed are to be interpreted as incremental (dx=1) or absolute (dx=0).
4	dy	0,1	A flag defining whether the y values listed are to be interpreted as incremental (dy=1) or absolute (dy=0).
5	rep	0,1	A flag defining whether the xy series is to be interpreted as cyclic (repeating)
6	begc	"+/-"	The x value in the series where the cyclic portion of the curve begins. Value is ignored if rep=0.
7	name	str	The name of the series
8-9	x,y	"+/-"	The xy values of the points defining the curve. Repeat n times.

Card Type	**XY3**
Decription	Defines a curve with a list of XY values. This card is identical to the XY1 card except that the number of points and the x values are assumed to be static and cannot be altered by the user.

<table>
<tr><td>Card Type</td><td colspan="3">XY3</td></tr>
<tr><td>Description</td><td colspan="3">Defines a curve with a list of Y values. The x values are defined by a beginning value, an increment, and a bias.</td></tr>
<tr><td>Required</td><td colspan="3">NO</td></tr>
<tr><td>Format</td><td colspan="3">XY3 id n x1 incx biasx dx dy rep begc name
y_1
y_2
.
.
y_n</td></tr>
<tr><td>Sample</td><td colspan="3">XY3 1 10 0 1 0 0 0 0 0 head
0.0
2.0
7.0
8.0
9.5</td></tr>
<tr><td>'Field</td><td>Variable</td><td>Value</td><td>Description</td></tr>
<tr><td>1</td><td>id</td><td>+</td><td>The id of the XY series.</td></tr>
<tr><td>2</td><td>n</td><td>+</td><td>The number of point in the series.</td></tr>
<tr><td>3</td><td>x1</td><td>"+/-"</td><td>The first x value.</td></tr>
<tr><td>4</td><td>incx</td><td>"+/-"</td><td>The increment in x used to compute the next x value.</td></tr>
<tr><td>5</td><td>pcx</td><td>+</td><td>The per cent change in x used to compute subsequent x values. Expressed as a decimal, i.e., 0.05 = 5%.</td></tr>
<tr><td>6</td><td>dx</td><td>0,1</td><td>A flag defining whether the x values listed are to be interpreted as incremental (dx=1) or absolute (dx=0).</td></tr>
<tr><td>7</td><td>dy</td><td>0,1</td><td>A flag defining whether the y values listed are to be interpreted as incremental (dy=1) or absolute (dy=0).</td></tr>
<tr><td>8</td><td>rep</td><td>0,1</td><td>A flag defining whether the xy series is to be interpreted as cyclic (repeating)</td></tr>
<tr><td>9</td><td>begc</td><td>"+/-"</td><td>The x value in the series where the cyclic portion of the curve begins. Value is ignored if rep=0.</td></tr>
<tr><td>10</td><td>name</td><td>str</td><td>The name of the series</td></tr>
<tr><td>11</td><td>y</td><td>"+/-"</td><td>The y values of the points defining the curve. Repeat n times.</td></tr>
</table>

Related Topics

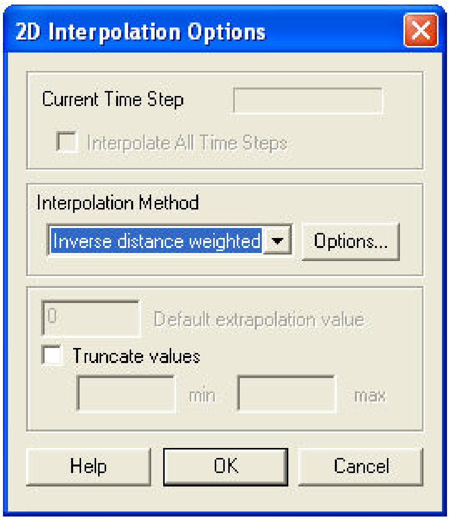
- XY Series Editor
- DSS Files
- WMS Native Files

4.1 Interpolation

Interpolation Options

Scatter point sets are used for interpolation to other data types such as grids or basin centroids. Since no interpolation scheme is superior in all cases, several interpolation techniques are provided in WMS.

The interpolation option is selected using the *Interpolation Options* dialog accessed through the **Interpolation Options** command in the *Interpolation* menu. Once an option is selected, that option is used for all subsequent interpolation commands (interpolation of elevations from DEM/TIN to TINs or 2D Grids, flood plain delineation, etc.).



Interpolation is always performed using the active scatter point set, or TIN. By default the active data set and time step are interpolated. The active dataset and time step can be selected from the Project Explorer or using the **Dataset** button at the top of the *Interpolation Options* dialog. This button also allows interpolation from all time steps of a transient dataset to be performed.

When interpolating a set of values, it is sometimes useful to limit the interpolated values to lie between a minimum and maximum value. For example, when interpolating rainfall values, a negative value of rainfall is meaningless. However, some interpolation schemes will produce negative values even if all of the scatter points have positive data values. This occurs in areas where the trend in the data is toward a zero value. The interpolation may extend the trend beyond a zero value into the negative range. In such cases it is useful to limit the minimum interpolated value to zero. Interpolated values can be limited to a given range by selecting the Truncate values option in the *Interpolation Options* dialog and entering a minimum and maximum interpolation value.

The interpolation methods are listed in the Interpolation Options dialog. To the right of most of the method names is a button used to bring up a dialog for entering more interpolation options specific to the interpolation method. The methods supported for 2D interpolation are:

- Linear
- Inverse distance weighted
- Clough - Tocher
- Natural neighbor

Interpolation of Rainfall to Basin Centroids

The **Interpolate to Basin Precip** command in the *Interpolation* menu is designed to interpolate rainfall values at scattered points to the xy series representing rainfall for a basin in either HEC-1 or TR-20. The scattered points typically represent either gaging stations or radar locations for NEXRAD data. Unlike interpolation to grids, this command does not use the active interpolation method, but rather uses the Thiessen method in order to assign the weights of each scatter point for each basin.

In addition to interpolating rainfall values to basin centroids it is often convenient to interpolate the rainfall values to a grid so that an animation sequence of a storm can be generated. The *Bounding Grid* options described below is useful for setting up a grid for this purpose.

Interpolation to Grids

Once an interpolation scheme has been selected and all of the parameters for the selected scheme have been input, the data associated with the active time step and dataset of the active scatter point set can be interpolated to a grid. During the interpolation process, a new dataset is constructed for the grid containing the interpolated values.

The interpolation is done either to the grid nodes or to the grid cell centers depending on whether the grid is a mesh or cell centered grid.

Interpolation Options for Floodplain Delineation

Since stages are defined by a 2D scatter set, the same interpolation options are used for flood plain delineation as are used in the 2D Scatter module.

Related Topics

- Flood Plain Delineation

Linear Interpolation

If the linear interpolation scheme is selected, the 2D scatter points are first triangulated to form a temporary TIN. The TIN is a network of triangles connecting the scatter points together. It is used to interpolate from the scatter points to another object such as a grid or a mesh.

The equation of the plane defined by the three vertices of a triangle is as follows:

$$Ax + By + Cz + D = 0$$

where A, B, C, and $D$are computed from the coordinates of the three vertices $(x1, y1, z1)$, $(x2, y2, z2)$, and $(x3, y3, z3)$:

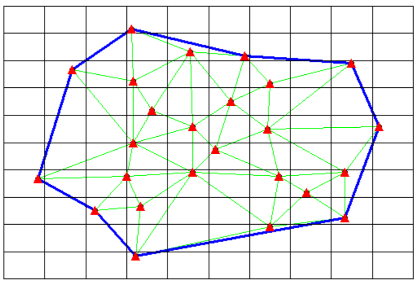
$$A = y_1(z_2 - z_3) + y_2(z_3 - z_1) + y_3(z_1 - z_2)$$
$$B = z_1(x_2 - x_3) + z_2(x_3 - x_1) + z_3(x_1 - x_2)$$
$$C = x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)$$
$$D = -Ax_1 - By_1 - Dz_1$$

The plane equation can also be written as:

$$z = f(x, y) = -\frac{A}{C}x - \frac{B}{C}y - \frac{D}{C}$$

which is the form of the plane equation used to compute the elevation at any point on the triangle.

Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the linear interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the default extrapolation value entered at the bottom of the *Interpolation Options* dialog. The figure below shows a 2D scatter point set (small red triangles) being interpolated to a 2D grid. The green lines represent a TIN constructed from a scatter point set. The thick blue line represents the convex hull of the dataset. No extrapolation will occur outside of this thick blue line.



Convex Hull of a Dataset

Related Topics

- Interpolation Options
- 2D Scatter Point Module

Inverse Distance Weighted Interpolation

One of the most commonly used techniques for interpolation of scatter points is inverse distance weighted (IDW) interpolation. Inverse distance weighted methods are based on the assumption that the interpolating surface should be influenced most by the nearby points and less by the more distant points. The interpolating surface is a weighted average of the scatter points and the weight assigned to each scatter point diminishes as the distance from the interpolation point to the scatter point increases. Several options are available for inverse distance weighted interpolation. The options are selected using the *Inverse Distance Weighted Interpolation Options* dialog. This dialog is accessed through the Options button next to the Inverse distance weighted item in the *2D Interpolation Options (3D Interpolation Options)* dialog. The options in the dialog are as follows:

Shepards Method

The simplest form of inverse distance weighted interpolation is sometimes called "Shepard's method" (Shepard 1968). The equation used is as follows:

$$F(x,y) = \sum_{i-1}^{n} w_i f_i$$

where $n$is the number of scatter points in the set, $f_i$are the prescribed function values at the scatter points (e.g. the dataset values), and $w_i$are the weight functions assigned to each scatter point. The classical form of the weight function is:

$$w_i = \frac{h_i^{-p}}{\sum_{j-1}^{n} h_j^{-p}}$$

where $p$is an arbitrary positive real number called the power parameter (typically, $p = 2$) and $h_i$is the distance from the scatter point to the interpolation point or

$$h_i = \sqrt{(x-x_i)^2 + (y-y_i)^2}$$

where $(x,y)$are the coordinates of the interpolation point and $(x_i, y_i)$are the coordinates of each scatter point. The weight function varies from a value of unity at the scatter point to a value approaching zero as the distance from the scatter point increases. The weight functions are normalized so that the weights sum to unity.

Although the weight function shown above is the classical form of the weight function in inverse distance weighted interpolation, the following equation is used in WMS:

$$w_i = \frac{\left[\frac{R-h_i}{Rh_i}\right]^{-2}}{\sum_{j-i}^{n} \left[\frac{R-h_j}{Rh_j}\right]^{-2}}$$

where $h_i$is the distance from the interpolation point to scatter point i, $R$is the distance from the interpolation point to the most distant scatter point, and $n$is the total number of scatter points. This equation has been found to give superior results to the classical equation (Franke & Nielson, 1980).

The weight function is a function of Euclidean distance and is radially symmetric about each scatter point. As a result, the interpolating surface is somewhat symmetric about each point and tends toward the mean value of the scatter points between the scatter points. Shepard's method has been used extensively because of its simplicity.

3D Interpolation The 3D equations for Shepard's method are identical to the 2D equations except that the distances are computed using:

$$h_i = \sqrt{(x-x_i)^2 + (y-y_i)^2 + (z-z_i)^2}$$

where $(x, y, z)$are the coordinates of the interpolation point and $(x_i, y_i, z_i)$are the coordinates of each scatter point.

Gradient Plane Nodal Functions

A limitation of Shepard's method is that the interpolating surface is a simple weighted average of the data values of the scatter points and is constrained to lie between the extreme values in the dataset. In other words, the surface does not infer local maxima or minima implicit in the dataset. This problem can be overcome by generalizing the basic form of the equation for Shepard's method in the following manner:

$$F(x,y)\sum_{i-1}^{n} w_i Q_i(x,y,)$$

where $Q_i$are nodal functions or individual functions defined at each scatter point (Franke 1982; Watson & Philip 1985). The value of an interpolation point is calculated as the weighted average of the values of the nodal functions at that point. The standard form of Shepard's method can be thought of as a special case where horizontal planes (constants) are used for the nodal functions. The nodal functions can be sloping planes that pass through the scatter point. The equation for the plane is as follows:

$$Q_i(x,y) = f_x(x - x_i) + f_y(y - y_i) + f_i$$

where $f_x$and $f_y$are partial derivatives at the scatter point that have been previously estimated based on the geometry of the surrounding scatter points. Gradients are estimated in WMS by first triangulating the scatter points and computing the gradient at each scatter point as the average of the gradients of each of the triangles attached to the scatter point.

The planes represented by the above equation are sometimes called "gradient planes". By averaging planes rather than constant values at each scatter point, the resulting surface infers extremities and is asymptotic to the gradient plane at the scatter point rather than forming a flat plateau at the scatter point.

3D Interpolation

The 3D equivalent of a gradient plane is a "gradient hyperplane." The equation of a gradient hyperplane is as follows:

$$Q_i(x,y,z) = f_x(x - x_i) + f_y(y - y_i) + f_z(z - z_i) + f_i$$

where f_x, f_y, and $f_z$are partial derivatives at the scatter point that are estimated based on the geometry of the surrounding scatter points. The gradients are found using a regression analysis which constrains the hyperplane to the scatter point and approximates the nearby scatter points in a least squares sense. At least five non-coplanar scatter points must be used.

Quadratic Nodal Functions

The nodal functions used in inverse distance weighted interpolation can be higher degree polynomial functions constrained to pass through the scatter point and approximate the nearby points in a least squares manner. Quadratic polynomials have been found to work well in many cases (Franke & Nielson 1980; Franke 1982). The resulting surface reproduces local variations implicit in the dataset, is smooth, and approximates the quadratic nodal functions near the scatter points. The equation used for the quadratic nodal function centered at point $k$is as follows:

$$Q_k(x,y) = a_{k1}+a_{k2}(x-x_k)+a_{k3}(y-y_k)+a_{k4}(x-x_k)^2+a_{k5}(x-x_k)(y-y_k)+a_{k6}(y-y_k)^2$$

To define the function, the six coefficients $a_{k1}..a_{k6}$must be found. Since the function is centered at the point $k$and passes through point k, we know beforehand that $a_{k1} = f_k$where $f_k$is the function value at point k. The equation simplifies to:

$$Q_k(x,y) = f_k+a_{k2}(x-x_k)+a_{k3}(y-y_k)+a_{k4}(x-x_k)^2+a_{k5}(x-x_k)(y-y_k)+a_{k6}(y-y_k)^2$$

Now there are only five unknown coefficients. The coefficients are found by fitting the quadratic to the nearest NQ scatter points using a weighted least squares approach. In order for the matrix equation used to solve for the coefficients to be stable, there should be at least five scatter points in the set.

3D Interpolation

For 3D interpolation, the equation for the quadratic nodal function is:

$$Q_k(x,y,z) = a_{k1} + a_{k2}(x-x_k) + a_{k3}(y-y_k) + a_{k4}(z-z_k)$$
$$+a_{k5}(x-x_k)(y-y_k) + a_{k6}(x-x_k)(z-z_k) + a_{k7}(y-y_k)(z-z_k)$$
$$+a_{k8}(x-x_k)^2 + a_{k9}(y-y_k)^2 + a_{k10}(z-z_k)^2$$

To define the function, the ten coefficients $a_{k1}..a_{k10}$must be found. Since the function is centered on point k, we know that $a_{k1} = f_k$where $f_k$is the data value at point k. The equation simplifies to:

$$Q_k(x,y,z) = f_x + a_{k2}(x-x_k) + a_{k3}(y-y_k) + a_{k4}(z-z_k)$$
$$+a_{k5}(x-x_k)(y-y_k) + a_{k6}(x-x_k)(z-z_k) + a_{k7}(y-y_k)(z-z_k)$$
$$+a_{k8}(x-x_k)^2 + a_{k9}(y-y_k)^2 + a_{k10}(z-z_k)^2$$

Now there are only nine unknown coefficients. The coefficients are found by fitting the quadratic to a subset of the neighboring scatter points in a weighted least squares fashion. In order for the matrix equation used to be solve for the coefficients to be stable, there should be at least ten non-coplanar scatter point in the set.

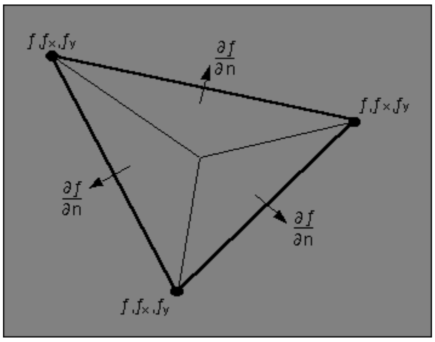
Related Topics

- Interpolation Options
- 2D Scatter Point Module

Clogh-Tocher Interpolation

The Clough-Tocher interpolation technique is often referred to in the literature as a finite element method because it has origins in the finite element method of numerical analysis. Before any points are interpolated, the scatter points are first triangulated to form a temporary TIN. A bivariate polynomial is defined over each triangle, creating a surface made up of a series of triangular Clough-Tocher surface patches.

The Clough-Tocher patch is a cubic polynomial defined by twelve parameters shown in the following figure: the function values, f, and the first derivatives, $f_x$and f_y, at each vertex, and the normal derivatives, , at the midpoint of the three edges in the triangle (Clough & Tocher, 1965; Lancaster & Salkauskas, 1986). The first derivatives at the vertices are estimated using the average slopes of the surrounding triangles. The element is partitioned into three subelements along seams defined by the centroid and the vertices of the triangle.



The Twelve Parameters Used to Define the Clough-Tocher Triangle.

A complete cubic polynomial of the form:

$$F(x,y)=\sum_{j-0}^{3-i} c_{ij}x^iy^j$$

is created over each sub-triangle with slope continuity across the seams and across the boundaries of the triangle. Second derivative continuity is not maintained across the seams of the triangle.

Since the Clough-Tocher scheme is a local scheme, it has the advantage of speed. Even very large scatter point sets can be interpolated quickly. It also tends to give a smooth interpolating surface which brings out local trends in the dataset quite accurately.

Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the Clough-Tocher interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the default extrapolation value entered at the bottom of the *Interpolation Options* dialog.

Related Topics

- Interpolation
- 2D Scatter Point Module

Natural Neighbor Interpolation

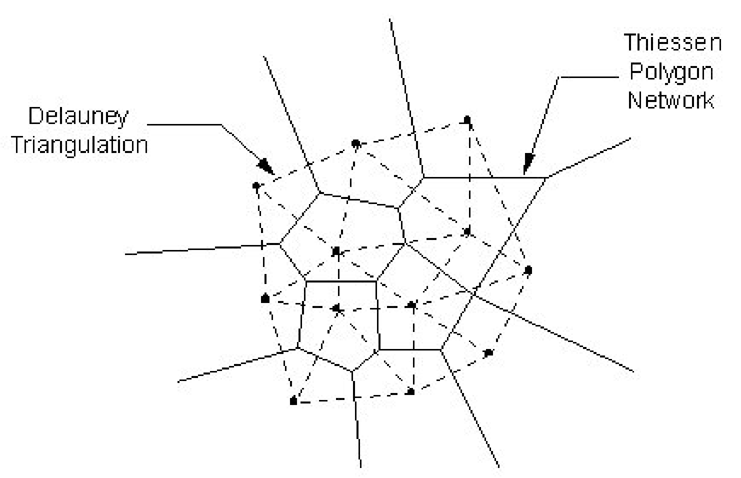
Natural neighbor interpolation is also supported in WMS. Natural neighbor interpolation has many positive features. It can be used for both interpolation and extrapolation and it generally works well with clustered scatter points. Natural neighbor interpolation was first introduced by Sibson (1981). A more detailed description of natural neighbor interpolation in multiple dimensions can be found in Owen (1992).

The basic equation used in natural neighbor interpolation is identical to the one used in IDW interpolation:

$$F(x.y)\sum_{i-1}^{n} w_i Q_i(x,y)$$

As with IDW interpolation, the nodal functions can be either constants, gradient planes, or quadratics. The nodal function can be selected using the *Natural Neighbor Interpolation Options* dialog. The difference between IDW interpolation and natural neighbor interpolation is the method used to compute the weights and the method used to select the subset of scatter points used for interpolation.

Natural neighbor interpolation is based on the Thiessen polygon network of the scatter point set. The Thiessen polygon network can be constructed from the Delauney triangulation of a scatter point set. A Delauney triangulation is a TIN that has been constructed so that the Delauney criterion has been satisfied.



Delauney Triangulation and Corresponding Thiessen Polygon Network for a Set of Scatter Points.

There is one Thiessen polygon in the network for each scatter point. The polygon encloses the area that is closer to the enclosed scatter point than any other scatter point. The polygons in the interior of the scatter point set are closed polygons and the polygons on the convex hull of the set are open polygons.

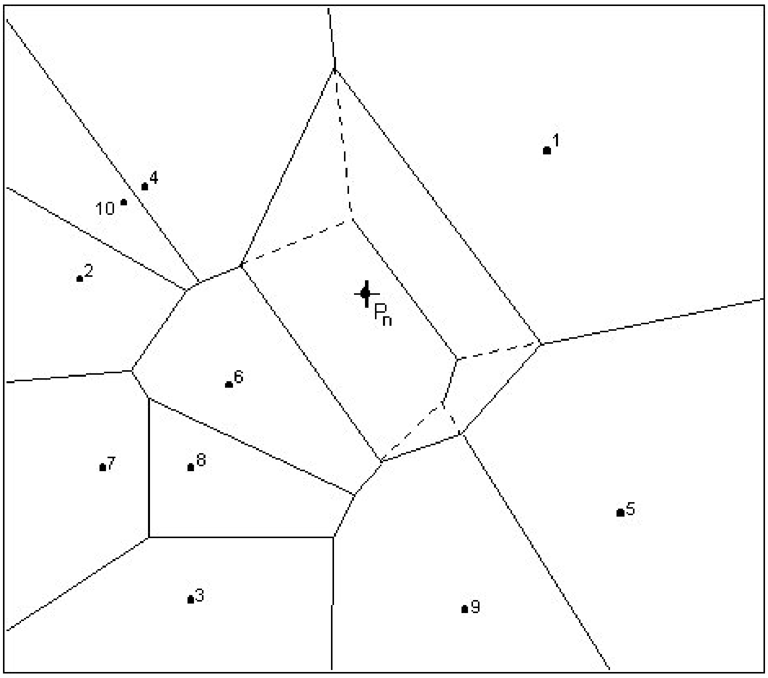
Each Thiessen polygon is constructed using the circumcircles of the triangles resulting from a Delauney triangulation of the scatter points. The vertices of the Thiessen polygons correspond to the centroids of the circumcircles of the triangles.

Local Coordinates

The weights used in natural neighbor interpolation are based on the concept of local coordinates. Local coordinates define the "neighborliness" or amount of influence any scatter point will have on the computed value at the interpolation point. This neighborliness is entirely dependent on the area of influence of the Thiessen polygons of the surrounding scatter points.

To define the local coordinates for the interpolation point, P_n, the area of all Thiessen polygons in the network must be known. Temporarily inserting $P_n$into the TIN causes the TIN and the corresponding Thiessen network to change, resulting in new Thiessen areas for the polygons in the neighborhood of P_n.

The concept of local coordinates is shown graphically in the following figure. Points 1-10 are scatter points and P_n is a point where some value associated with points 1-10 is to be interpolated. The dashed lines show the edges of the Thiessen network before $P_n$is temporarily inserted into the TIN and the solid lines show the edges of the Thiessen network after P_n is inserted.



Overlapping Thiessen Polygon Areas Used in Computation of Local Coordinates.

Only those scatter points whose Thiessen polygons have been altered by the temporary insertion of P_n are included in the subset of scatter points used to interpolate a value at P_n. In this case, only points 1, 4, 5, 6, & 9 are used. The local coordinate for each of these points with respect to P_n is defined as the area shared by the Thiessen polygon defined by point P_n and the Thiessen polygon defined by each point before point P_n is added. The greater the common area, the larger the resulting local coordinate, and the larger the influence or weight the scatter point has on the interpolated value at P_n.

If we define $k(n)$ as the Thiessen polygon area of P_n and $km(n)$ as the difference in the Thiessen polygon area of a neighboring scatter point, P_m, before and after P_n is inserted, then the local coordinate $\lambda_m(n)$ is defined as:

$$\lambda_m(n) = \frac{K_m(n)}{K(n)}$$

The local coordinate $lm(n)$ varies between zero and unity and is directly used as the weight, wm(n), in the interpolation equation. If P_n is at precisely the same location as P_m, then the Thiessen polygon areas for P_n and P_m are identical and $\lambda_m(n)$ has a value of unity. In general, the greater the relative distance P_m is from P_n, the smaller its influence on the final interpolated value.

Extrapolation

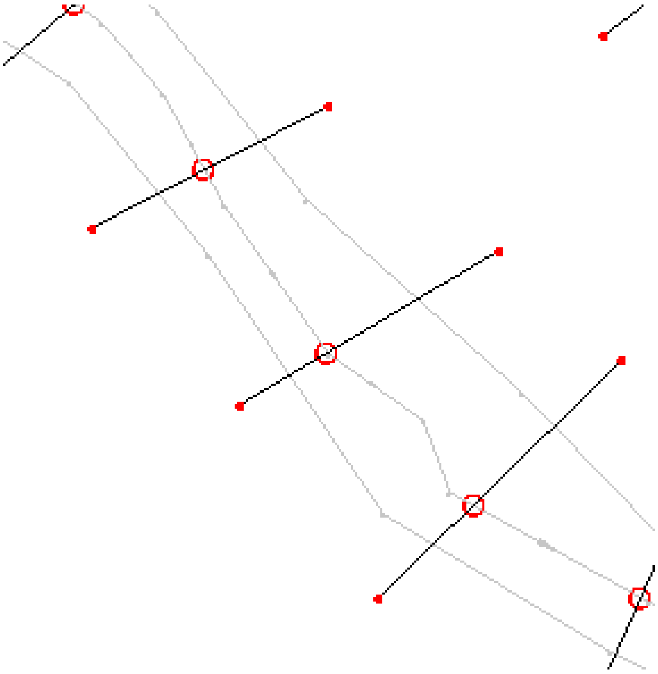
As shown in the figure above, the Thiessen polygons for scatter points on the perimeter of the TIN are open-ended polygons. Since such polygons have an infinite area, they cannot be used directly for natural neighbor interpolation. Thus, a special approach is used to facilitate extrapolation with the natural neighbor scheme. Prior to interpolation, the x and y boundaries of the object being interpolated to (grid, mesh, etc.) are determined and a box is placed around the object whose boundaries exceed the limits of the object by approximately 10% (this value can be modified by the user). Four temporary "pseudo-scatter points" are created at the four corners of the box. The inverse distance weighted interpolation scheme with gradient plane nodal functions is then used to estimate a data value at the pseudo-points. From that point on, the pseudo-points with the extrapolated values are included with the actual scatter points in the interpolation process. Consequently, all of the points being interpolated to are guaranteed to be within the convex hull of the scatter point set. Once the interpolation is complete, the pseudo-points are discarded.

Related Topics

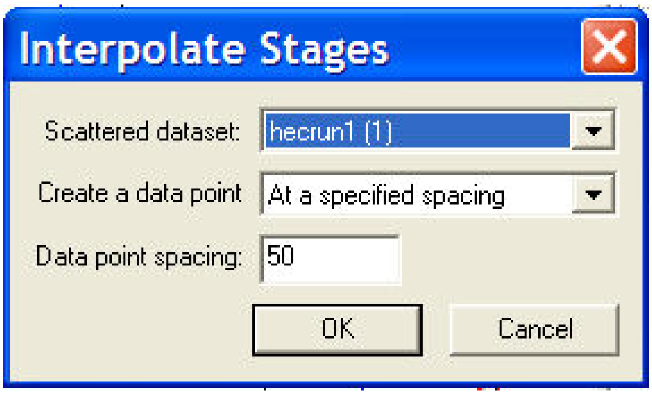
- Interpolation Options
- 2D Scatter Point Module

Interpolating Hydraulic Model Results

After running a 1D Hydraulic model like HEC-RAS, the result is a water surface elevation (or hydrograph for a dynamic solution) at the cross section as indicated by the red circles in the river section shown in the figure below.

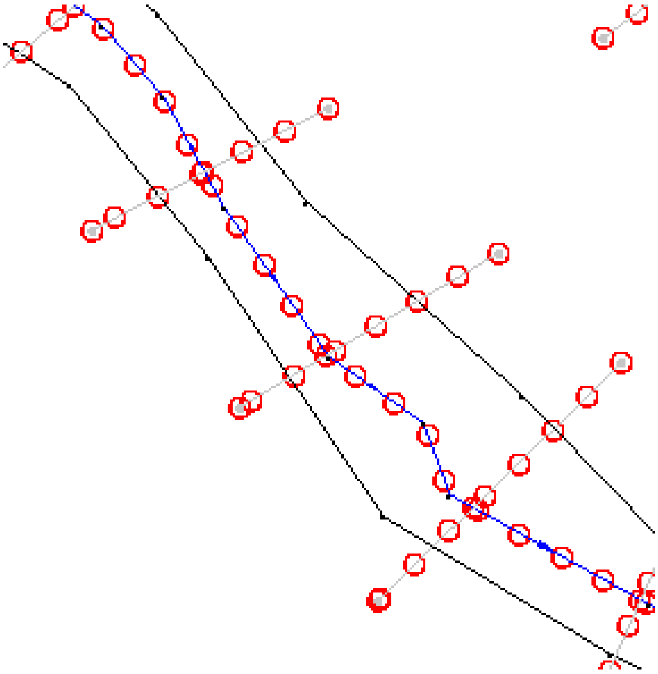


The flood plain delineation uses interpolation from the scatter set created by the hydraulic model solution, but the interpolation is not very accurate with such a sparse dataset (it often takes 6-8 points to interpolate and with such a sparse dataset these six or eight points could cover a long distance within the reach and result in a poor interpolation value). The **Interpolate Water Surface Elevations** command of the *River Tools* menu will interpolate for either the **1D-Hydraulic Centerline**, or **1D-Hydraulic Cross-sections** command new scatter points either at the vertices of the feature arcs (centerline or cross sections) or at a specified distance along the arcs.



Along cross sections the same value is used at every point along the cross section and along the centerline linearly interpolation is used between consecutive cross sections (this would be consistent with the assumption of a 1D hydraulic model that the water surface is linear between cross sections - if not more detail to the model should be added).

You specify either the centerline or the cross-sections by making it the active coverage in the Project Explorer prior to choosing this command. The following figure shows the result of interpolating along both the cross sections and the centerline of the model shown above.



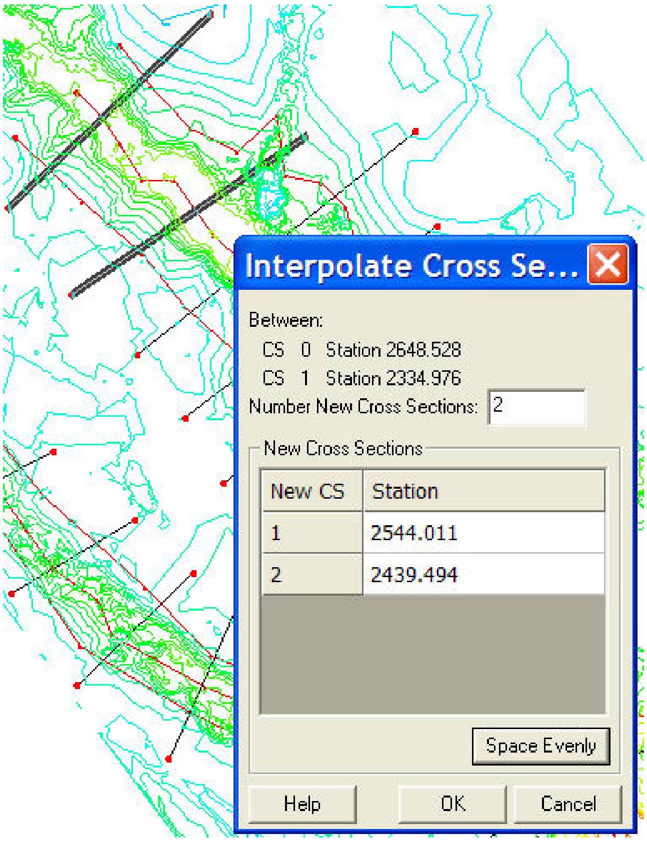
Related Topics

- Read HEC-RAS Simulation
- Flood Plain Delineation
- Preparing Stage Data

Interpolate Cross Sections

Using the **Interpolate Cross Sections** command in the *River Tools* menu you can create any number of additional cross sections between two selected cross sections. This allows you to provide more detail to the hydraulic model. The interpolated cross sections are derived from the geometries of the selected cross-sections. The stationing of the new cross sections are established by evenly distributing the length between the selected cross sections.

New Feature Arcs are created for the interpolated cross sections.



Related Topics

- Extracting Cross Sections
- Editing Cross Sections
- Interpolating Results

Computation of Interpolation Weights

When computing the interpolation weights, three options are available for determining which points are included in the subset of points used to compute the weights and perform the interpolation: subset, all points, and enclosing triangle.

Subset of Points

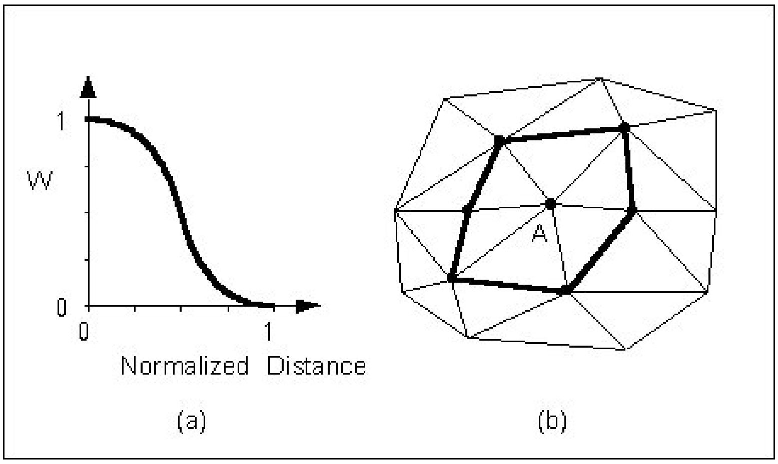
If the Use subset of points option is chosen, the Subset Definition dialog can be used to define a local subset of points.

All Points

If the Use all points option is chosen, a weight is computed for each point and all points are used in the interpolation.

Enclosing Triangle

The Use vertices of enclosing triangle method makes the interpolation process a local scheme by taking advantage of TIN topology (Franke & Nielson, 1980). With this technique, the subset of points used for interpolation consists of the three vertices of the triangle containing the interpolation point. The weight function or blending function assigned to each scatter point is a cubic S-shaped function as shown in part a of the figure below. The fact that the slope of the weight function tends to unity at its limits ensures that the slope of the interpolating surface is continuous across triangle boundaries.

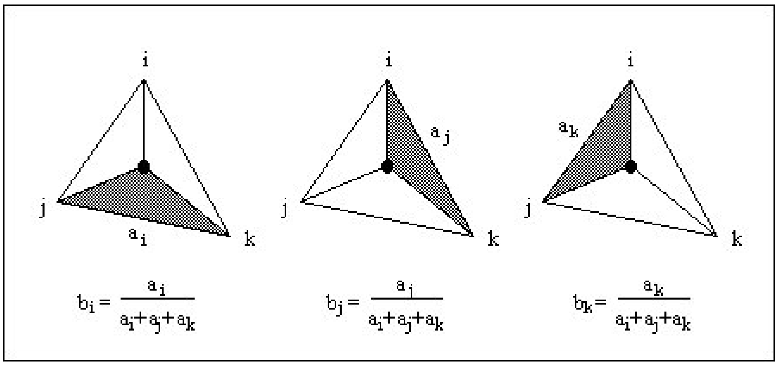


(a) S-Shaped Weight Function and (b) Delauney Point Group for Point A.

The influence of the weight function extends over the limits of the Delauney point group of the scatter point. The Delauney point group is the "natural neighbors" of the scatter point, and the perimeter of the group is made up of the outer edges of the triangles that are connected to the scatter point as shown in part b. The weight function varies from a weight of unity at the scatter point to zero at the perimeter of the group. For every interpolation point in the interior of a triangle there are three nonzero weight functions (the weight functions of the three vertices of the triangle). For a triangle T with vertices i, j, & k, the weights for each vertex are determined as follows:

$$w_i(x,y) = b_i^2(3-2b_i) + 3\frac{b_i^2 b_j b_k}{b_i b_j + b_i b_k + b_j b_k}$$
$$\left\{ b_j \left[\frac{\|e_i\|^2 + \|e_k\|^2 - \|e_j\|^2}{\|e_k\|^2} \right] + b_k \left[\frac{\|e_i\|^2 + \|e_j\|^2 - \|e_k\|^2}{\|e_j\|^2} \right] \right\}$$

where ||ei|| is the length of the edge opposite vertex i, and b_i, b_j, b_k are the area coordinates of the point (x,y) with respect to triangle T. Area coordinates are coordinates that describe the position of a point within the interior of a triangle relative to the vertices of the triangle. The coordinates are based solely on the geometry of the triangle. Area coordinates are sometimes called "barycentric coordinates." The relative magnitude of the coordinates corresponds to area ratios as shown below:



Barycentric Coordinates for a Point in a Triangle.

The xy coordinates of the interior point can be written in terms of the xy coordinates of the vertices using the area coordinates as follows:

$$x = b_i x_i + b_j x_j + b_k x_k$$
$$y = b_i y_i + b_j y_j + b_k y_k$$
$$1.0 = b_i + b_j + b_k$$

Solving the above equations for b_i, b_j, and b_k yields:

$$b_i \frac{1}{2A}[(x_j y_k - x_k y_j) + (y_j - y_k)x + (x_k - x_j)y]$$
$$b_j \frac{1}{2A}[(x_k y_i - x_i y_k) + (y_k - y_i)x + (x_i - x_k)y]$$
$$b_k \frac{1}{2A}[(x_i y_j - x_j y_i) + (y_i - y_j)x + (x_j - x_i)y]$$
$$A = \frac{1}{2}(x_i y_j + x_j y_k + x_k y_i - y_i x_j - y_j x_k - y_k x_i)$$

Using the weight functions defined above, the interpolating surface at points inside a triangle is computed as:

$$F(x,y) = w_i(x,y)Q_i(x,y) + w_j(x,y)Q_j(x,y) + w_k(x,y)Q_k(x,y)$$

where w_i, w_j, and w_k are the weight functions and Q_i, Q_j, and Q_k are the nodal functions for the three vertices of the triangle.

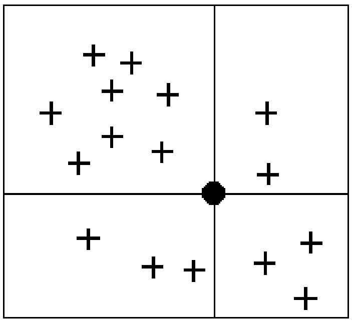
Related Topics

- Inverse Distance Weighted Interpolation
- Subset Definition
- Shepard's Method
- Gradient Plane Nodal Functions
- Quadratic Nodal Functions

Subset Definition

In the IDW Interpolation Options dialog, an option is available for using a subset of the scatter points (as opposed to all of the available scatter points) in the computation of the nodal function coefficients and in the computation of the interpolation weights. Using a subset of the scatter points drops distant points from consideration since they are unlikely to have a large influence on the nodal function or on the interpolation weights. In addition, using a subset can speed up the computations since less points are involved.

If the *Use subset of points* option is chosen, the **Subsets** button can be used to bring up the *Subset Definition* dialog. Two options are available for defining which points are included in the subset. In one case, only the nearest N points are used. In the other case, only the nearest N points in each quadrant are used as shown below. This approach may give better results if the scatter points tend to be clustered.



The Four Quadrants Surrounding an Interpolation Point.

If a subset of the scatter point set is being used for interpolation, a scheme must be used to find the nearest N points. Two methods for finding a subset are provided in the *Subset Definition* dialog: the global method and the local method.

Global Method

With the global method, each of the scatter points in the set are searched for each interpolation point to determine which N points are nearest the interpolation point. This technique is fast for small scatter point sets but may be slow for large sets.

Local Method

With the local methods, the scatter points are triangulated to form a temporary TIN before the interpolation process begins. To compute the nearest N points, the triangle containing the interpolation point is found and the triangle topology is then used to sweep out from the interpolation point in a systematic fashion until the N nearest points are found. The local scheme is typically much faster than the global scheme for large scatter point sets.

Related Topics

- Inverse Distance Weighted Interpolation
- Computation of Interpolation Weights

4.2 Equations

Assigning an Equation

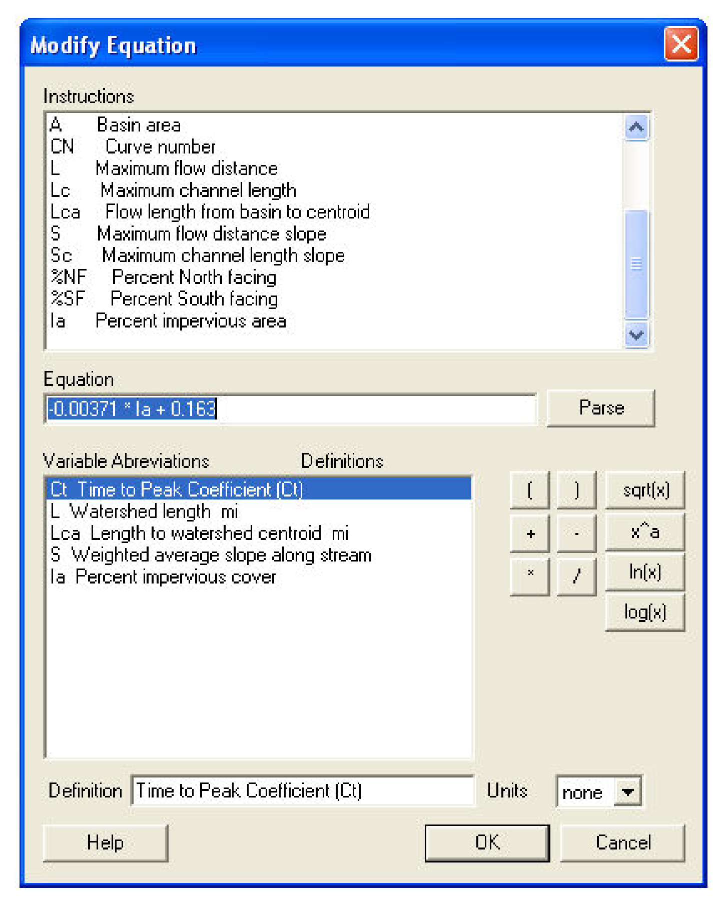
The *Basin Time Computation* dialog allows you to select from one of several pre-defined equations. The Equation drop-down box lists the available methods for computing lag time or time concentration and is controlled by setting the computation type combo box to the desired setting. Once you have selected an equation, the *Instructions / Results* window will identify any variables with zero values and instruct you to define values for them. Variables are assigned values by selecting the variable from the Variables text window and then assigning a value to it in the edit box to the right. Once all variables are defined the *Instructions / Results* window will display the computed lag time or time of concentration.

Related Topics

- Overview of Basin Data Equations
- Travel Times from Basin Data

Customizing Equations

Any of the equations can be edited (i.e. you can pick an equation and then modify it slightly to meet your needs) or your own equation created using the **Modify Equation** or **User Defined** buttons. Both of these buttons bring up the *Modify Equation* dialog shown below.



If you enter this dialog using the **Modify Equation** button the currently selected equation will appear in the Equation edit box. If the current equation has more than one equation (some equations have separate equations for lag time/time of concentration and peaking or storage coefficients) then you will be asked to choose which equation you want to modify. If you use the **User Defined** button to enter this dialog then the *Equation edit* box will be blank. You can type a new equation or modify an existing equation from the keyboard or by using the buttons for add, subtract, multiply, divide, etc. You may also enter variable names corresponding to the basin data computed by WMS, or enter variables that you wish to define yourself. The variable definition is displayed and edited in the Definition edit box and units associated with the variable can be set using the Units drop-down combo box.

When you have finished typing the equation you should select the **Parse** button. Doing so will result in any variables being identified and displayed in the Variables text window along with their definitions. The rules of precedence are typical for mathematical computations (order of operations) and are as follows:

1. Parenthetical statements
2. Functions like ln, log, etc.
3. The power function (^)
4. Multiply and divide
5. Add and subtract
6. Left to right

Related Topics

- Travel Times from Basin Data
- Assigning an Equation
- Editing Basin Variables
- Variables Computed by WMS

Repeated Use of an Equation

If your local hydrology manual requires the use of an equation not currently supported by WMS and you must repeatedly use this equation for projects, you can set up the equation in a text file in the same directory as WMS and it will be read in and set up as the User Defined equation automatically. To do this you must create a file named "wmstc.equ" and define one or more equation groups (a group may be used because sometimes your equation to compute a travel time may be a function of another equation). The following is an example wmstc.equ file.

```
EQGROUP Myeq1

EQ 1.49 / n * R * Sc DEF Manning's equation UNITS laghour

VAR n DEF Manning's roughness coefficient VAL .002 UNITS none WHICH -1

VAR R DEF Hydraulic Radius VAL 0.0 UNITS ft WHICH 2

EQ XA / Pw DEF Hydraulic Radius UNITS ft

VAR XA DEF Cross section area VAL 0.0 UNITS sqft WHICH -1

VAR Pw DEF Wetted Perimeter VAL 0.0 UNITS ft WHICH -1

EQGROUP Myequ2

EQ Lc / V DEF Time of travel UNITS lagmin

VAR V DEF Velocity VAL 2.5 UNITS none WHICH -1
```

The following describes the keywords used in the file as well as the possible values that can be entered following each keyword.

Each equation group begins with the **EQGROUP** card. The rest of the line is interpreted as a character string and used as the name of the equation group in the drop-down combo box allowing you to select an equation.

Each equation is defined using the following three cards with their associated inputs:

EQ – The equation

DEF – A character string definition for the equation

UNITS – Units of the computed result. Possible values for units include: laghour, lagmin, tchour, tcmin, clarkr, tp. The first four identify whether the equation is intended for use in computing lag time or time of concentration (a conversion to the other is done using this equation) and whether the result is in minutes or hours. The last three are used if the equation is used to compute the Clark storage coefficient or one of the peaking parameters used in HEC-1.

If one of the WMS recognized variables are used (A, Lca, etc.) then you do not need to include a VAR record as it will automatically map the WMS variable for use in the equation. For variables not recognized as something WMS computes you should include a VAR line that uses the following keywords:

VAR – Name of the variable

DEF – Variable definition or description

VAL – The initial default value (should be 0.0 if this will be entered separately for each basin)

UNITS – The units of the variable. The following list of keywords are recognized for units. If you use a variable with units not in this list you should enter "none" and then make sure that the equation is dimensionally consistent with the units defined for the equation on the EQ line.

laghour	
tcmin	
tchour	
lagmin	
mi	mile
m	meter
ft	feet
km	kilometer
hr	hour
sec	second
none	none
acre	acre
sqkm	square kilometer
sqmi	square mile
sqft	square feet
sqm	square meter
in	inch
mm	millimeter
hect	hectare

WHICH – If a variable represents another equation that is part of the equation group then you should follow the WHICH keyword with the equation number. If it does not represent another equation then you should follow it with a -1. See the example above where the hydraulic radius variable R represents the second equation in the group (XA/Pw).

You may define as many equations as you want in the file and each time WMS is started the equations will be read and become members in the list of equations you can select from when defining travel times from basin data.

Related Topics

- Customizing Equations
- Travel Times from Basin Data

Variables Computed by WMS

A complete list of variables computed for drainage basins that are available for use in defining equations, along with the acronyms used in WMS is given below:

A= The area of the basin in the units specified prior to computing basin parameters.

BS= The average basin slope, or average slope of the triangles comprising this basin. A triangle's slope is computed as the change in elevation divided by the change in XY or plan distance.

$AOFD$= The average overland flow distance within the basin. This is computed by averaging the overland distance traveled from the centroid of each triangle to the nearest stream.

$\%NF$= The percentage of the basin whose aspect is directed North where North is defined as the positive Y direction.

$\%SF$= The percentage of the basin whose aspect is directed South where South is defined as the negative Y direction.

L= The basin length.

P= The perimeter of the basin.

$Shape$= The shape factor of the basin, or the length divided by the width.

Sin= The sinuosity factor of the stream in the basin. Defined by dividing the maximum stream length in the basin by the length.

$AVEL$= The mean basin elevation.

MFD= The maximum flow distance within a basin including both overland and channel flow.

$MFDS$= The slope of the MFD.

$CTOMFD$= The distance from the centroid of the basin to the nearest point associated with the MFD.

CSD= The distance from the centroid of the basin to a point in the stream which is a part of the MFD. The CSD differs from the CTOMFD in that it is only concerned with the channel or stream flow portion of the MFD, whereas the CTOMFD also incorporates the portion of the MFD which is overland flow.

CSS= The slope of the CSD.

MSL= The maximum stream length within the basin. This is computed by determining the maximum distance traveled when "flowing" down from the top of streams in a basin and where the streams exit the basin.

MSS= The slope of the MSL.

In addition to the basin attributes defined above the following stream attributes are computed:

L= Stream lengths for each segment.

SS= Stream slopes for each segment.

Related Topics

- Computing Basin Data from DEMs
- Computing Basin Data from TINs
- Travel Times from Basin Data

4.2.a. Basin Data Equations

Overview of Basin Data Equations

Dodson (Dodson & Associates, 1992) compiled several equations from hydrologic publications. WMS has implemented many of these equations and allows you to choose from the ones listed below to automatically compute lag times/times of concentration. Because most of the equations were developed for specific watersheds (e.g. size, land cover etc.) you should consider the assumptions made about a given equation, and try to identify one that used watershed conditions similar to the one you are studying. The following is a list of the equations available in WMS. The SCS also found that for many cases the lag time could be related to the concentration time by the following equation:

$$T_{LAG} = 0.6 * T_c$$

This relationship is always used by WMS to determine lag time when a method for computing time of concentration is chosen, or to compute time of concentration when a method for lag time is chosen. Methods used for lag time begin with " $T_{LAG} =$" whereas methods used to compute time of concentration begin with " $t_c =$".

Lag Time Based Equations:

Colorado State

Customized

Denver

Eagleson

Espey

Putnam

Riverside County

SCS

Taylor Schwartz

Tulsa District

Time of Concentration Based Equations:

Fort Bend

Kerby

Kirpich

Ramser

Related Topics

- Travel Times from Basin Data
- Travel Times from Map Data
- Assigning an Equation

Travel Times from Basin Data

WMS computes many geometric parameters when using the **Compute Basin Data** command for either a TIN or DEM. These parameters form the basis of the empirical equations used to compute lag time and time of concentration. The **Basin Time Computation** dialog allows you to select from a series of pre-defined equations (Dodson & Associates, 1992), or create your own equation using any of computed basin parameters to compute time of concentration or lag time. If the equation is a function of variable not computed by WMS then you will have to enter the value in this dialog before a travel time can be computed. The process of computing travel times for a selected basin from computed basin data involves the following two steps:

1. Select an equation (or create a new one).
2. Follow the instructions provided in the Instructions / Results text window until all of the necessary data are correctly defined (you may also edit any of the variables computed by WMS).

Auto Recomputing

By default once an equation is specified for a basin, the lag time and time of concentration will be computed automatically each time that basin data are computed or when the curve number changes. If you do not wish to have the equation updated when basin data changes then you should turn on the **Do not Auto-recompute parameters** check box for the selected basin.

Exporting Results to a Report

The **Export Data** and **Copy To Clipboard** buttons are used to create a text report that summarizes the equation, variables, and computed time of concentration or lag time for the basin. Exporting the data will create a text file and allows you to either append to an existing file (so that a single report for multiple basins can be created) or create a new file. Copying to the clipboard places the report text on the Windows clipboard so that it is available for pasting into other documents.

Related Topics

- Assigning Equations to a Basin
- Updating a travel time when basin parameters change
- Customizing Equations
- Exporting Results
- Geometric Parameters Used by Equations
- Travel Times from Map Data

Colorado State Lag Time Equation

An equation used to compute lag time for watersheds in the Denver Colorado area was developed at Colorado State University. This equation was primarily used for watersheds in which there was some amount of developed land. It is not valid for watersheds with less than 10 percent impervious area (Ia). The equation uses a runoff coefficient which represents variations in topography. This coefficient can also be used to compute the peaking factor using the relationship shown in the chart below.

$$C_t = \frac{7.81}{I_a^{0.78}}$$

$$T_{LAG} = C_t(L * L_{ca})^{0.3}$$

where:

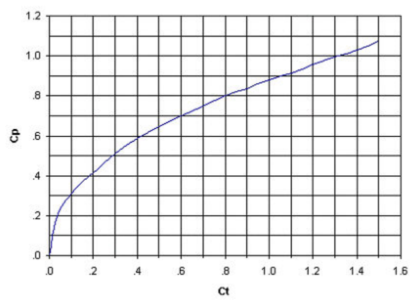
C_t= coefficient of watershed topography based on impervious area.

I_a= percentage of impervious area in the watershed (must be defined in the HEC-1 Loss methods).

T_{LAG}= watershed lag time in hours.

L= maximum flow path length in miles.

L_{ca}= length to the centroid in miles.



Graph for Determining the Peaking Coefficient Cp from Ct.

Related Topics

- Overview of Basin Data Equations

Customized Lag Time Equation

Almost all of the lag time equations are of the form:

$$T_{LAG} = C_t * \left(\frac{(L * L_{ca})}{\sqrt{S}} \right)^m$$

where:

T_{LAG}= watershed lag time in hours.

C_t= coefficient accounting for differences in watershed slope and storage.

L= the maximum flow length of the watershed along the main channel from the point of reference to the upstream boundary of the watershed, in miles.

L_{ca}= the distance along the main channel from the point of reference to a point opposite the centroid, in miles.

S= slope of the maximum flow distance path in ft/mile.

m= lag exponent

Therefore, if the equation that your state, county, etc. uses to compute lag time is not available, it can often be set up using this equation by entering your own coefficient, C_t, and exponent, m.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Denver Lag Time Equation

The Denver Lag Time Equation was developed by the Denver Area Urban Drainage and Flood Control District (Wright-McLaughlin Engineers, 1975)

$$T_{LAG} = C_t * \left(\frac{L * L_{ca}}{\sqrt{S}} \right)^{0.48}$$

where:

T_{LAG}= watershed lag time in hours.

C_t= time to peak coefficient.

L= length along the stream from the study point to the upstream limits of the basin in miles.

L_{ca}= length along the stream from the study point to a point along the stream adjacent to the centroid of the basin in miles.

S= weighted average slope of the basin from the study point to the upstream limits of the basin in feet per foot.

The percent impervious (Ia) must already be defined in one of the Loss methods used for HEC-1.

This equation was developed for small urban watersheds (less than 5 square miles) with mild slopes. The peaking coefficient can be computed from the percent impervious (I_a) using the following equations:

$$C_t = -0.00371\ I_a + 0.146$$

$$C_t = 0.000023\ I_a^2 - 0.002241\ I_a + 0.146$$

$$C_t = 0.0000033\ I_a^2 - 0.000801\ I_a + 0.12$$

The Denver method used a peaking parameter $P$and the relationships below to compute the peaking coefficient Cp .

$$P = 0.002450\, I_a^2 - 0.0120\, I_a + 2.16$$
$$P = -0.00091\, I_a^2 + 0.228\, I_a - 2.06$$
$$C_p = P * C_t * A^{0.15}$$

where:

C_t= coefficient as defined by the previous three equations.

P= peaking parameter.

A= basin area in square miles.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Eagleson Lag Time Equation

Eagleson's method (1962) for computing lag time in completely storm-sewered watersheds is given as follows:

$$T_{LAG} = 0.32\left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.39}$$

where:

T_{LAG}= watershed lag time in hours.

L= maximum flow length in miles.

L_{ca}= length to the centroid in miles.

S= weighted average slope of the maximum flow path in ft/mile.

The typical characteristics of watersheds for which the Eagleson method was applied include the following:

- Areas from 0.22 to 7.5 square miles.
- $L$from 1 to 7 miles.
- $L_{ca}$from 0.3 to 3 miles.
- $S$from 6 to 20 ft/mile.
- Impervious cover from 33 to 83 percent.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Espey Lag Time Equations

Espey's equations for Snyder's parameters were developed for a series of small watersheds in Texas, Oklahoma, and New Mexico. Rather than defining the lag time, Espey (1966) used the time to rise. The difference is that the lag time is the time from the centroid of rainfall to the peak of the hydrograph, whereas the time to rise is the time from the beginning of effective rainfall to the peak of the hydrograph. The lag time can be computed by subtracting one-half the computation time interval from the time rise. Equations to compute Tr and Cp are given below:

$$T_r = 2.65\, L_f^{0.12} S_f^{-0.52}$$

Rural Areas

$$T_r = 20.8\, U\, L_f^{0.29} S_f^{-0.11} I_a^{-0.61}$$

Urban Areas

where:

T_r= time from the beginning of effective rainfall to the peak of the unit hydrograph.

L_f= stream length in feet.

S_f= stream slope in feet per foot.

I_a= percent impervious cover.

U= urbanization factor equal to 0.6 for extensive urbanization, 0.8 for some storm sewers, and 1.0 for natural conditions.

Typical conditions for typical rural watersheds include:

- $L_f$from 3,250 feet to 25,300 feet.
- $S_f$from 0.008 ft/ft to 0.015 ft/ft.
- $T_r$from 30 to 150 minutes.
- Areas from 0.1 sq. miles to 7 sq. miles.

Typical conditions for typical urban watersheds include:

- $L_f$from 200 feet to 54,800 feet.
- $S_f$from 0.0064 ft/ft to 0.104 ft/ft.
- $I_a$from 25 to 40 percent.
- $T_r$from 30 to 720 minutes.
- Areas from 0.0125 sq. miles to 92 square miles.

Espey developed the following equations to compute Snyders peaking coefficient.

$$q_p = 1700 A^{-0.12} T_r^{-0.30}$$

Rural Areas

$$q_p = 19300 A^{-0.09} T_r^{-0.94}$$

Urban Areas

Once $q_p$is computed, the peaking coefficient can be determined using the following relationships:

$$T_{LAG} = T_r - \Delta t/2$$

$$C_p = \frac{q_p * T_{LAG}}{640}$$

where $\Delta t$is the computational time interval as define in the *HEC-1 Job Control* dialog (by default it is 15 minutes).

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Putnam Lag Time Equation

Putnam (1972) developed a lag equation for watersheds around the Wichita, Kansas area as follows:

$$T_{LAG} = 0.49\sqrt{\frac{L}{\sqrt{S}}I_a^{-0.57}}$$

where:

T_{LAG}= lag time in hours.

L= maximum flow length in miles.

S= weighted slope along the maximum flow path in ft/mile.

I_a= Impervious cover as a fraction.

This equation was used for watersheds ranging in size from 0.3 to 150 sq. miles, impervious covers less than 0.3 and a ratio of between 1.0 and 9.0.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Riverside County Lag Time Equation

The Riverside County Flood Control and Water Conservancy District developed three different lag equations corresponding to mountainous, foothill, and valley areas near Riverside County, California (Anonymous, 1963).

$$T_{LAG} = 1.20\left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.38}$$

(Mountainous) 15.19

$$T_{LAG} = 0.72\left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.38}$$

(Foothills) 15.20

$$T_{LAG} = 0.38\left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.38}$$

(Valleys) 15.21

where:

T_{LAG}= lag time in hours.

L= maximum flow length in miles.

L_{ca}= length to the centroid in miles.

S= weighted slope along the maximum flow path length in ft/mile.

The typical characteristics of watersheds for which the Riverside County equations were used include the following:

- Areas from 2 to 650 square miles.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

SCS Lag Time Equation

Perhaps the most commonly used equation for lag time is the SCS equation (1972) as shown. This equation may be used when computing the unit hydrograph using Snyder's method and any of the preceding equations for lag time may also be used when computing the unit hydrograph using the SCS method. Also remember that the SCS used the relationship relating lag time and time of concentration (Overview of Basin Data Equation) to compute lag time (and the other way around) from any of the time of concentration equations which follow.

$$T_{LAG} = L^{0.8} \frac{(S+1)^{0.7}}{1900\sqrt{Y}}$$

where:

T_{LAG}= lag time in hours.

L= hydraulic length of watershed in feet.

S= maximum retention in the watershed in inches as defined by:

$$S = \frac{1000}{CN} - 10$$

Y= watershed slope in percent.

CN= SCS curve number for the watershed as defined by the loss method.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Taylor Schwartz Lag Time Equation

Taylor and Schwartz (1952) developed an equation for estimating Snyder unit hydrograph parameters that was used for 20 different watersheds in the northeastern region of the U.S. Their equations are as follows:

$$C_t = \frac{0.6}{\sqrt{S}}$$

$$T_{LAG} = C_t * (L * L_{CA})^{0.3}$$

where:

C_t= coefficient of watershed topography based on watershed slope.

S= weighted slope of maximum flow path in ft/ft.

T_{LAG}= watershed lag time in hours.

L= maximum flow length in miles.

L_{ca}= length to the centroid in miles.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Tulsa District Lag Time Equation

The Tulsa District of the US Army Corps of Engineers has developed the following family of equations for computing Snyders lag time:

$$T_{LAG} = C_t * \left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.39}$$

where:

T_{LAG}= watershed lag time in hours.

C_t= 1.42 for natural watersheds in rural areas of central and northeastern Oklahoma.

C_t= 0.92 for the same type areas that are 50% urbanized.

C_t= 0.59 for the same type areas that are 100% urbanized.

L= watershed maximum flow distance length in miles.

S= slope of the maximum flow distance path in ft/mile.

L_{ca}= length to centroid.

The range of watershed characteristics for which these equations apply include:

- Sizes ranged from 0.5 to just over 500 square miles.
- Slopes ranged from 4 to 90 feet per mile.
- Lengths ranged from 1 to 80 miles.
- Length to centroid ranged from 1 to 60 miles.

In addition to developing an equation for lag time, the Tulsa district developed the following relationship for the peak flow rate which can be used in the second equation to solve for Snyder's peaking coefficient.

$$q_p = 380 * {T_{LAG}}^{-0.92}$$

$$C_p = \frac{q_p * T_{LAG}}{640}$$

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Fort Bend Tc Equation

The county of Fort Bend, Texas (Espey, Huston, & Associates, 1987) used the equation shown in the equation below to compute tc. In addition to defining an equation for time of concentration to be used in the Clark unit hydrograph method, they also defined a relationship for the Clark watershed storage coefficient that is given by the equation following the tc equation.

$$t_c = 48.64\frac{\left(\frac{L}{\sqrt{S}}\right)^{0.57}}{{S_o}^{0.11}10^I}\log S_o$$

$$R = 128\frac{\left(\frac{L}{\sqrt{S}}\right)^{0.57}}{{S_o}^{0.11}10^I}\log S_o - t_c$$

where:

t_c= Clark time of concentration in hours.

R= Clark watershed storage coefficient.

L= length of longest flow path within the watershed in miles.

S= average slope along the longest flow path.

S_o= average basin slope.

I= percent impervious as a fraction (decimal).

Typical characteristics of the watersheds for which these equations were applied are:

- Area between 0.13 and 400 square miles.
- Length of longest flow path between 0.5 and 55 miles.
- Slope of longest flow path from 2 ft/mi. to 33 ft/mi.
- Slope of basin from 3 to 80 ft/mi.
- Impervious area from 0 to 100%.

Others have simply used the simple relationship defined by the equation below to compute the Clark watershed storage coefficient from the time of concentration.

$$R = 2t_c$$

Russell, Keening and Sunnell in their study of watersheds around Vancouver, British Columbia found the $R = c * TC$, where the calibration parameter $c$for rural watersheds ranged from 1.5 to 2.8. ("Estimating Design Flows for Urban Drainage." Russell, S. Kenning, B. and Sunnell, G. ASCE Journal of the Hydraulics Division, Vol 105, No. HY1, pp 49, January 1979.)

For coastal watersheds in Southern California the USACE proposes that "the storage coefficient Requals 0.8 time the time of concentration, TC." ("Generalized Standard Project Rainflood Criteria, Southern California Coastal Streams." Hydrologic Engineering Center. Sacramento: U.S. Army Corps of Engineers, pp 6, 1967.)

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Kerby Tc Equation

Kerby (1959) developed an equation for computing the time of concentration for overland flow distances of less than 500 feet and greater than 300 feet.

$$t_c = \left(\frac{0.67 * n * L_o}{\sqrt{S}}\right)^{0.467}$$

where:

t_c= time of concentration in minutes.

S= overland slope in ft/ft.

n= roughness coefficient.

L_o= length of overland flow in feet.

A table of recommended values for $n$is given below:

Recommended surface roughness values

Surface Description	n
Smooth, impervious surface	0.02
Smooth, packed bare soil	0.1
Poor grass, cultivated row crops of moderately rough bare soil	0.2
Pasture or average grass	0.4
Deciduous timberland	0.6
Timberland with deep forest litter or dense grass	0.8

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Kirpich Tc Equation

Kirpich's equation (1940) was developed for small, agricultural watersheds. It was derived by examining the required time for the stream to rise from low to maximum stage during a storm. The time of concentration was then assumed equal to that time.

$$t_c = \frac{0.00013L^{0.77}}{S^{0.385}}$$

where:

t_c= time of concentration in hours.

L= length of the overland flow in feet.

S= average overland slope in ft/ft.

This equation given below was developed for overland flow on bare earth. For overland flow on grassy earth, tc should be multiplied by 2.0. On concrete and asphalt surface it should be multiplied by 0.4. An adjustment is made for watersheds with a CN number less than 80 using the following equation:

$$t_c = t_c * (1 + (80 - CN) * 0.04)$$

The CN value must be defined for the given model (HEC-1, TR-20, etc.), otherwise a default CN of 50 is used.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

Ramser Tc Equation

Ramser (1927) developed an equation for computing the time of concentration in well-defined channels. The equation is based on the length and slope of the channel.

$$t_c = 0.008L_c^{0.77}S_c^{-0.385}$$

where:

t_c= time of concentration in minutes.

L_c= length of channel reach in feet.

S_c= average channel slope in ft/ft.

For flow in concrete channels, t_c should be multiplied by 0.2.

Related Topics

- Computing Lag Time or Time of Concentration from Basin Data

5. Modules

Modules

The interface for WMS is divided into eight modules. A module is provided for each of the basic data types or modeling environments supported by WMS. As you switch from one module to another module, the Tool Palette and the menus change. This allows you to focus only on the tools and commands related to the data type you wish to use in the modeling process. Switching from one module to another can be done instantaneously to facilitate the simultaneous use of several data types when necessary. Modules are also changed depending on the selected data folder or object in the Project Explorer.

The following modules are supported in WMS:

- Terrain Data Module
- Drainage Module
- Map Module
- Hydrologic Modeling Module
- Hydraulic Modeling Introduction
- GIS Module
- 2D Grid Module
- 2D Scatter Point Module

Related Topics

- Module Palette

5.1. Digital Elevation Model (DEMs)

DEM Guidelines

A digital elevation model (DEM, as defined in WMS) is simply a two-dimensional array of elevation points with a constant x and y spacing. While a DEM results in data redundancy for surface definition, its simple data structure and wide-spread availability have made them a popular source for digital terrain modeling and watershed characterization. Several researchers, including Puecker and Douglas (1975) and Garbrecht and Martz (1995) have developed methods to extract watershed geomorphology from DEMs. WMS includes many of these same tools.

The most common form of DEM elevations are the USGS digital maps. DEMs can be downloaded free of charge from the EROS home page at http://edcwww.cr.usgs.gov/doc/edchome/ndcdb/ndcdb.html [1]. Other sources of elevation data may include federal, state, and local government agencies, universities, or private data publishers. WMS can read digital elevation in standard USGS (the older single file format or the new SDTS formatted files), ARC/INFO®/ArcView® ASCII grid, DTED, and GRASS grid formats. Flow directions and flow accumulations for DEM points are computed using TOPAZ (**Compute TOPAZ Flow Data** command in the **Drainage** menu). This program uses an eight point pour technique to determine the direction of flow. This technique specifies that the flow will be directed toward the neighboring (in a structured grid there are eight neighbors for each point) DEM point with the lowest elevation. The algorithms typically include functionality for eliminating pits and resolving ambiguities when the lowest elevation is shared by more than one neighboring point.

DEMs can be used to delineate watershed and sub-basin boundaries which can subsequently be converted to a series of arcs and polygons. DEMs can also be converted to TINs and used to develop hydraulic models. Additionally, DEMs can be contoured and displayed in oblique view. When creating a TIN, they can be used as a background elevation map.

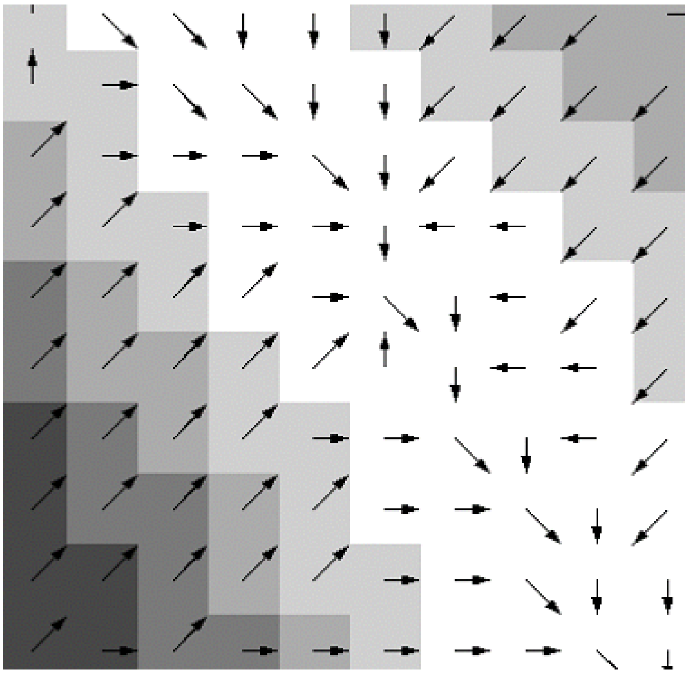
The typical steps for using DEMs to develop hydrologic models are:

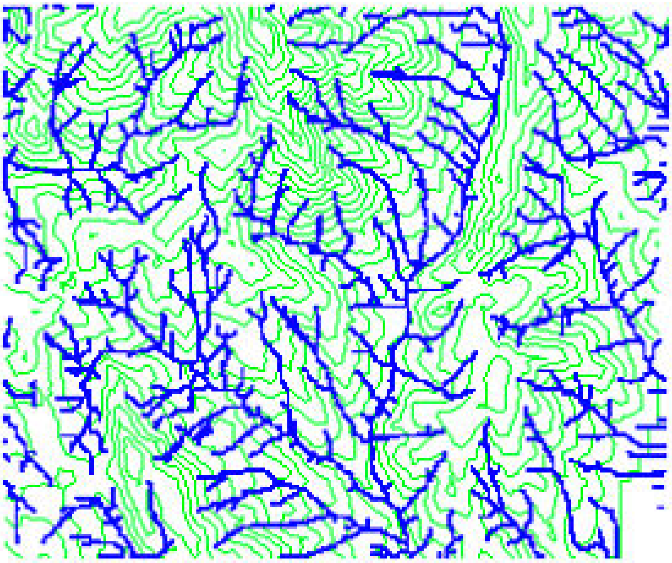
1. Obtain and Import a Digital Elevation Model (DEM)

As mentioned above, USGS DEMs can be downloaded from the Internet or obtained from government agencies, universities, or private vendors. The **File | Open** command can be used to import the DEM from one of the supported formats. The figure below shows a contoured DEM after it has been imported.

2. Compute Flow Directions and Flow Accumulations

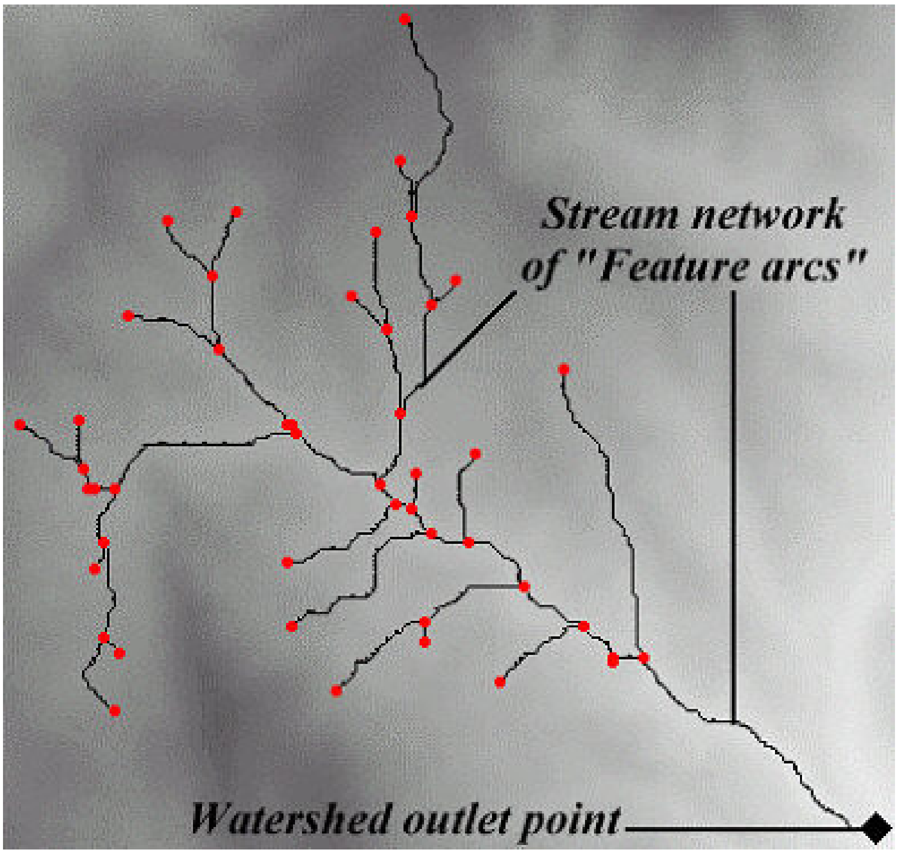
Flow directions are computed from the active DEM region using a custom version of the TOPAZ model distributed with WMS. Once flow directions are assigned for each DEM point, TOPAZ then computes flow accumulations at each DEM point as well. The flow accumulation for a given DEM point is defined as the number of DEM points whose flow paths eventually pass through that point. For example, DEM points that are part of a stream have high flow accumulation values since the flow paths of all "upstream" points will pass through them. Streams are easily identified by displaying all DEM points with a flow accumulation value greater than a user-defined threshold as shown below. Flow accumulations can be computed in WMS from the flow directions, or read from ARC/INFO®, GRASS, or TOPAZ formatted files.





3. Identify the Watershed Outlet and Convert DEM Streams to Arcs

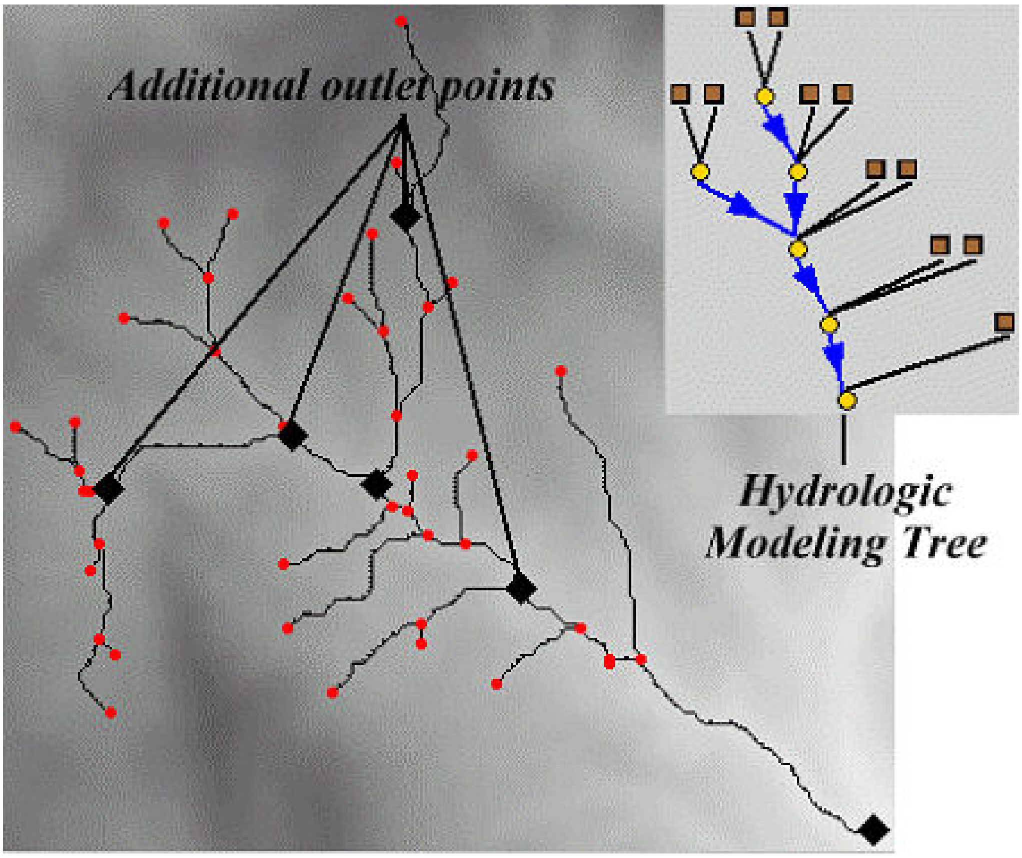
With the aid of the flow accumulations, the location of the watershed outlet needs to be determined and an outlet feature point created there. A minimum threshold is then defined and all of the DEM points “upstream” from the defined outlet(s) are connected together to form a stream network of feature arcs.



You should note that the stream feature arcs can be created in any fashion. For example, in an urban area the streams will not likely be well-defined from the DEM elevations and flow directions. The flow directions for the DEM are then used for basic overland flow whereas the stream vectors are used for conveyance channels. Practically, you can think of WMS modifying the flow directions of the DEM points underlying the stream vectors so that flow always follows the defined stream vectors.

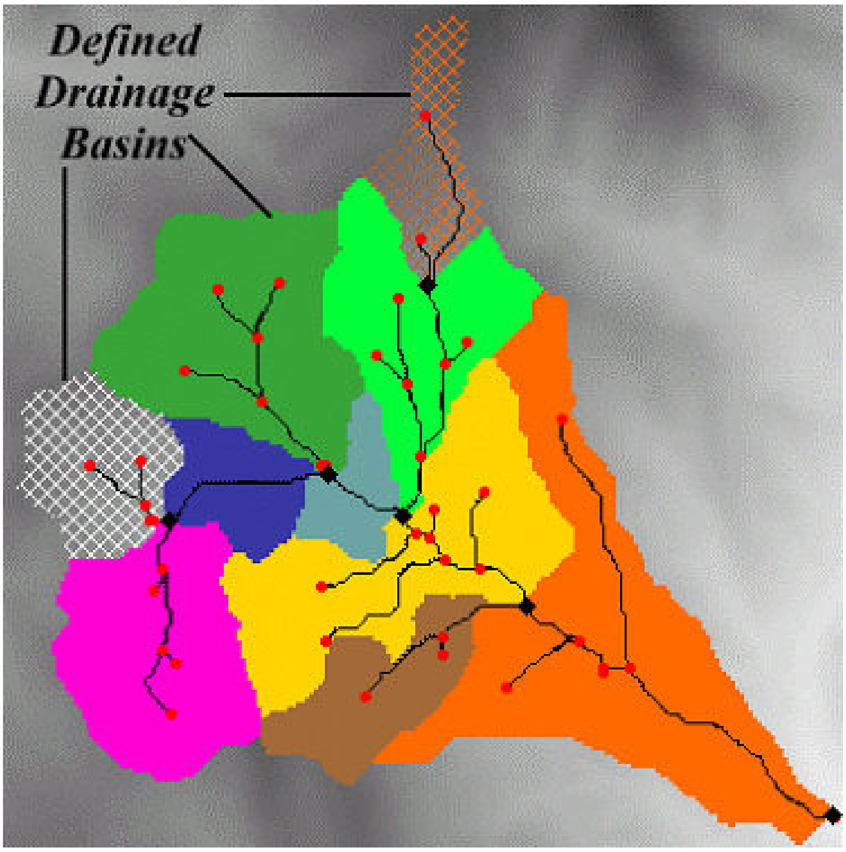
4. Define Interior Sub-basin Outlet Points

If you wish to further subdivide the watershed into sub-basins then nodes along the stream feature arcs should be converted to "outlet" nodes by using the feature point/node attributes dialog. As these nodes are converted the hydrologic modeling tree is automatically updated.



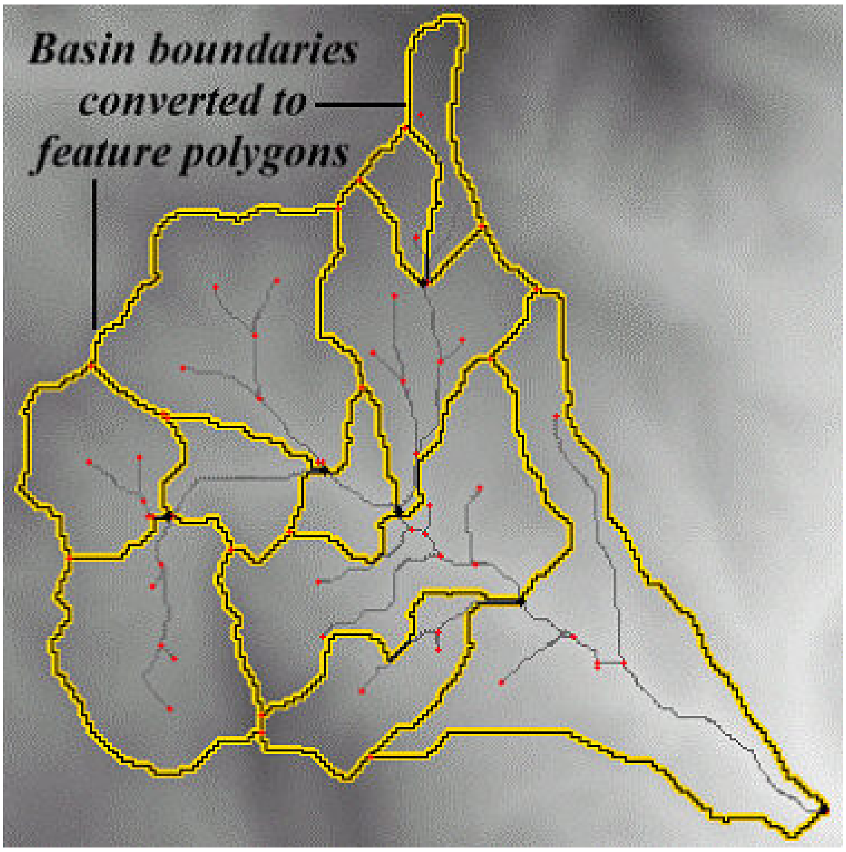
5. Define Basins

Using the outlets on the stream network and the flow directions, the contributing DEM points for each outlet are assigned the proper basin ID.



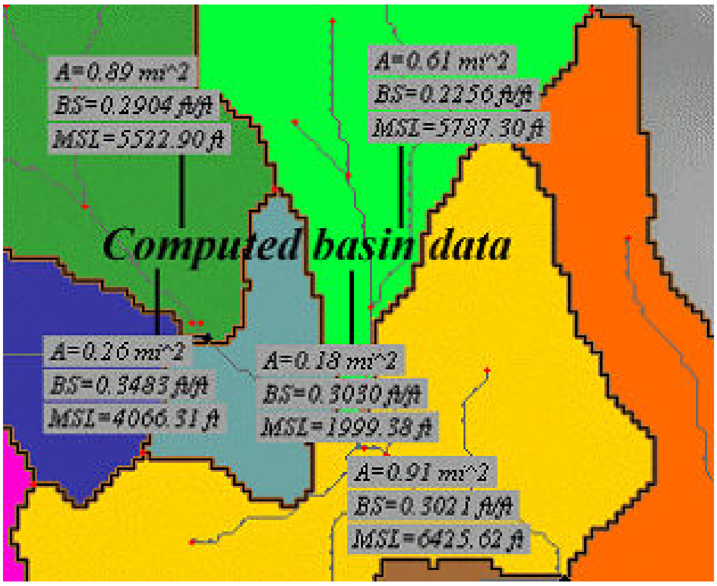
6. Convert DEM Basins to Polygons

Similar to how flow accumulations were converted to stream arcs, the boundaries between DEM points with different basin IDs can be converted to feature polygons. Storing a basin as a single polygon rather than several hundreds (or thousands) of DEM cells is much more efficient.



7. Compute Basin Geometric Data

Once the boundaries of the sub-basins have been determined geometric properties important to hydrologic modeling (area, slopes, runoff distances, etc.) can be computed from the DEM data.



8. Define the Hydrologic Model

At this point you will have the same model as described in the previous section, where watersheds are defined strictly from the feature points, lines (arcs), and polygons. The computed data from step nine is automatically stored in the appropriate locations for hydrologic model definition, and the remaining parameters for the desired hydrologic model can be entered using the appropriate interface dialogs.

Related Topics

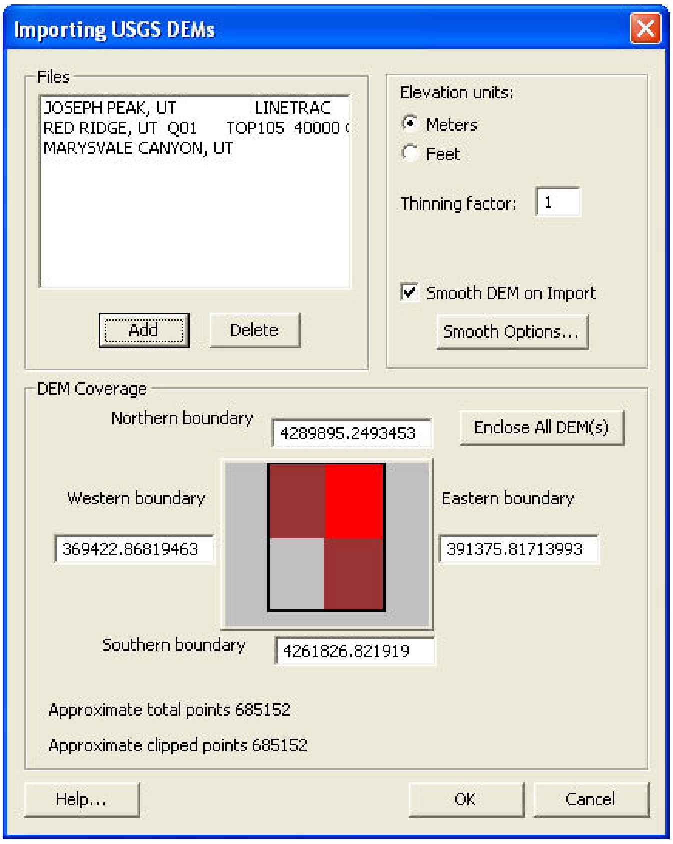
- Hydrologic Modeling
- Feature Object Guidelines
- TIN Guidelines

References

[1] http://edcwww.cr.usgs.gov/doc/edchome/ndcdb/ndcdb.html

Importing USGS DEMs

The **Import USGS DEMs** dialog is used to examine the limits of DEM files as well as defining a clipping boundary to eliminate regions outside the area of interest prior to actually reading the elevations in. The **Add** button is used to add a new file to the list of files (you can now select multiple DEM files at the same time rather than adding them one at a time, but they must all be of the same format) that will be read. The standard file opening dialog appears from which you may select DEM files. Once the file(s) are added to the list a bounding rectangle is displayed in the small graphics window in the center of the dialog. As additional files are added the graphics region is updated with new rectangles in order to provide an understanding of where DEMs are located in relation to one another.



A small black rectangle is displayed in the central graphics window. Only elevation points inside this rectangular region will be read in when hitting OK from this dialog. This boundary rectangle can be modified in three different ways.

- **DRAGGING –** Using the mouse you can click near one of the four edges of the bounding rectangle and drag it to a new location. If you click near a corner both edges will be adjusted. If you click in the center of the rectangle then the entire rectangle can be translated to a new location. As you drag edges to new locations their corresponding values are automatically updated in the edit fields.
- **EDIT FIELDS –** Any one of the western, eastern, northern, or southern boundaries can be explicitly set by changing the values in their corresponding edit fields. As new values are entered the display in the small graphics window can be updated by tabbing or by clicking the cursor outside the current edit field.
- **ENCLOSE ALL DEMs –** This button can be used to force the edges of the bounding to rectangle to correspond to the limits of the DEM files which have been added to this point. By default when a new DEM file is added the bounding rectangle is adjusted to enclose all DEMs.

The thinning factor can be used to reduce the number of elevation points read. A thinning factor of 2 means that every other row and column would be read, reducing the number of total points by a factor of 4. A factor of three means that every third row and column would be read reducing the total by a factor of 9, etc.

The elevation units toggle can be used to specify whether imported DEM points have meter or feet for units of elevation. If a DEMs base elevation units are feet and the toggle specified meters, all elevations are converted when reading. This is particularly important when trying to read two adjacent DEMs with different base elevation units. Also see about transforming coordinate systems.

NOTE: This option does not change the base planimetric units of the DEM and you should make the elevation units consistent with the planimetric units in order to ensure that slopes are computed properly when computing basin geometric parameters.

At the bottom of this dialog the total number of DEM points from all DEM files which have been added and the approximate number of points inside the bounding rectangle are displayed. These numbers can be used to determine how many points your system is capable of reading. For example, each DEM point requires 5 bytes of memory, so that if you read an entire 1:250,000 DEM with about 1.4 million points, 1.4 meg * 5 bytes = 7 meg of memory would be required (in addition to whatever other memory being used by WMS). This means that you would need at least 8 meg of RAM, or some type of virtual memory capabilities would be required to read in the entire DEM file.

This same dialog is used for all five types of DEMs supported in the WMS import options: USGS, ARC/INFO®, GRASS, DTED, and SDTS.

DEMs in Different UTM Zones

Occasionally two adjacent USGS DEM files will be read in but do not appear adjacent in the import dialog. This occurs because while the two DEMs are adjacent, they lie in different UTM zones. X coordinates within UTM zones repeat and therefore the DEMs do not lie adjacent to one another as they should. WMS does not contain the utility functions necessary to transform a DEM from one UTM coordinate zone to another. However, WMS, ARC/INFO® and possibly other GIS software can be used to convert from one zone to another.

Related Topics

- Finding Data for use in WMS
- Coordinate Systems

Importing Flow Directions and Accumulations

A flow direction grid consists of a flow direction value for each DEM point The flow direction identifies which neighboring point has the lowest elevation. A flow accumulation grid consists of an integer value for each DEM point that represents the number of “upstream” DEM points whose flow path passes through it. High accumulation values indicate points in the stream, whereas low values represent areas of overland flow.

Flow directions and accumulations are typically determined using a program, such as ArcView®, GRASS, or TOPAZ. Resulting grid files can then be imported into WMS. These programs all use variations of the eight-point pour model (see References). The figure below illustrates how flow directions are computed by determining which of the eight neighboring DEM points has the lowest elevation. The flow direction value for that DEM point is then assigned an integer number representing the given direction.

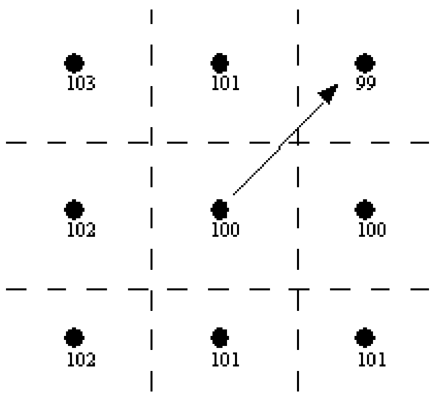


Figure 1: Flow directions are computed by determining which of the eight neighboring DEM points has the lowest elevation

Eight-Point Pour Model.

If all DEM points had one and only one lower neighbor, the process of determining flow directions would be simple and the requirement to use other programs would not exist. However, there are many subtle problems dealing with depressions and flat areas which make the algorithm for determining flow directions complex. Computation of flow accumulations are fairly straight-forward once the flow directions are determined.

At this point computations of flow directions can not be done directly by WMS. However, a version of the TOPAZ program, modified specifically to work with WMS, creates as output the flow direction and flow accumulation grids. These grids can then be imported as DEM attributes and used for basin delineation.

Besides TOPAZ, flow directions may also be computed by ARC/INFO®, ArcView®, or GRASS. No matter which program you use to create flow directions, you will need to import them using the **Import Flow Directions/Accumulations** command. If you use TOPAZ the file will always be named FLOVEC.DAT for flow directions and UPAREA.DAT for flow accumulations. WMS automatically reads these files when the TOPAZ computations are complete.

Flow accumulations are computed by counting, for each DEM point, the number of DEM points whose flow paths pass through the DEM point, and then represented as accumulated area (area of a DEM cell times the number of upstream DEM cells). Streams will be identified by large accumulation values since the flow paths of many points pass through the stream points. For example the outlet of a watershed should have the highest flow accumulation of any of the DEM points since the flow paths all points in the watershed will eventually pass through the outlet point. Flow accumulations may also be imported as a result of using one of the other GIS programs previously mentioned.

Related Topics

- Delineation with DEMs
- DEM -> Stream Arcs
- TOPAZ

DEM to Stream Arcs

The **DEM→Stream Arcs** command is used to create feature arcs from DEM points whose flow accumulation areas are above a defined threshold. An arc vertex is created for each DEM point that has a flow accumulation value greater than the threshold entered. Consecutive stream DEM points are then joined together as arcs with nodes created at junction points where the stream splits.

Outlet points should be created where the watershed outlet of your study area is. These outlet points could be at any DEM point, but should be in a DEM point that has a high enough flow accumulation to pass the threshold (WMS will snap outlet points to the closest threshold cell when you create them in the Drainage module). The Flow Accumulations display option can be very useful for identifying these points and for determining what an appropriate threshold area is.

The **DEM Streams→Feature Arcs** command can also be very useful for defining stream arcs which are later used for creating a TIN surface.

The resulting stream arcs will be jagged because they are created by inserting a vertex at the center of each DEM cell that make up the stream. In order to make the stream arcs appear smoother and more visually appealing you can redistribute vertices along a cubic spline. If you are using a 30 meter resolution DEM, the average length between vertices will be approximately 30 meters and it is suggested that you redistribute to about 100 meters spacing (be sure to turn on the cubic spline option). In general, redistributing to about 3 times the DEM resolution will produce good results.

Related Topics

- Flow Directions and Accumulations
- TOPAZ
- Creating TINs from Feature Objects

Interpolation of DEMs elevations to TINs and 2D Grids

Interpolation to TIN Elevations

The **Interpolate to TIN** command of the *DEM* menu is used to interpolate the elevations of the DEM to an existing TIN. If TIN vertices lie outside the bounds of the active region of the DEM, no interpolation is performed. This interpolation is done automatically when creating a TIN from feature objects and a DEM is used for a background elevation map. If you are trying to create a TIN from a very large DEM (large number of DEM points) it may be advantageous to create the TIN first and then interpolate elevations from the DEM in a piece-wise fashion by dividing the DEM into several smaller regions and interpolating to the TIN one at a time.

Interpolation to 2D Grids

Elevations from the DEM can be interpolated to a 2D finite difference grid used for GSSHA models. This is accomplished using the **Interpolate to 2D Grid** command of the *DEM* menu.

Related Topics

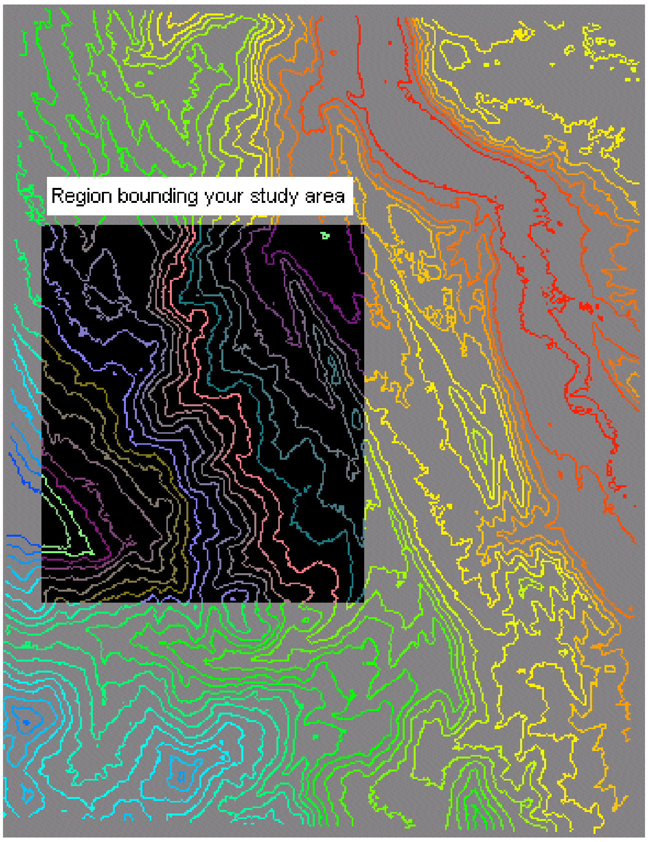
- Creating TINs from Feature Objects
- Creating 2-D Grids

Editing DEMs

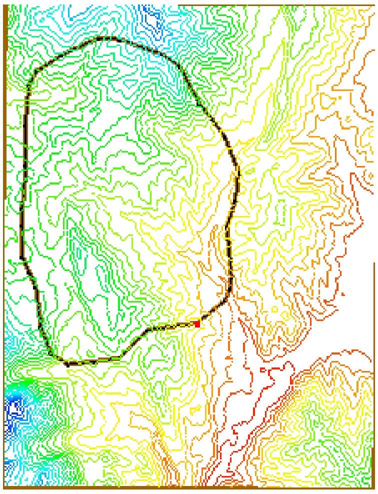
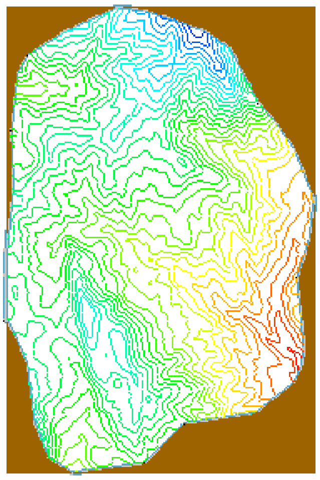
Trimming DEMs

The **Trim** command is used to set the area of DEM that you wish to work with. A DEM by definition includes a rectangular area, but you may not need to process in the entire area to delineate a watershed, or prepare information for a hydraulic model so it is convenient to be able to trim away the parts of the DEM that are not needed.

The illustration belows indicates a rectangle that bounds the area of interest within a larger DEM model.



When choosing the **Trim** command you can enter a polygon interactively or select an existing feature polygon in order to identify the region of interest within the DEM. If a polygon is already selected prior to choosing the **Trim** command then the selected polygon will be used for trimming without a prompt. When trimming with a polygon, only the DEM points within the polygon will have active elevations as everything outside the polygon, but within the bounding rectangle of the polygon is set to a NODATA value as illustrated in the figures below.

Filling Gaps Between DEMs

When reading in multiple DEM files that are adjacent, a small area or gap between the DEMs may have no elevation data. The Fill command will interpolate an elevation for the selected DEM cells (or scan automatically for all such regions if a cell block is not selected) that are classified as "NODATA." The elevation for a selected NODATA cell is determined using inverse distance weighted interpolation from its eight nearest neighbor cells (if any of the eight neighboring cells are NODATA cells then they are not used in the interpolation). This command is intended to correct single isolated DEM points or a single row/column, and is not intended to create data for large regions of NODATA cells, especially regions on the border of the DEM.

Deleting DEMs

The **Delete** command in the **DEM** menu will delete from memory (not the hard drive) the current DEM. A DEM may also be deleted by right-clicking in the Project Explorer and choosing the **Delete** command.

Related Topics

- Inactive NULL Basin
- Trimming a TIN
- Project Explorer
- Deleting Data

Editing DEM Elevations

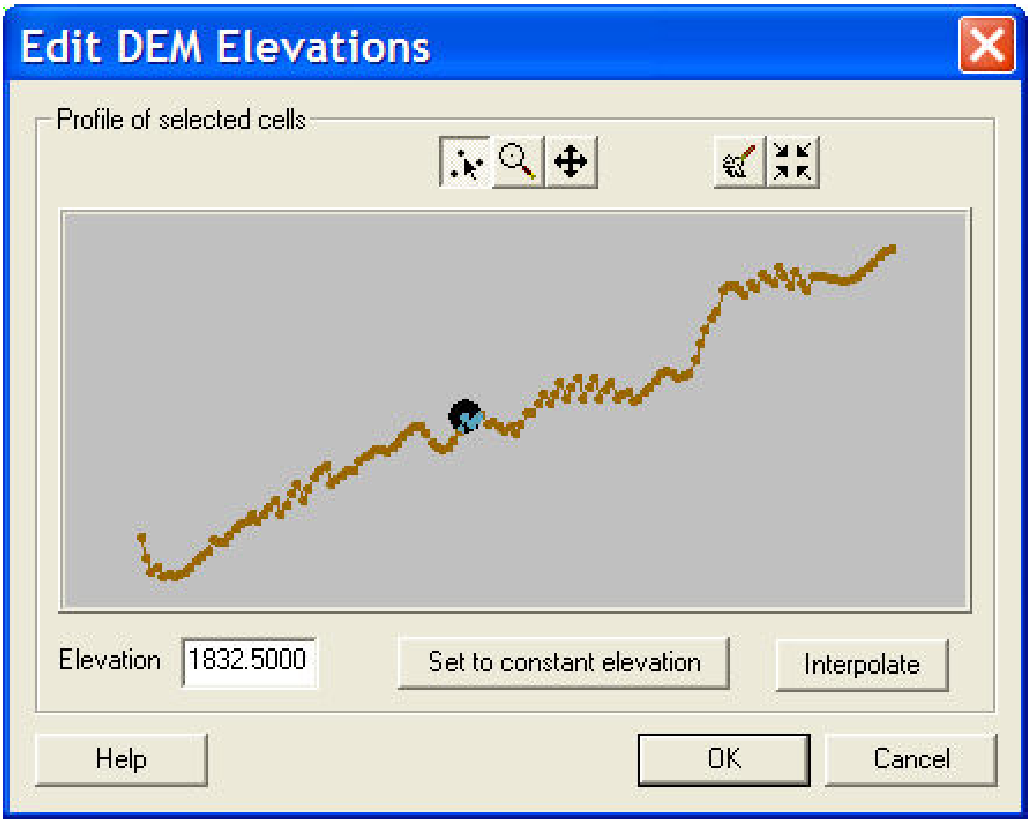
While DEMs are simple to use and more commonly available than TIN data, they are more difficult to edit or adjust, primarily because of the number of elevation points that must be changed in order to have a corresponding effect on the drainage characteristics of a watershed. However, there are a few things that can be done, and if kept to a reasonable amount can be powerful in using a DEM for watershed delineation, but you should remember that data can not be manufactured and if you find yourself making numerous edits then you probably need to think about acquiring a different elevation source.

Editing Single Points

A single elevation can be edited by selecting it with the Select DEM Points tool  and adjusting the z value in the Edit Window, or by double-clicking and bringing up the DEM Point Attributes. Other properties such as defining a depression point or manually changing the flow direction can also be edited in the DEM Point Attributes.

Editing Elevations with a Feature Arc

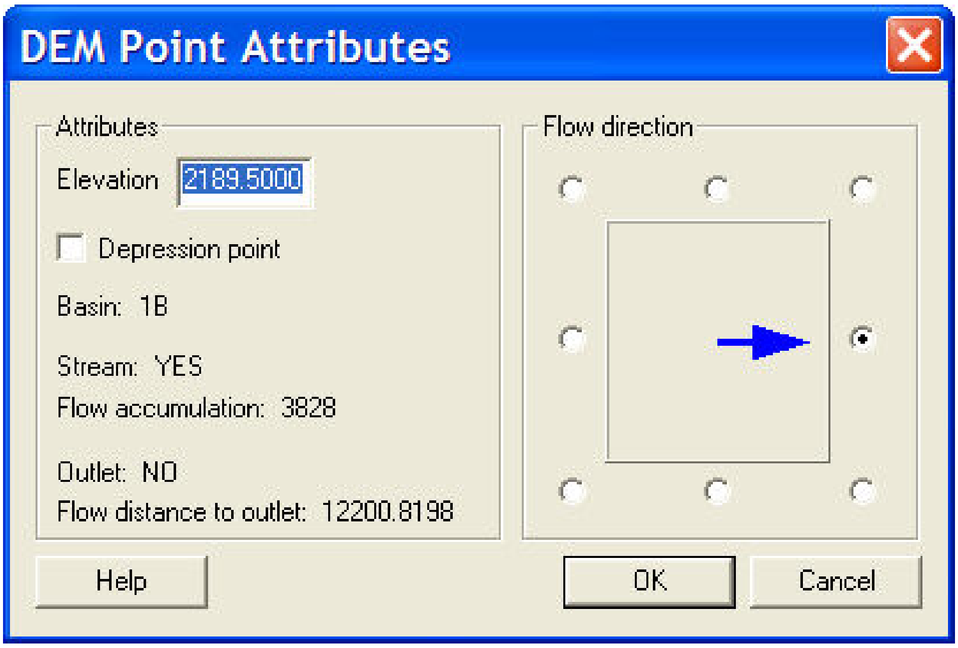
The Edit Elevations command in the DEM menu is used in conjunction with a selected arc to adjust the elevations of all DEM cells that lie beneath the selected arc. Double-clicking on an arc while in the Terrain Data module will also activate the Edit DEM Elevations dialog.



The tools at the top of the dialog can be used to select and edit individual locations along the DEM, zoom, pan, refresh, or re-center. There are three ways to edit the elevations:

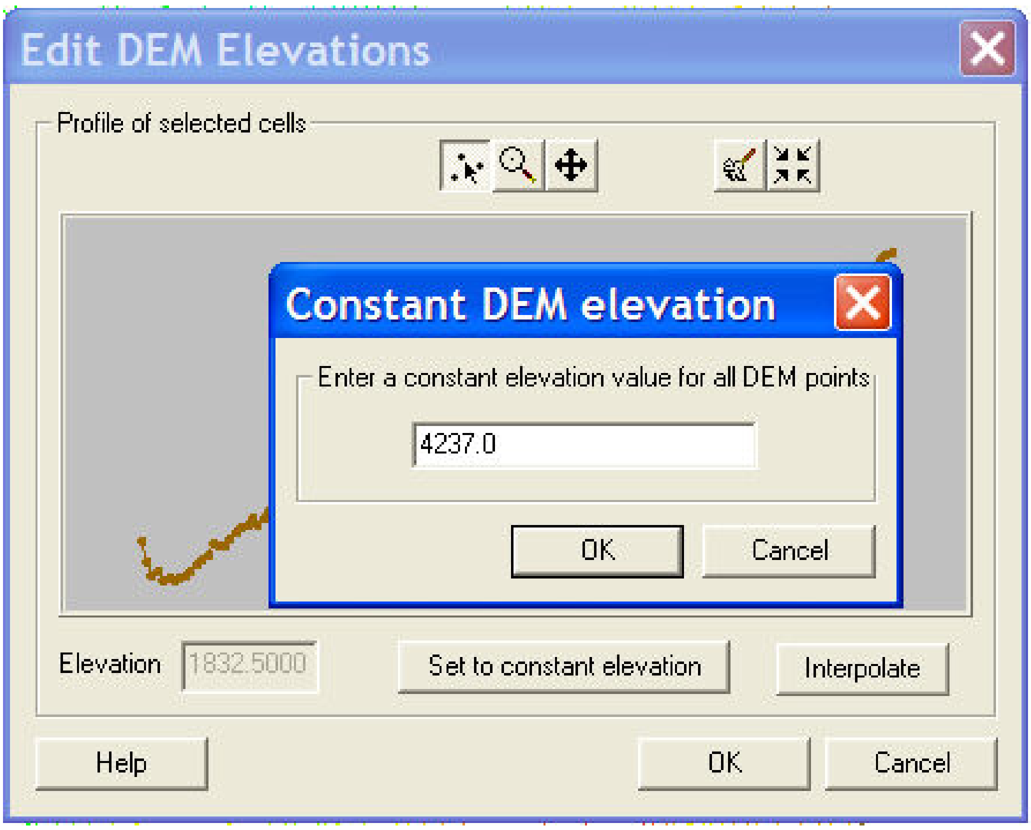
Select individual DEM points and adjust the elevation

Using the Select DEM Point tool in the Edit DEM Elevations dialog you can select a point along the arc and edit the elevation.



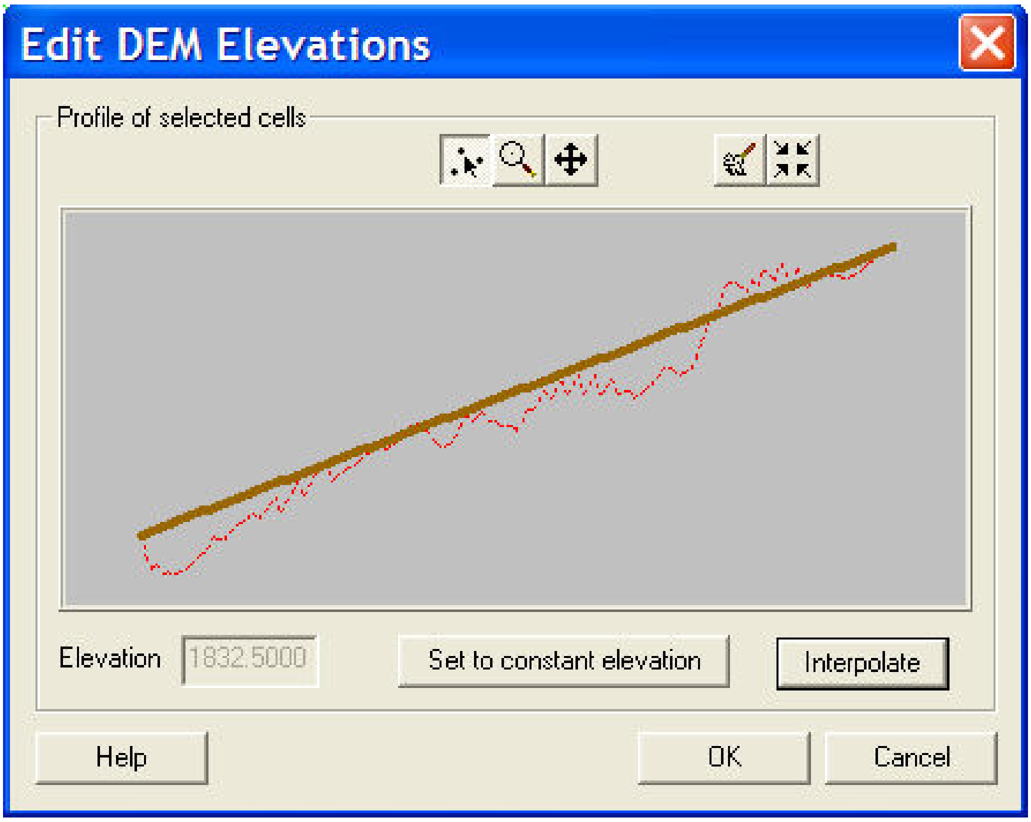
Set/Offset all elevations by a given value

You can set all elevations to a constant elevation, or enter an elevation to adjust all selected DEM points up (positive offset) or down (negative offset).



Linearly Interpolate between first and last elevation

You can linearly interpolate all elevations between the first and last elevation. You may wish to edit the first and last elevation first. Of course this will make the channel artificially smooth, but it still may be more realistic for the channel than the irregular nature of DEM elevations as a result. In this fashion you can adjust the channel to slope to a known value.



Related Topics

- Point Attributes
- Edit Window

Delineation with DEMs

Watershed delineation with DEMs is one of the most common and simplest methods available for automatically characterizing a watershed. Many raster GIS programs have similar capabilities, but WMS has been designed specifically with the purpose of hydrologic and hydraulic modeling in mind. One program that has been developed for delineation on a DEM is the TOPAZ program. A special version of TOPAZ has been created for use with WMS that only requires an elevation grid as input and produces a flow direction grid and a flow accumulation grid as outputs.

After defining basins with a DEM the results are converted to a drainage coverage for easier data storage and manipulation. It is also possible to modify an existing delineation using feature objects.

Related Topics

- Overview of Basin Delineation with DEMs
- TOPAZ
- Flow Directions and Flow Accumulations
- DEM to Stream Arcs
- DEM Basins

Smoothing DEMs

In order to conserve the amount of disk spaced required to store a DEM, many DEM formats store elevations rounded to the nearest integer value. This causes elevation changes to occur in discrete steps rather than smoothly, as would be the case in nature. In regions of low relief, rounded elevations can cause an area to be artificially "flat." Because so many DEMs are stored in this fashion the default in WMS when importing a DEM is to automatically smooth according to the default Smoothing Options. This option can be turned off in the DEM Import dialog if you do not wish to smooth automatically. Once read in the DEM can be smoothed using the **Smooth** command from the **DEM** menu.

Different smoothing options can be set to specify how the smoothing process operates. A description of the different options follows.

Filter Size

When a DEM is a smoothed an N x N filter matrix is placed over each elevation point and a new elevation is computed by taking an inverse-distance weighted average of all elevations within the filter (The weight assigned to the central cell is determined from the filter ratio). The dimension of N can be specified as either 3 or 5, meaning that new elevations are computed from either the nearest 8 or 24 neighboring points. When computing new elevations for points near the boundary, the number of neighboring points is modified to include only those portions of the filter which overlap the DEM.

Iterations

The number of smoothing iterations can be specified in the Smoothing Options dialog. By default only one iteration is done, but sometimes several smoothing iterations are required to propagate a change in elevations across a large flat area (If all neighboring points have the same elevation, no change will be made during the smoothing iteration).

Maximum Change in Elevation

A maximum change in elevation can be specified to ensure that the integrity of the original DEM elevations is maintained. For example, if DEM elevations are rounded to the nearest meter, then smoothing should not adjust the elevation by more than plus or minus one half meter. This value can be increased, but care should be used in doing so in order to keep from "over smoothing" the original data.

Filter Ratio

The filter ratio should be between 0-1, and is used to specify the weight of the central cell of the filtering matrix. It can be used to establish how much effect the DEM point itself has on a newly computed elevation, and how much effect the neighboring cells have. For example, if the filter ratio is set to .75, then 75% of the newly computed elevation will be based on the point itself and 25% will be based on the neighboring elevation points.

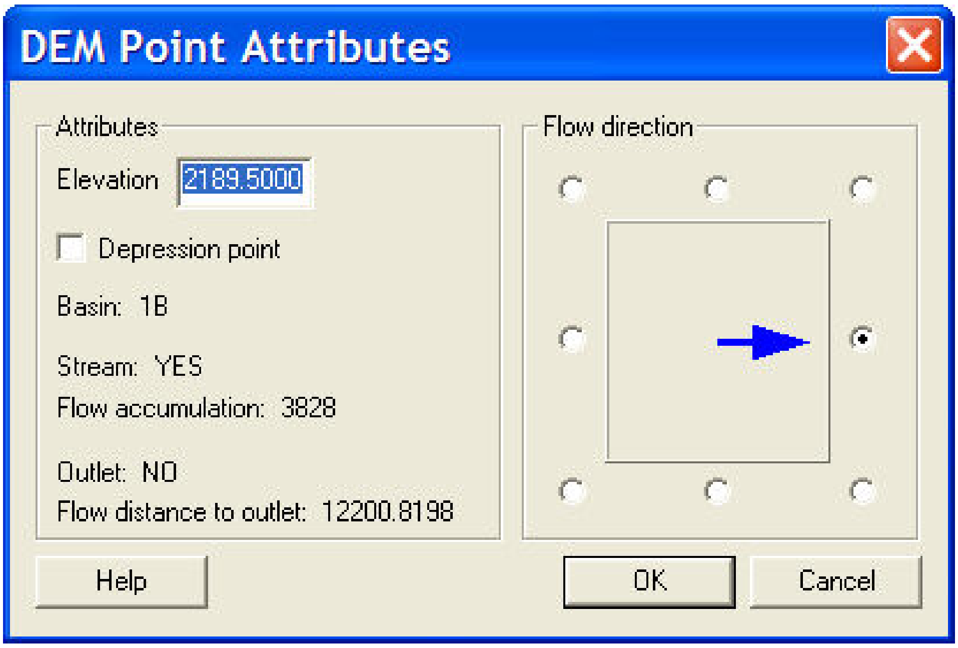
Related Topics

- Importing DEMs

DEM Point Attributes

Point Attributes

Besides the DEM elevation, there are several attributes that can be associated with a DEM grid cell point. The point attributes are generally computed with other functions in WMS (like TOPAZ and as part of the automated delineation), but occasionally it is useful to manually edit the attributes to alter the flow directions in certain locations where the elevations might not be adequate to define the proper drainage, or to create a low spot so that a depression is not filled during the TOPAZ processing. The DEM Point Attributes dialog can be used for such editing, but it should be remembered that editing should be done with care and only with a complete knowledge of the terrain data source and the local drainage around the points being edited. The DEM Point Attributes dialog is accessed by selecting a DEM point with the Select DEM Points tool  and choosing the Point Attributes command from the DEM menu (or by double-clicking on the DEM point from within either the Terrain Data or Drainage modules). The DEM Point of each attribute and how it is used follows:



Elevation

This refers to the elevation of the DEM grid cell. If you have already computed flow directions and accumulations with TOPAZ and you edit the elevation you will want to re-run TOPAZ. Editing elevations one at a time can be very tedious and should only be used for small local changes. You may wish to Edit elevations using feature arcs if you want to make changes along a river, ridge, levee, or some other feature line.

Depression Point

When running TOPAZ it is assumed that all depressions exist because of limited resolution. This means that it is impossible to compute drainage areas for natural depressions as TOPAZ allows "fills" depressions, no matter the size, until they "pour" out. By identifying a location as a depression WMS will write the TOPAZ data for that point as NODATA so that in effect TOPAZ will think it is a DEM boundary and not raise elevation within the depression. This will allow you to use DEMs and TOPAZ to delineate basins of natural depressions. The elevation will remain unchanged and within WMS the DEM cell will not be treated as NODATA, this is only done in order to "trick" TOPAZ into not filling the depression.

Flow Direction

Occasionally it may be desirable to change a flow direction manually, but you should only do so with care and for limited areas. WMS will check to make sure that in changing the flow direction you do not create a circular path (the flow ends up back in the cell you are editing), and if it does you will be prompted that it is not possible and that direction will become dimmed.

After editing a flow direction and selecting OK you will be prompted to recompute flow accumulations, and if basins have been delineated to recompute basin data. This only needs to be done when you think you have finished all edits. If you plan on editing several DEM cells then wait until the last one has been finished before forcing WMS to recompute the flow accumulations and/or the basin data.

Drainage Characteristics

The basin, stream status, flow accumulation, and travel distance to the outlet are displayed for your information, but are not editable within the DEM Point Attributes dialog.

Related Topics

- Delineation with DEMs
- TOPAZ
- Flow Directions and Accumulations
- Computing Basin Data

DEM Files

DEM files are used for storing DEMs processed by WMS. After clipping, thinning, or smoothing an imported DEM you may wish to save it to a WMS formatted file so that it can be recalled later without having to perform the same processing steps. The DEM file format is shown in Figure 1 and a sample file is shown in Figure 2.

DEM	/* File type identifier */
ORIGIN xlowerleft ylowerleft	/* Southwest coordinare of DEM */
DELTAX deltx	/* X spacing of elevation points */
DELTAY delty	/* Y spacing of elevation points */
ELEVATIONS ncol nrow	/* Number of columns and rows in DEM */
z_{11}	/* elevation of row 1 column 1 */
z_{21}	/* elevation of row 2 column 1 */
z_{31}	/* elevation of row 3 column 1 */
.	
.	
.	
$z_{ncol,nrow}$	/* elevation of row nrow column ncol */

Figure 1. DEM File Format.

DEM
ORIGIN 1000.0 1500.0
DELTAX 30.0
DELTAY 30.0
ELEVATIONS 450 300
101
104
.
.
250

Figure 2. Sample DEM File.

The cards used in the DEM file are as follows:

Card Type	**DEM**		
Description	File type identifier. Must be on first line of file. No fields.		
Required	YES		

Card Type	**ORDER**		
Description	Defines the order in which elevations are read in.		
Required	NO (By default row major starting in upper left corner is assumed)		
Format	ORDER ordertype		
Sample	ORDER 0		
Field	*Variable*	*Value*	*Description*
1	xlowleft	0-3	0 - Elevations start at upper left, one row at a time. 1 - Elevations start at upper left, one col. at a time. 2 - Elevations start at lower left, one row at a time. 3 - Elevations start at lower left, one col. at a time.

Card Type	**ORIGIN**		
Description	Defines the lower left (southwest) coordinates for the DEM.		
Required	YES		
Format	ORIGIN xlowleft ylowleft		
Sample	TNAM 1000.0 1500.0		
Field	*Variable*	*Value*	*Description*
1	xlowleft	±	Lower left X coordinate of the DEM
2	ylowleft	±	Lower left Y coordinate of the DEM

Card Type	**DELTAX**		
Description	The X spacing between DEM points.		
Required	YES		
Format	DELTAX deltx		
Sample	DELTAX 30.0		
Field	*Variable*	*Value*	*Description*
1	deltx	+	X spacing of DEM points.

Card Type	**DELTAY**		
Description	The Y spacing between DEM points.		
Required	YES		
Format	DELTAY delty		
Sample	DELTAY 30.0		
Field	*Variable*	*Value*	*Description*
1	deltx	+	Y spacing of DEM points.

Card Type	**ELEVATIONS**		
Description	Elevations for the DEM.		
Required	YES		
Format	ELEVATIONS ncol nrow		
Sample	ELEVATIONS 450 300 101 104 . . . 98		
Field	*Variable*	*Value*	*Description*
1	ncol	+	Number of columns.
2	nrow	+	Number of rows.
3-n	z	+	Elevations of DEM points.

Converting DEMs

Converting DEMs to TINs

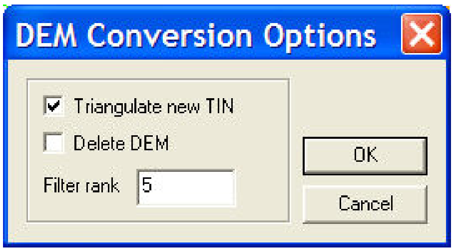
All DEM Points

Using the **Conversion | DEM→TIN** command from the *DEM* menu, you can convert all of your DEM cell points to TIN vertices and triangulate. In general it is not efficient to do this, but it may be desirable to create the TIN for use in extracting cross sections for a hydraulic model, or other purposes where a TIN is required. More efficient ways of converting DEMs to a TIN include using a bounding polygon and interior breakline arcs (i.e. streams and/or ridges), or filtering out less important elevation points using the **Filtered DEM Points** command.

Filtered DEM Points

An alternative to converting every DEM elevation to a TIN vertex (especially for very large DEMs) is to filter out DEM points where the curvature in the terrain between elevation points is small (i.e. a relatively constant slope). The filter algorithm is based on Southard (1990) where a second difference (difference of differences) is computed for each of the eight neighbors (adjusted appropriately for DEM points on the boundary or corners). This second difference is the second derivative of elevation or curvature (the first difference is slope) and represents large changes in slope.

A filter rank is then specified to determine the sensitivity. By default the filter rank is 5 which means a point will be kept if 5 or more of the 8 neighbors have a lower curvature. The maximum value for the rank is 8, meaning the point is only kept if all 8 neighbors have lower curvature, and the minimum is 0, meaning all points will be kept. By adjusting based on a rank and not just the magnitude of curvature, points in flatter areas where curvature may be small, but is changing rapidly relative to the points around it.



DEM Contours to Feature Objects

The DEM Contours to Feature Objects command (**Conversion|DEM Contours→Feature**... command from the *DEM* menu) converts the current linear contours from the DEM to a series of Feature Arcs which could then be exported as a shapefile. Feature arcs carry an elevation attribute and the elevation of the contour is stored in this attribute. When exporting the arcs as a shapefile the elevation attribute field will be saved automatically.

Related Topics

- Creating TINs from Feature Objects
- References
- Saving a Shape File
- TIN Contours to Feature Objects

Create Reservoir in DEMs

The **Create Reservoir** command is used to create a new storage capacity curve for a selected outlet point.

Creating a Storage Capacity Curve

At this point only the Create storage capacity curve option is available for DEMs. The storage capacity curve defines a relationship between elevation/area/volume and is computed from the DEM cell elevations. These three curves are stored in the storage list used by the time series editor so that they can be used later to define routing in one of the supported hydrologic models or in the detention basin calculator. WMS computes these relationships by beginning at the outlet elevation and incrementing the elevation by the number of specified divisions until the specified water surface elevation is obtained. At each increment the area between that elevation and the outlet elevation is computed and then volumes between adjacent surface elevations are computed using the conic method. The storage capacity (elevation, area, volume) data can be stored in either English or metric units.

Related Topics:

- Detention Basin Calculator
- Storage (RS)
- Storage Capacity Curves-Overview

Draw Flow Patterns in DEMs

The **Draw Flow Patterns** command initiates a flow path from the DEM points according to the current display step. By drawing flow paths from the DEM points a visual queue of the watershed flow patterns can be obtained. If a basin polygon is selected prior to issuing the command then flow paths will only be drawn for the DEM points that are part of the selected basin. The display step of the flow patterns can be controlled by modifying the Point Display step option in the DEM Display Options dialog.

Individual flow paths may also be drawn using the draw flow path tool  .

Related Topics:

- Flow paths on TINs

Nodes and Vertices to Outlets

A selected Node or vertex can be converted to an Outlet Node in a drainage coverage using the **Node↔Outlet** command in the *DEM* menu (an existing outlet can be converted back to a node as well). If a vertex is converted to an outlet the vertex is first converted to a node and the attribute of the node is then set to be Drainage Outlet rather than Generic. This is equivalent to setting the attribute of the node within the map module in the normal fashion, but is added to the *DEM* menu for ease and for completeness of the process.

Related Topics

- Drainage Coverage
- Arcs to Outlets
- Drainage Delineation with DEMs
- Converting Vertices to Nodes

DEM Basins

Defining Basins

Each time a feature outlet point is created, a basin for each upstream feature arc is created for the hydrologic modeling tree. This means that the stream arcs themselves are associated with a basin even before the **Define Basins** command is issued. When the command is used the DEM points intersected by the stream arcs are assigned the basin ID already given to the arcs. The Define Basins procedure then continues by tracing the flow paths of the remaining DEM points until a point which has already been assigned a basin ID is intersected. The result is that each DEM point gets assigned the ID of the sub-basin it belongs to.

Knowing that DEM flow paths only run until a stream is intersected can help you modify the results of a delineation by creating your own stream arcs (independent of those created using the **DEM→Stream Arcs** command) that "intercept" flow along a line that might not be represented by the underlying elevations. An example of this is along a highway embankment. The scale of most DEMs does not capture a feature such as a highway embankment, yet it is an important feature in basin delineation. You can create a stream arc along the highway alignment and leading to a culvert or bridge crossing. The resulting basin delineation will then "capture" all flow that intersects the line and account for the raised highway embankment. The following figures illustrate the use of this concept.

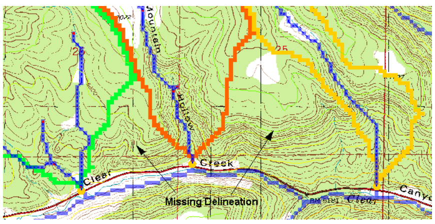


Figure 1: Delineation from the TOPAZ results only (not including the roads as streams)

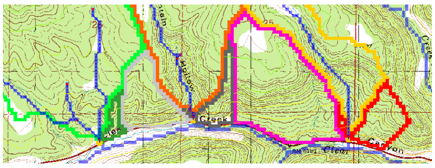


Figure 2: Delineation after including the roads as streams. It is important to note that streams have been made in both directions from the outlets in order to "capture" the area that will flow to each outlet.

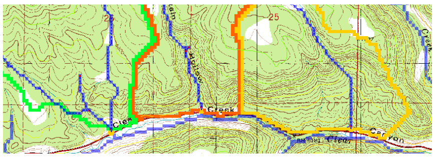


Figure 3: If you leave the outlet of the basin at the bottom node it will create separate basins for the main branch and each branch along the road. In order to combine them into a single basin a small (about one cell in length or less) stub stream is created so that only a single drainage will be used. Remember that WMS will create a separate basin for each upstream branch from an outlet point.

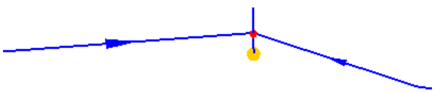


Figure 4: Close up of an outlet so that a single basin is defined.

Additional outlet points can be created by changing the attribute of existing arc nodes to outlets or by converting arc vertices to nodes and then changing the attribute to outlet. The Node↔Outlet command can be used to accomplish this. Any selected node or vertex will automatically be converted to an outlet node when using this command. Any selected outlet node will automatically be converted to a generic node type when using this command.

The Define Basins command can be used any number of times to redefine basins after the addition/deletion of outlet nodes.

Basin Boundaries to Polygons

Once the desired sub-basin delineation from the DEM points has been defined, the basin boundaries can be converted to feature polygons. This is done by tracing the boundaries between sub-basins to generate arcs. After all of the boundaries have been defined the arcs are converted to polygons and the polygons assigned the appropriate basin ID.

The resulting polygon boundaries will be jagged because the arcs created for the polygons trace around each raster DEM cell. In order to make the boundaries appear smoother and more visually appealing you can redistribute vertices along a cubic spline. If you are using a 30 meter resolution DEM then the average length between vertices will be approximately 30 meters and it is suggested that you redistribute to about 100 meters spacing (be sure to turn on the cubic spline option). In general, redistributing to about 3 times the DEM resolution will produce good results.

Computing Basin Data

After defining basin boundaries, attributes such as basin areas, slopes, and stream lengths can be computed using the **Compute Basin Data** command. These attributes are all geometric parameters used in defining basins and routing networks in HEC-1, TR-20, and other hydrologic models. If the basins are changed in any way, the drainage data must be recomputed using this command. When computing basin data the model units and the parameter units must be specified. The only choices available for model units are feet and meters whereas the parameter for area include square miles, square kilometers, and acres, and for distance include mile, kilometer, feet, and meters. A complete definition of the different geometric attributes computed and how they may be used to compute travel times (lag time, time of concentration) is given in the Hydrologic/Hydraulic Calculations page.

Editing Basin Variables

The **Basin Variables** button will let you view/edit any of the basin variables that are computed by WMS. While it is unnecessary to edit these variables, you may find some cases where you want to override what WMS has computed and use a value you have derived through a separate analysis.

Merge Selected Basins

The **Merge Selected Basins** command merges two selected sub basins into a single basin. The two sub basins must be directly connected to the same outlet point.

By default WMS will create a separate basin for each upstream branch of an outlet point. However, you may wish to combine basins of all branches of an outlet into a single basin. The **Merge Selected Basins** command allows you to do this.

An alternative approach (and really a better way to do it for data management purposes) is to define the basin just downstream of the junction as illustrated in the following figure.

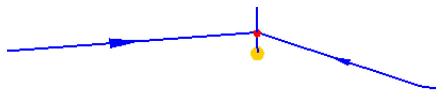


Figure 1: Define basin just downstream of junction

Delete NULL Basins Cell Data

The **Delete Null Basins Cell Data** command is used to delete all DEM points which are not currently assigned a basin ID. The DEM is reduced to a bounding rectangle of the watershed and elevation values within the rectangle but outside of the watershed are converted to NODATA.

Polygon Basin IDs to DEM

The **Polygon Basin IDs→DEM** command assigns Basin IDs to the DEM from a set of polygons that represent basin boundaries. This command is useful if you have not delineated your basin using the flow directions and flow accumulations from within WMS, but instead have a set of polygons representing basin boundaries with unique IDs. Once basin IDs have been assigned, basin data such as area, average slope, etc. can be computed.

You would likely only use this option if you had a set of feature objects already and/or wish to “over-ride” the basin boundaries that are determined from elevation data. This might occur in an urban watershed where streets, canals, etc. may not be apparent in the digital elevation data.

Delineate Basins Wizard

The DEM Guidelines page outlines as steps the basic process for watershed characterization with DEMs. Some of these steps are repeated quite frequently and rather than needing to perform them one at a time you can select the Delineate Basins Wizard command and the steps will be performed in succession so that you do not have to do them one at a time.

In order to run the **Delineate Basin Wizard** you should have already read in your elevations (edited if desired), run TOPAZ to compute the flow directions and flow accumulations, and established your initial outlet points. The wizard will then convert the outlets to streams, define basins, convert the basins to polygons, and compute basin data in succession.

The delineation wizard is best suited for use when you have a single basin to delineate rather than several smaller basins that make up a larger basin. For such cases it is probably best to follow the individual steps to prevent lumping the multiple smaller sub-basins into a single large basin.

Manually Edit Delineated Basin

Occasionally the DEM does not contain all elevation data so when delineating a basin, the resulting watershed is inaccurate. This could occur if there is a road or other man-made obstacle dividing the watershed in half. In cases like these, it is necessary to manually edit the watershed boundaries. NOTE: care should be taken when manually editing a watershed as this can lead to inaccurate results if not done correctly.

Upon delineating a basin, WMS creates a coverage that is usually called "Drainage". Select this coverage then use the Select Feature Vertex tool to drag out where the new border should be. Once you have the polygon where you want it, switch to the Drainage Module then select *DEM | Polygon Basin IDs→DEM*. This should redefine your basin where the polygon was.

Related Topics

- Watershed Delineation with DEMs
- Importing Flow Directions and Accumulations
- Node to Outlet
- DEM → Stream Arcs
- Compute Basin Data

Compute Basin Data

When the drainage coverage feature objects are used to create a hydrologic model the area of the polygons can be determined and used in any of the supported hydrologic models. If the points/vertices used to create the feature arcs also have z values associated with them then WMS will make a rough estimate of the longest flow path and slope along such a path. Finally, if you have derived a watershed from feature objects, but have a background DEM you can compute most of the basin data parameters using the following steps:

1. Import/Read the DEM.
2. Compute the flow directions using TOPAZ.
3. Use the Polygon Basin IDs→DEM command found in the *DEM* menu in the Drainage module to assign DEM cells a basin id from the feature object polygons.
4. Choose the **Compute Basin Data** command from the *Feature Objects* menu of the Map module.

Of course, the results will only be approximate since the actual basin boundaries will not have been derived from the computed flow direction data, but it will provide a reasonable estimate. Ideally you will want to derive the basin boundaries from the DEM and flow direction data and then use the Compute Basin Data command in the *DEM* menu of the Drainage module.

Related Topics

- Feature Object Guidelines
- Computing Basin Data from DEMs
- Computing Basin Data from TINs

TOPAZ

The Topographic Parameterization Program (TOPAZ) was developed by the USDA-ARS, National Agricultural Water Quality Laboratory under the direction of Dr. Jurgen Garbrecht. TOPAZ is a public domain program that is distributed free of charge to interested persons. A modified version of the program is distributed with WMS for the purpose of computing flow directions and flow accumulations for use in basin delineation with DEMs. However, TOPAZ is capable of further DEM elevation processing, including raster smoothing, basin and stream delineation and ordering, and development of other watershed parameters. If you are interested in obtaining the latest, complete version of TOPAZ, visit the United States Department of Agriculture's (USDA) Agricultural Research Service website (here [1]) and click on TOPAZ.

If you have or obtain the complete TOPAZ program, WMS is capable of writing an input file for DEDNM (the primary TOPAZ module). DEDNM requires as input a file containing the elevations (must be named DEDNM.INP) and a control file named DNMCNT.INP.

TOPAZ crashes if the file name is 60 characters or longer.

Computing Flow Data with TOPAZ

This command will save the current DEM elevations to a TOPAZ formatted input file along with a TOPAZ control file and then start the TOPAZ program. When TOPAZ is finished WMS will automatically read the flow directions (FLOVEC.DAT) and flow accumulations (UPAREA.DAT) computed by TOPAZ.

Because WMS is communicating with TOPAZ through disk files you must specify the directory where WMS/TOPAZ perform this interaction. If your specified directory has an unusually long name TOPAZ may have some difficulties. In such cases you will want to specify a directory with a shorter character string length.

TOPAZ assumes that all depressions are a function of a lack of resolution in the DEM data and therefore it is not possible to delineate the watershed of a closed basin unless you identify the DEM cell of a natural depression to be a Depression Point from the DEM Point Attributes.

TOPAZ also creates a file named RELIEF.DAT that has the elevations of the depressionless or filled DEM. TOPAZ (and most grid-based delineation algorithms) assumes that any depression in the DEM is a result of inadequate resolution and not a natural depression. This is true for most depressions, but means that basins for natural depressions cannot be determined. However, with WMS you can define a low point in a natural depression as such and then WMS will flag it so that TOPAZ does not fill the depression and the drainage computed. This is done using the DEM Point Attributes dialog prior to running TOPAZ.

Exporting a TOPAZ Input File

The **Export TOPAZ File** command is provided for those who wish to use TOPAZ for purposes outside of what WMS needs (the flow direction and accumulation files). This format should be close to the public domain version of TOPAZ, but may be slightly out of date.

Related Topics

- Flow Directions and Accumulations
- Defining Basins
- References

References

[1] http://www.ars.usda.gov/main/main.htm

5.2. MapInfo Professional Transaction File(TINs)

TIN Guidelines

TINs are formed by connecting a set of xyz points (scattered or gridded) with edges to form a network of triangles. Points used to create TINs can be obtained by digitizing a contour map (or a scanned image inside of WMS), by generated automatically from feature arcs and polygons, or by using DEMs or existing TINs as background elevation maps. TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and have several other display options that can be set to visualize and understand the terrain surface better. TINs should primarily be used for elevation data conversion, merging various elevation sources, creating DEMs, and extracting cross sections and other data for hydrologic models. The WMS developers recommend using DEMs for hydrologic modeling. Hydrologic modeling using DEMs is simpler and more robust than hydrologic modeling using DEMs.

However, TINs can be used for basin delineation and drainage analysis. Basin areas and several other geometric parameters can be computed and combined with hydrologic analyses. This section focuses on steps that can be used to process TINs for watershed delineation, but some of the features described here can also be useful for other needs.

The *Project Explorer* can be used to set the name of the TIN by either double-clicking on the TIN name, or by right-clicking on the name and choosing **Rename**. The name of the TIN is always displayed along with the TIN icon when using the **Select TIN** tool.

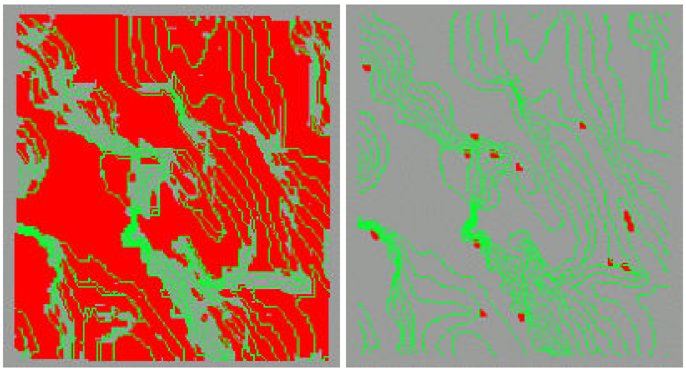
Developing watersheds from TINs often involves the use of both feature objects and DEMs. The following steps can be used as a guideline for watershed characterization with TINs.

1. Obtain Background Elevation

An elevation source is required for creating a TIN. Even if you already have a TIN data source we recommend that in most cases you use it as a background elevation source and a create a new TIN from it. A background elevation source can be a DEM, TIN, or both.

2. Smooth the Background Elevation

Digital elevation data are often rounded to the nearest integral value (foot or meter) for storage efficiency. However, this can cause problems for automated basin delineation techniques, especially where there is relatively little relief. WMS has utilities for smoothing both DEM and TIN background elevation data. The results can be dramatic as seen in the set of figures below, and often make the difference in being able to successfully complete a watershed modeling project.



3. Create a Conceptual Model with Feature Objects

In order to ensure that triangle edges in the resulting TIN will conform to streams and other important drainage features, you need to identify them with feature objects. A rough basin boundary defining the domain of the TIN region needs to be created. Additionally, any lines such as streams and roads that should be represented with triangle edges should also be created as part of the conceptual model.

Conceptual models can be created in many different ways, but some of the easiest ways in WMS include:

1. Import existing digital data in GIS, CAD (DXF), DLG, or other simplified xy formats.
2. Use a contour display of a DEM and on-screen digitize the rough boundary and stream arcs.
3. Use a registered image as a background for on-screen digitizing.
4. Use a DEM to define flow accumulations and then convert to stream arcs as described in step five of the previous section.
5. Use a hillshaded DEM image as a background for on-screen digitizing. The Screen Capture command in the *Images* menu can be useful for saving a hillshaded DEM as an image file.

4. Redistribute Vertices

The density of vertices in the TIN created from the conceptual model and background elevation data can be controlled by the spacing of arc vertices in the conceptual model. WMS has tools to automatically increase or decrease arc vertex density. The density may vary along the arcs, allowing for higher definition in some regions and lower in others.

5. Create TIN

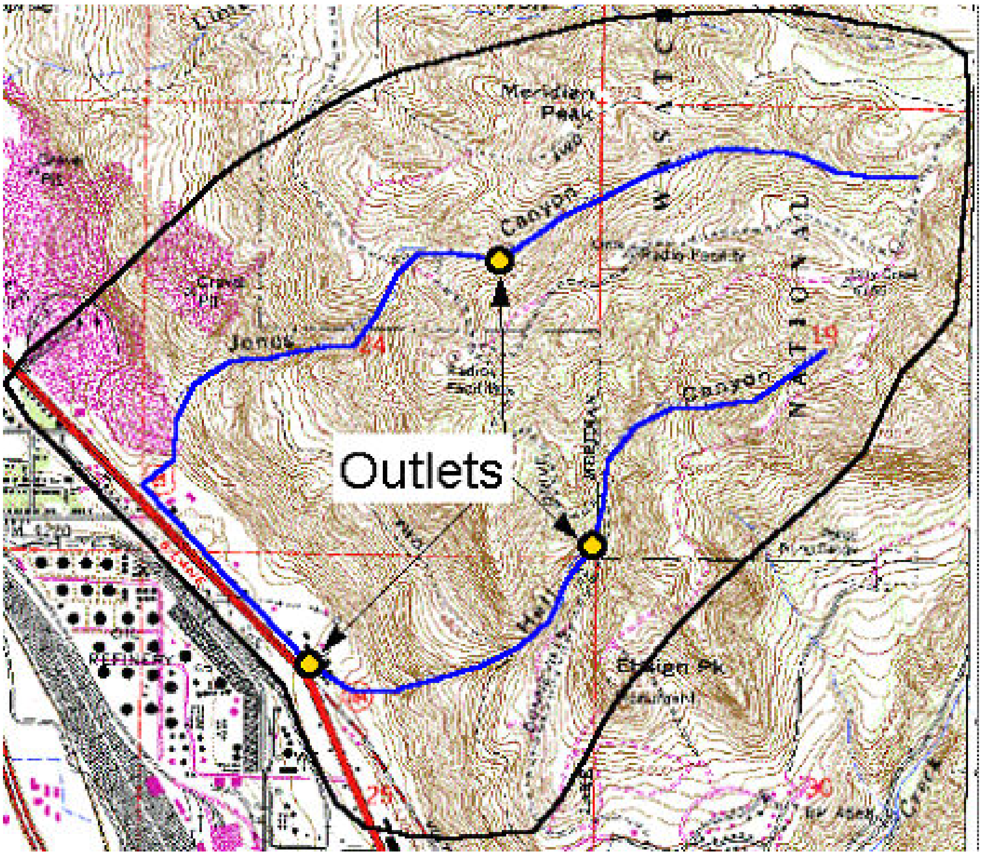
TIN creation builds on the previous four steps. The outer polygon is used to define the limits or extents of the TIN. TIN vertices are created inside this polygon at a density proportional to the spacing of vertices on the nearest arc. After the TIN is created, the stream (and other interior) arcs are forced as breaklines so that they are honored in the TIN as triangle edges, and the elevations for the vertices are interpolated from the background elevation map.

6. Edit TIN

Even though the newly created TIN conforms to the topographic features defined by feature arcs, there are inevitably some anomalies that must be corrected in order to use the TIN for basin delineation. These include flat triangles, flat edges, and pits. WMS contains several tools for both automatic and manual (user interaction) elimination of these anomalies.

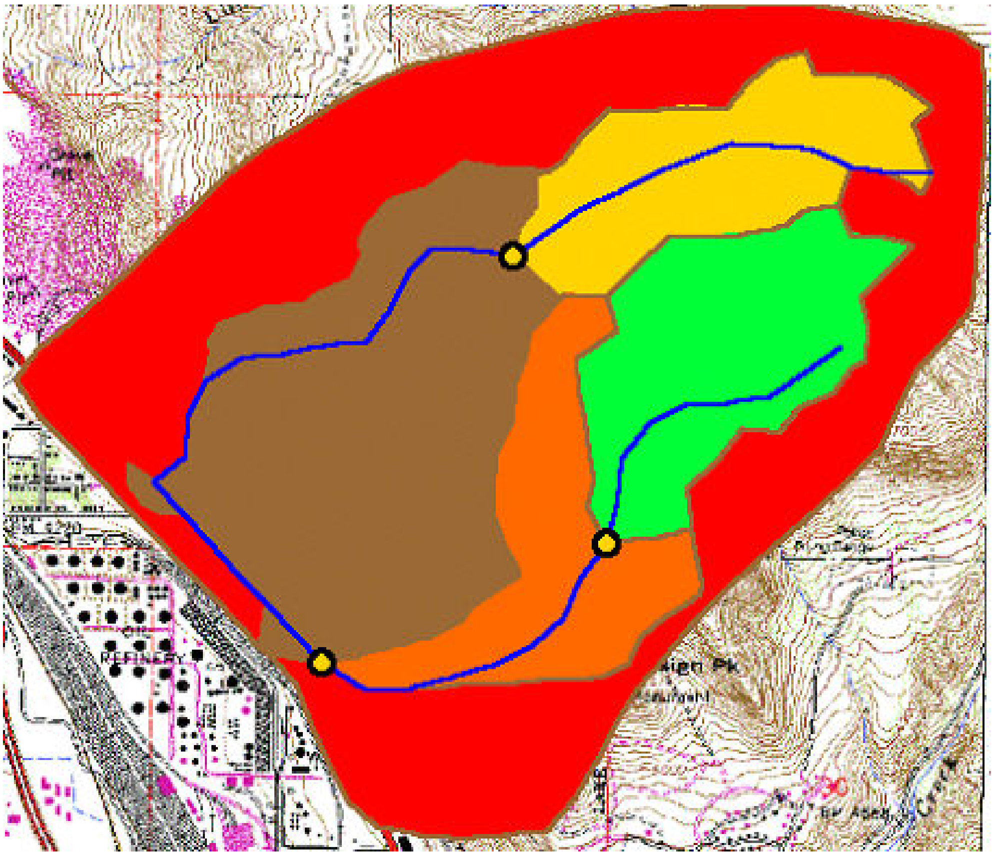
7. Complete Stream Network and Outlet Definitions

By default there may only be a single outlet point for the watershed defined, or perhaps only a portion of the stream network. WMS can be used to add additional outlet points (representing sub-basin outlets, culverts, etc.) and stream branches after the TIN has been created from the feature objects. Even after delineation you can return to this step and redefine the locations where you would like sub-basins created.



8. Delineate Basins

A flow path is initiated from the centroid of each triangle and followed until the first outlet point. The triangle is then assigned the basin ID corresponding to the branch of the stream the flow path entered from (by default, a separate basin is created for each upstream branch of an outlet). Basins can be merged later, providing they both belong to the same outlet point.



9. Refine TIN

During the basin delineation process in step eight some problems with divergent or splitting flow paths may occur. Again, WMS has tools which will allow you to correct these problems automatically and manually. The automatic method will correct the problem about 90% of the time, while in 10% of the cases you may need to swap edges or edit the TIN in some other way in order to eliminate the split flow vertices.

If there are no split flow vertices, or after you have corrected them, you will want to refine your boundaries and then eliminate triangles exterior to the actual watershed. Triangle basins are defined based on flow paths from the centroid of the triangle, so some triangles will actually straddle the basin boundary. The Refine Boundaries command of the *TIN* menu will split these triangles along the true boundary and results in "smoother" basin boundaries. Once you are satisfied with your watershed boundary the Delete Null Basin Triangles command is used to eliminate all triangles whose flow paths do not pass through an outlet (basin).

10. Compute Basin and Stream Parameters

With the stream network and basin boundaries defined you can compute several important geometric parameters. These parameters (area, slope, length, etc.) are automatically tied to the hydrologic models (HEC-1, TR-20, etc.) where appropriate.

11. Define the Hydrologic Model

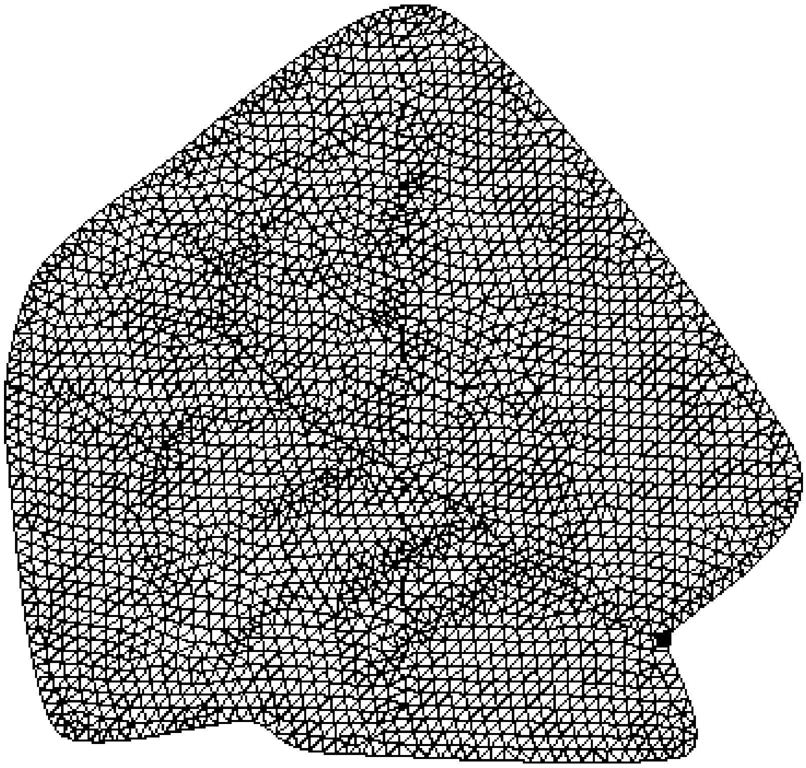
Along with the watershed definition on the TIN, an accompanying topologic model is created. You can then interact with the TIN or tree representation of the watershed to complete input for and run the supported hydrologic models.

Related Topics

- DEM Guidelines
- Feature Object Guidelines
- TIN Interpolation
- Hydrologic Modeling
- Data Acquisition

Creating TINs

A TIN can be created from a set of feature objects in a drainage coverage using the **Create TIN** command in the **Feature Objects** menu. The density of vertices in the TIN will be proportional to the vertex spacing along arcs. The Redistribute command in the Feature Objects menu can be used to adjust vertex spacing and locally refine the TIN in important areas. Either an existing TIN or a DEM can be used as a background elevation map when interpolating z values for the vertices of the TIN. If appropriate z values have been assigned to the feature arcs then the z values from the arcs will override z values interpolated from the background elevation map for TIN vertices created from feature arcs vertices.



For TINs requiring a lot of memory (high resolution of vertices or covering a large spatial extent), it may be advantageous to build the TIN in the absence of a background DEM. Interpolation of elevations to the TIN from the DEM afterwards can be done in a “block by block” fashion using the **Interpolate | ...to TIN** command found in the **DEM** menu of the Terrain Data module. In other words you can read in portions of the DEM and interpolate to TIN multiple times. Elevations for TIN vertices that are not within the extents of the current DEM are not interpolated. No such option exists if a TIN is used as the background elevation source.

You may wonder why if you already have TIN data, you would ever use a TIN as a background elevation set. The primary purpose of creating TINs from feature objects is to insure that stream channels and other important hydrologic features are adequately represented in the TIN as triangle edges. If you simply triangulate a set of xyz scatter points, or import a TIN from another data source, it is not likely that this condition will exist. Creating a new TIN from feature objects will insure that the TIN is optimal for performing drainage analysis because the new TIN will be built “around” the feature objects.

Creating a TIN from a Scanned Image

One of the easiest ways to create a TIN for a small area where a paper contour map exists is to use the digitize toolbar and follow the steps outlined below:

1. Scan the paper map and save it as a TIFF (*.tif) image.
2. Register the image (you may want to mark the map with your register points prior to scanning it).
3. Turn on the Digitize toolbar and then turn on **digitize mode**.
4. In the value box in the digitize toolbar, set the value to the desired contour value.
5. Using the **Create Vertex** tool, digitize or create vertices along the specified contour value (the spacing of points along the contour lines should be approximately the same distance as the spacing between adjacent contours).
6. Repeat steps 4 and 5 for each contour line. Spot elevations can be entered by setting the z value to the value of the spot elevation and then creating a vertex at that location.
7. Triangulate the vertices once you are done.

Of course, this method is awfully tedious for larger areas, but is ideally suited for smaller areas where there are not many contours to be digitized.

Related Topics

- Feature Objects
- Creating Watershed Models
- Feature Object Guidelines
- Creating Grids
- Converting DEMs
- Fill Command
- Merging TINs
- TIN Options
- Registering Images
- Creating Vertice

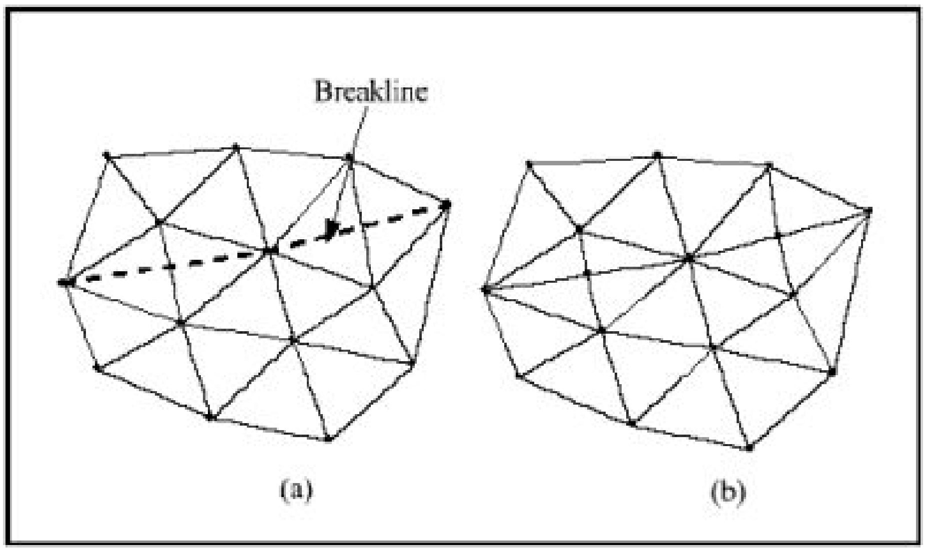
TIN Breaklines

A breakline is a feature line or polyline representing a stream channel, ridge or some other feature that you wish to preserve in a TIN. In other words, a breakline is a series of edges that the triangles should conform to. Breaklines can be very useful when trying to eliminate unwanted pits on the interior of a TIN.

Breaklines can be processed using the **Triangles | Insert Breakline(s)** command from the **TIN** menu. Before selecting the command, one or more sequences of vertices defining the breakline(s) should be selected using the Select Vertices tool in the Tool Palette.

Breakline Options

Breakline Options option are controlled in the TIN Options of the **TINs** menu. This dialog allows you to specify to either interpolate the z values from the existing TIN or to get the z values from feature arcs. The elevations of the new vertices are based on a linear interpolation of the breakline segments. The locations of the new vertices are determined in such a way that the Delauney criterion is satisfied.



Related Topics

- Edge Swapping
- Creating Vertices
- Triangulation
- Delauney Triangulation
- TIN Options

Streams

Streams can be defined for a TIN by manually connecting, or linking, consecutive channel edges together. They appear similar to stream feature arcs, but are not. On a TIN the streams are defined simply as connected vertices.

Because TINs are created from a sparse set of points, it is often difficult to explicitly define channel edges, particularly in urban areas where well defined channels may not even exist. For this reason, a stream can be created by manually selecting a set of vertices which defines a channel. This method also allows a street or storm drain to be incorporated into the TIN and used as part of the "stream network."

Create Streams

When the **Create Streams** command is chosen, a stream network for the current selected string of TIN vertices is created. The vertex string is made by connecting points which are known to lie in a stream, street, or other drainage structure. If the entered vertex string crosses triangle edges, the user will be asked if he wishes to insert a breakline. The breakline is inserted by creating new points where the line crosses a triangle edge. The elevation of the new points is determined by linear interpolation along the edge. Inserting the breakline in this fashion alters the topology without affecting the geometry.

Since flow through a stream network is defined strictly by the “linked” set of vertices, a downstream vertex does not even have to be lower than the upstream vertex. However, care should be taken to ensure that the general direction of the stream is downhill. The intent of creating streams in this fashion is to eliminate the need of defining a continuous set of channel edges with the TIN editing techniques described above. This type of stream creation is particularly important when doing basin delineation for urban areas.

Deleting Stream Segments

The **Delete Stream Segments** command deletes the segment of the stream between two selected stream nodes. An outlet point is inserted at the upstream node. If a single stream node is selected this command deletes the portion of the stream network from the selected stream node upward, including the selected stream node.

Create Pipe

When the **Create Pipe** command is chosen, a pipe connecting stream nodes in the current selected string of TIN vertices is created. A pipe can be used to create a "stream" from any stream node to another without enforcing a continuous set of triangle edges between the two stream nodes. Pipes differ from streams in that when a flow path intersects a stream it then continues down the stream from node to node, whereas when a flow path intersects a pipe it continues overland across the pipe.

Displaying the Stream Profile

The **Display Stream Profile** command is used to display the elevation profile between two selected stream nodes. Because of the limited resolution of most elevation data sets, it is difficult to get a continuous set of stream bed elevations. Within this dialog you can select and edit the elevation of individual stream nodes or you can select two different stream nodes from the profile plot and linearly interpolate the elevation of all nodes in between. In this way you can smooth out pits in the stream that may exist only because of a lack of resolution.

When using this command you must select at least two stream nodes and the second one selected must be upstream of the first.

Related Topics

- Outlets
- Defining Basins
- Reservoirs

TIN Basins

Defining Basins

The **Define Basins** command assigns each triangle in the TIN to a drainage basin. This is accomplished by initiating a flow path from the centroid of each triangle and "flowing" down until an outlet point is encountered. The triangle is then assigned the appropriate basin ID. The boundaries may appear rough or jagged because each triangle is assigned according to the flow from its centroid, when in fact the triangle may actually straddle basin boundaries. Boundaries may be corrected by issuing the **Refine Boundaries** command. The drainage basin boundaries option in the Drainage Display Options dialog is automatically set when defining basins.

Refining Boundaries

After the initial definition of drainage basins, many of the boundaries are rough or irregular. Triangles straddling true basin boundaries can be split using the **Refine Boundaries** command. This process is accomplished by tracing paths of maximum upward gradient along boundaries, splitting triangles when the path crosses over them, and then reassigning all affected triangles to their new basins. The process is displayed graphically.

Merging Basins

For each stream branch upstream from an outlet point, a drainage basin is automatically created. The recommended method of merging basins is to establish the outlet at a node just downstream of the junction.

However, if you need an alternative method to merge basins, there is an option to merge selected basins together using the **Merge Basins** command. The **Merge Basins** command allows you to combine the basins for a given outlet. In order to merge basins, they must be adjacent to each other and belong to the same outlet. In order to select drainage basins, the *Select Drainage Basins* tool must be active.

Editing Basin Variables

The **Basin Variables** button will let you view/edit any of the basin variables that are computed by WMS. While it is unnecessary to edit these variables, you may find some cases where you want to override what WMS has computed and use a value you have derived through a separate analysis.

Assigning Triangles to Basins

Occasionally you may wish to manually define triangles to a basin. This is particularly important if you have a large flat area (such as a lake) within a basin and you do not wish to edit the elevations such that they would flow to the outlet. You can select the triangles you wish to assign and then choose the **Assign Triangles to Basin** command in order to assign the triangle to a basin. You will be prompted to select the outlet of the basin you wish to assign the selected triangles to.

Deleting NULL Basin Triangles

The **Delete Null Basin Triangles** command can be used to delete all triangles whose flow path does not encounter an outlet. Before defining drainage basins all triangles are classified as belonging to the null drainage basin. After defining drainage basins some triangles still belong to this null basin since they do not contribute flow through any of the given outlets. They should not be deleted if further editing is to be done. However, once all sub-basins have been properly defined, they can be deleted in order to reduce the size of the model to the region of interest.

Computing Basin Data

After defining basin boundaries, attributes such as basin areas and slope and stream lengths and slopes can be computed using the **Compute Basin Data** command. These are all geometric parameters used in defining basins and routing networks in HEC-1 and TR-20. If the TIN is edited or sub basin configuration changed, the drainage data must be recomputed using this command.

Once computed, they can be displayed along with the basins in the Graphics Window or written to a file using the **Export File** command found in the *File* menu. By default, basin IDs and areas are displayed after computing drainage data. The other attributes can be toggled on for display in the Basin Attributes dialog accessed from within the *Drainage Display Options* dialog.

Units

When the **Compute Basin Data** command is given the *Model and Parameters Units* dialog appears allowing you to specify the current Units of the model (TIN vertices) and the units you wish to use for computed parameters. TIN vertices must either be in feet or meters, but the computed areas and lengths can be given separate units. If you wish to change the units of computed parameters at a later point you must recompute the basin parameters with different selections. You may wish to convert the coordinate system prior to computing the basin data.

Related Topics:

- Outlets
- Streams
- Reservoirs
- HEC-1 Reservoirs
- Delineation with TINs
- Delineation with DEMs
- Travel Times from Basin Data
- Variables Computed by WMS
- Compute Basin Data
- Delineation with TINs
- Delineation with DEMs
- Units

TIN Menu

Trimming TINs

Trimming allows you to eliminate all vertices that are outside of a selected feature polygon. You must first create the feature polygon in the Map module, then select the polygon you wish to use prior to selecting the **Trim** command from the *TIN* menu (Terrain Data module).

Correcting Split Flow

Split flow vertices can usually be corrected by finding a channel edge leading into the split flow vertex, and swapping it. This edge swapping can be done automatically using the **Correct Split Flow** command from the **TIN** menu of the Drainage module. If the edge cannot be swapped without creating overlapping triangles the split flow vertex will not be eliminated and you will have to add new vertices, adjust elevations, swap multiple edges or some other form of manual editing technique.

TIN Interpolation

Interpolate TIN To TIN

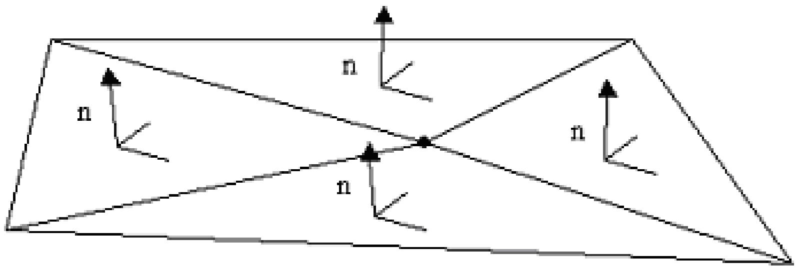
The **Interpolate to TIN** command of the **TIN** menu allows you to interpolate the elevations of the active TIN to one or more TINs. If more than one other TIN exists a dialog will appear that lets you select the TIN(s) you wish to interpolate to. No changes occur in the elevations of the active TIN.

Interpolate TIN To 2D Grid

The **Interpolate to 2D Grid** command of the **TIN** menu interpolates the elevations of the active TIN to the 2D Grid. Only available if both a TIN and a 2D Grid exist.

Filter

Redundant and overlapping data may exist in a scattered XYZ data sets. WMS offers the ability to filter the data and remove unnecessary data points in relatively flat areas in the **Filter** command from the **TIN** menu. The user specifies an angle. Each data point is checked to see if it is in a flat region by dotting the normals of the surrounding triangles.



If the normals are all within the specified angle, the region is considered flat and the point is deleted.

This type of processing is very important when trying to use LIDAR data. Typical LIDAR collection results in resolutions of 1-3 meters so that good detail of flood plains and channel banks is achieved. However this leaves massive amounts of points in relatively flat areas where the increased resolution is not required (it would be like asking your surveyor to get points every 5 feet just so he doesn't miss anything, when hopefully he can be much smarter). The filtering eliminates the points where they are not needed and retains the important points. (The reference to Creighton Omer's paper is about a study using this technique on LIDAR data for hydraulic modeling that concludes a filter angle of 4-8 degrees can be used that will result in up to 85% data reduction without impacting hydraulic modeling results.)

Fill

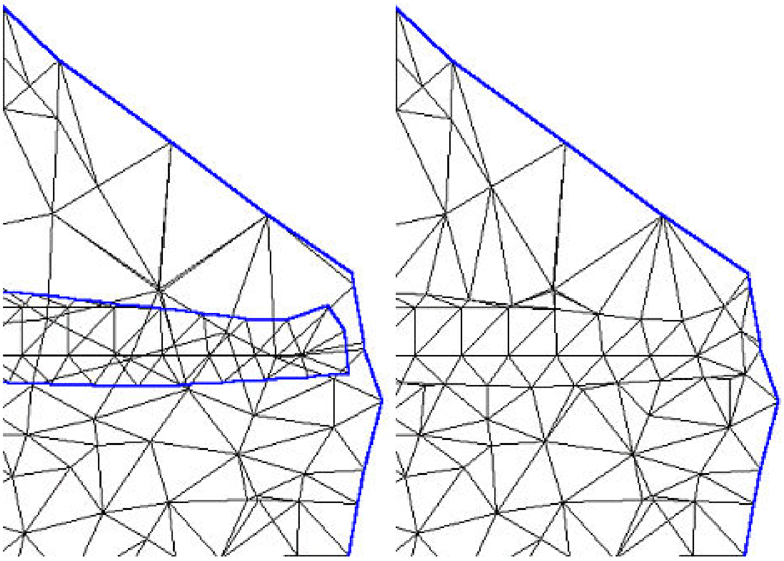
The **Fill** command triangulates a selected polygon and merges it with the original TIN. Vertices should be distributed according to the density of TIN vertices desired on the interior of the TIN. The primary purpose is to fill in an area where data is missing on a TIN, or where you wish to retriangulate to a higher density.

You must have some kind of background elevation behind the polygon (e.g. a DEM or another TIN), or the resulting vertices within the polygon will all have zero elevations.

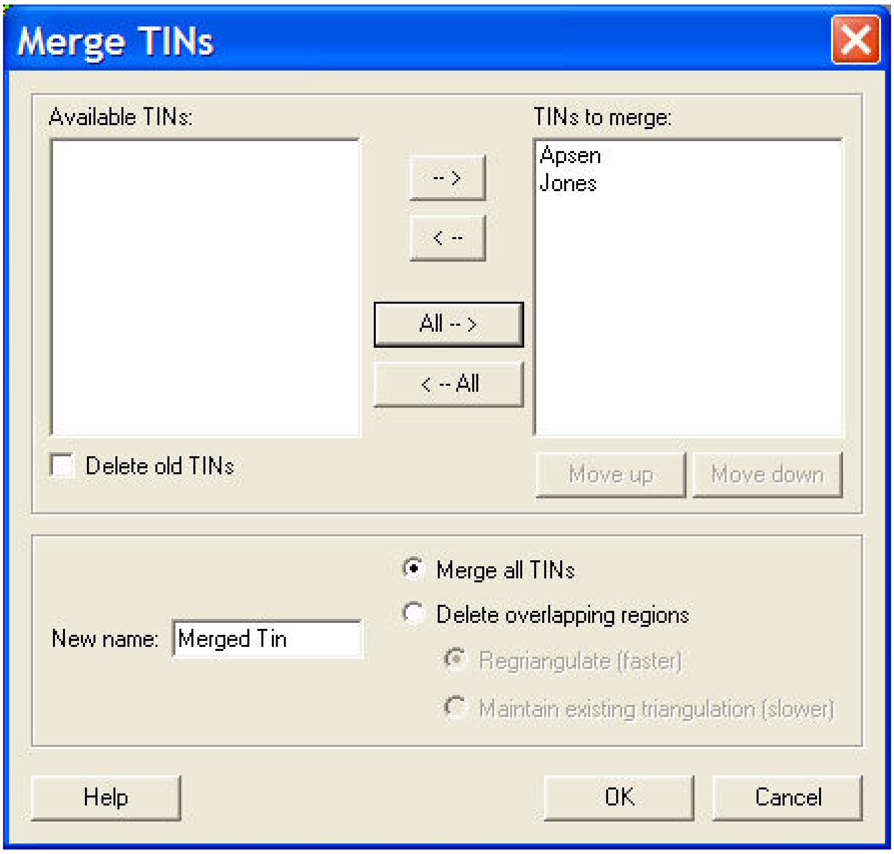
Merging TINs

You can merge a selected TIN with another TIN using the **Merge** command in the **TIN** menu. This is particularly useful if you wish to merge a TIN generated from one program with a TIN derived from a background elevation source such as a DEM.

For example you may have surveyed data with a lot of detail for a part of your project (such as a roadway profile) and wish to combine that with a TIN derived from a DEM of the surrounding area.



The following dialog is used to select existing TINs for merging. The merged TINs can be completed merged (all vertices combined and retriangulated), or you can specify to have vertices in overlapping regions deleted. You can delete old TINs if you wish, or keep them.



The following rules are adhered to when merging a list of TINs:

- The TIN at the bottom of the list of TINs to merge has the highest priority, meaning that all of its triangle edges will be preserved while vertices that overlap from TINs higher in the list will be deleted.
- If a region being deleted from one TIN overlaps any stream vertices on the TIN being deleted, the stream will be split. The stream will be cutoff wherever it "enters" the TIN selected for merging and begin again upstream where it "exits."
- Drainage data will be lost and not transferred as part of the merged TIN.

Related Topics

- TIN Guidelines
- Triangulating
- Deleting Vertices
- 2D Grids
- Interpolation Options

Convert TINs

TIN to DEM

A DEM may be created from any TIN using the **Convert | TIN→DEM** command in the *TIN* menu. A resolution (x and y spacing) must be defined and then an interpolation method specified (linear is the default) to interpolate DEM elevations from the TIN. Since a DEM must be rectangular, elevations that are outside of the TIN boundary are given a NODATA value of -9999.

TIN to Scatter Points

The **Convert | TIN→Scatter Points** command creates a 2D scatter point set from the TIN. One data point is created for each of the vertices in the TIN. A dataset is made of each from the elevations of the TIN vertices. This command can be used as part of the process to create smooth contours by subdividing the TIN and then interpolating from the newly created scatter points.

TIN Boundary to Polygon

The **Convert | TIN Boundary→Feature** command in the *TIN* menu can be used to create a boundary feature polygon from the vertices on the boundary of the TIN. Such a polygon could then be used to create a grid from a feature polygon using the **Create Grid** command found in the *Feature Objects* menu (Map module).

TIN Contours to Feature Objects

The **TIN Contours to Feature Objects** command (**Convert | TIN Contours→Feature**) converts the current linear contours from the TIN to a series of Feature Arcs which could then be exported as a shapefile. Feature arcs carry an elevation attribute and the elevation of the contour is stored in this attribute. When exporting the arcs as a shapefile the elevation attribute field will be saved automatically.

Related Topics

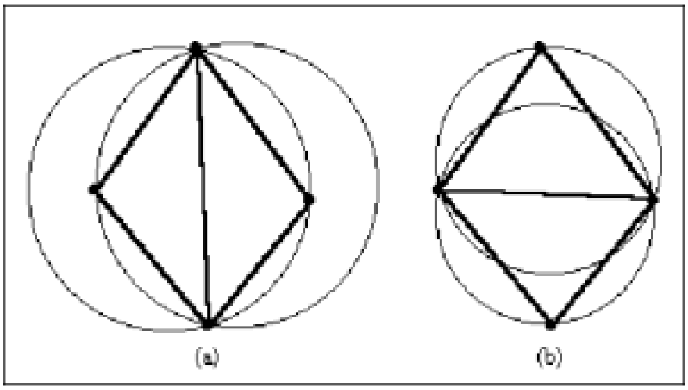
- Interpolation Options
- Scatter Point Sets
- Creating Grids
- Drainage Data to Feature Objects
- Saving a Shapefile
- DEM Contours to Feature Objects

Triangulation

A TIN can be constructed by triangulating a set of vertices. WMS connects the vertices with a series of edges to form a network of triangles. The resulting triangulation satisfies the Delauney criterion The Delauney criterion ensures that no vertex lies within the interior of any of the circumcircles of the triangles in the network (figure shown below).

As the triangulation process proceeds, adjacent triangles are compared to see if they satisfy the Delauney criterion. If necessary, the adjacent edge of the two triangles is swapped (the diagonal of the quadrilateral defined by the two triangles is changed to the other two vertices) in order to satisfy the Delauney criterion. This edge swapping process forms the basis of the triangulation algorithm.

When a new point is inserted into a TIN, the point is incorporated into the TIN and the edges of the triangles adjacent to the new point are swapped as necessary in order to satisfy the Delauney criterion. If the Delauney criterion is satisfied everywhere on the TIN, the minimum interior angle of all of the triangles is maximized. The result is that long thin triangles are avoided as much as possible.



It is important to note that the triangulation described above is used as a preliminary step to creating a TIN conditioned for basin delineation and is not sufficient in most cases for actually doing the drainage delineation. Even if you begin with TIN data you will want to create another TIN using feature objects.

Triangulating

Vertices can be triangulated using the currently selected triangulation algorithm by selecting the **Triangles | Triangulate** command from the **TIN** menu. It is important to recognize that the Delauney triangulation is not necessarily the best for performing drainage delineation because it does not insure that important linear features such as streams and ridges will be honored in the TIN as triangle edges. For this reason you should always use a TIN triangulated in this fashion as a "background" elevation source for creating a new TIN from a "conceptual" model of feature objects.

Triangulation Optimization

The **Triangles | Optimize Triangulation** command of the **TIN** menu will optimize triangulation according to the following criterion:

- If Angle optimization is selected in the TIN Options dialog, the edges of triangles will be swapped to form edges that match the Delaunay criterion.
- If Area Optimization is selected in the TIN Options dialog, the edges of neighboring triangles will be swapped if the area of one triangle is more than the bias times the area of the smaller triangle.

The criteria is specified in the TIN Options dialog.

Remove Flat Triangles

The **Triangles | Remove Flat Triangles** command attempts to eliminate flat triangles on a TIN. A first pass is made in attempt to adjust the triangulation or slightly alter vertex elevation and a second pass is then made which inserts new vertices in flat triangles and interpolates the elevation.

Interpolating Flat Triangles

This method inserts new points in flat triangles and adjusts the elevation of the new points by using an interpolation technique. This method works well when there is a small number of clustered (2-10) flat triangles. However, when large regions of flat triangles exist, the TIN filtering should be used before trying to remove flat objects.

When the **Triangles | Remove Flat Triangles** command in the **TIN** menu is issued, WMS computes the differences between the elevations of the flat triangles and the elevations of the surface defined by the IDW quadratic interpolation scheme. This technique is used because of its ability to accurately infer important terrain features such as pits, peaks, streams, and ridges. The difference between the flat surface and the interpolated surface is referred to as the "deviation" of the flat triangles. The deviation is computed at regularly spaced points on the interior of each flat triangle. The subdivision factor in the Interpolation Scheme dialog controls the level of subdivision or the number of interior points. The point in each flat triangle with the maximum deviation is assumed to represent the maximum for that triangle.

Once the deviations are determined, WMS locates the flat triangle whose deviation is the maximum. A new point is added at the xy location of the maximum deviation. The elevation of the new point is computed using the IDW quadratic interpolation scheme. The new point is inserted into the TIN, and the TIN is adjusted locally to accommodate the new point. Many times the insertion of a new point in a flat triangle combined with the local retriangulation of the TIN results in the removal of several flat triangles. The list of flat triangles is updated, the flat triangle with the next largest deviation is found, and the process is repeated. By inserting new points in this fashion, the minimum number of new points will be added in the best possible locations to infer local minima and maxima such as pits, peaks, streams, and ridges.

Once all flat triangles have been eliminated, further processing to remove flat edges and pits is done. Prior to performing the **Triangles | Remove Flat Triangles** command all TIN vertices are locked. Any new vertices created as part of this process are set to ulocked status. When completed you will be able to distinguish the new vertices from the original by observing the ones that are unlocked. You may wish to unlock all vertices for further editing.

Boundary Triangles

The perimeter of the TIN resulting from the triangulation process corresponds to or approximates the convex hull of the data points. This may result in some long thin triangles or "slivers" on the perimeter of the triangulated region. There are several ways to deal with the long thin triangles.

Selecting Boundary Triangles

The thin triangles can be selected and deleted using the normal selection procedures. There is also an option for selecting thin triangles when the Select Triangles tool is selected. If the CTRL key is held down, it is possible to drag out a line with the mouse. All triangles intersecting the line will be selected.

Another technique can be used to select long thin triangles on the perimeter of the TIN. By selecting the **Select Boundary Triangles** command from the **TIN** menu, the thin triangles on the perimeter of the TIN are automatically selected.

The **Select Boundary Triangles** command checks triangles on the outer boundary first. If the length ratio of the triangle is less than the critical length ratio, the triangle is selected and the triangles adjacent to the triangle are then checked. The process continues inward until none of the adjacent triangles violate the minimum length ratio.

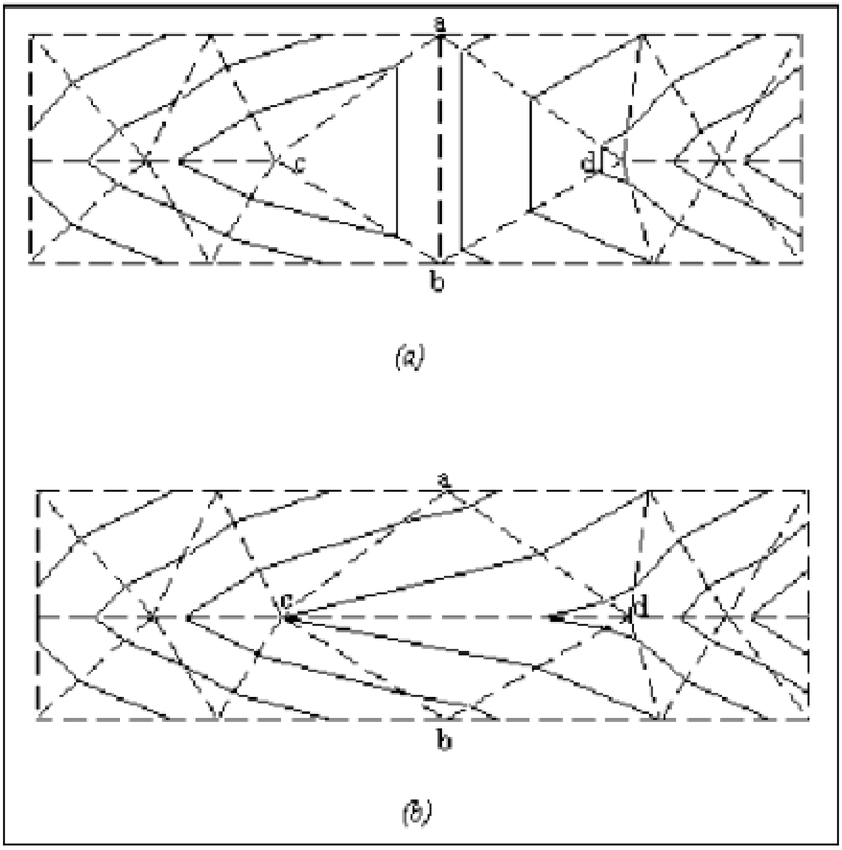
Thin Triangle Aspect Ratio

The critical length ratio for selecting thin triangles can be set by selecting **Length Ratio** from the **TIN** menu. The length ratio is defined as the longest side of the triangle divided by the sum of the two shorter sides.

A maximum edge length may also be specified in the TIN Options dialog.

Edge Swapping

TINs are generated in WMS using the Delauney criteria. This method creates a set of triangles which are as equiangular as possible, and while this generally creates a good terrain surface, it does not ensure that all important hydrologic features such as streams and ridges will be honored with triangle edges. A classic problem which occurs and inhibits drainage analysis is the "false dam". A false dam occurs during the triangulation process when an edge straddles a natural channel, forming a dam in the bottom of the channel as shown in Figure a. False dams are easily corrected by swapping the triangle edge ab to cd as illustrated in Figure b. Triangle edges are swapped using the Swap Edge tool and clicking on the edge which needs to be swapped.



Related Topics

- Creating TINs from Feature Objects
- TIN Options
- Creating TINs from Feature Objects
- Trimming TINs
- Filtering TINs
- Smooth Pits
- Locked/Unlocked Vertices
- Interpolation Options
- Selecting Objects
- Delete
- Breaklines
- Creating Vertices

Reservoirs

Creating Reservoirs (TINs)

A set of triangles can be grouped together to create a reservoir. When creating reservoirs an outlet point must be specified for the triangles so that any flow path intersecting a triangle belonging to a reservoir can be routed directly to the outlet. The **Node↔Outlet** command is used to convert a stream node into an outlet point.

Deleting Reservoirs (TINs)

An entire reservoir can be deleted by selecting the reservoir outlet using the Select Vertices tool and issuing the **Delete Selected Reservoir** command.

Related Topics

- Outlets
- Streams
- Defining Basins

Watershed Delineation with TINs

A TIN terrain model can be used in WMS to delineate stream networks and drainage basin boundaries. Since the terrain model is an accurate geometric description of the watershed, parameters such as areas, slopes, and flow distances can automatically be computed by WMS. This terrain model then serves as a map to guide entry of all data necessary to run HEC-1, TR-20, or other hydrologic analysis programs.

It should be emphasized that it is highly recommended you prepare your TIN for drainage analysis by using feature objects with a background elevation source. The elevation source could be TIN or DEM, but you should retriangulate using feature objects as guides in order to insure that stream edges are honored by triangle edges.

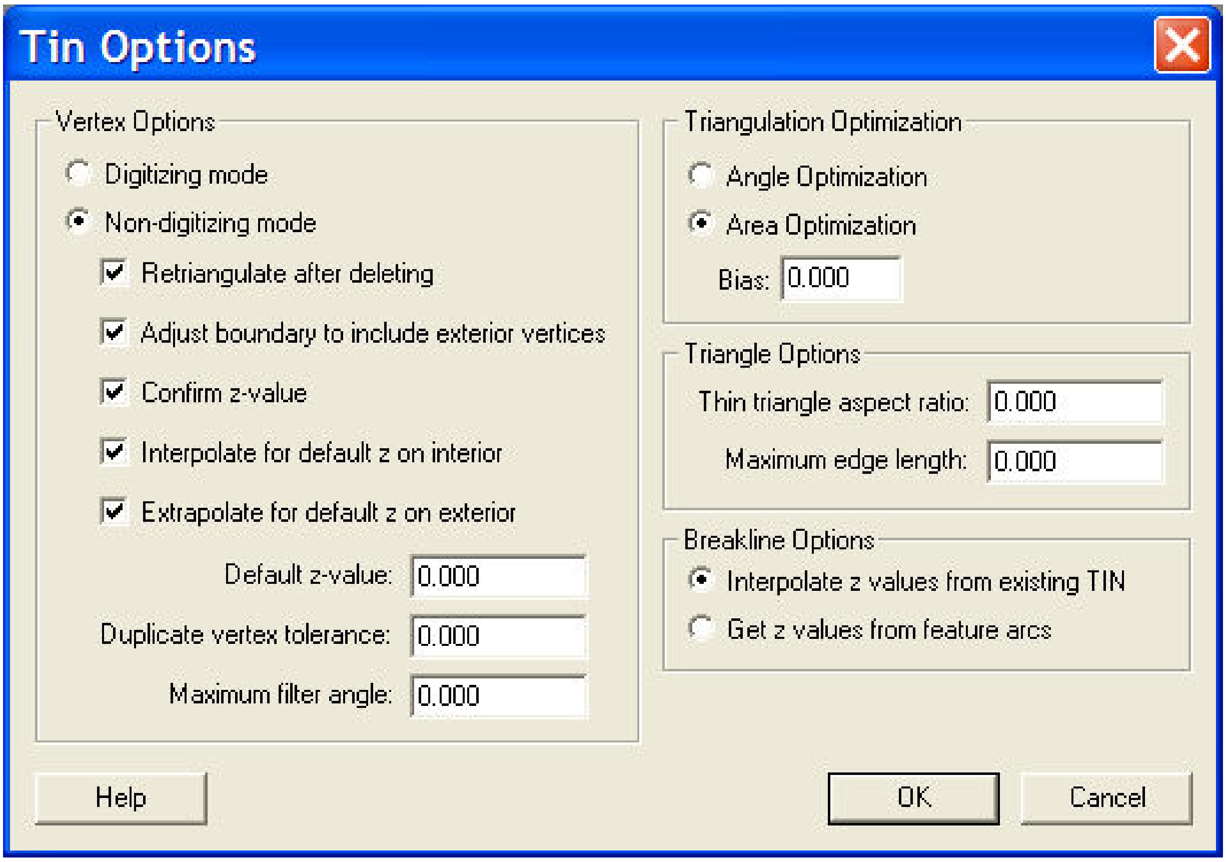
The first process in performing drainage analysis is to edit the TIN where necessary. Flat triangles, flat channel edges, and flat ridge edges must all be eliminated before trying to delineate stream networks and basin boundaries. Automatic editing procedures, such as TIN filtering and removal of flat objects, should be used. In addition, manual insertion of breaklines, the addition of new points, and edge swapping can aid in removing anomalies which are introduced into the TIN as a byproduct of the triangulation process. With the TIN properly edited, stream networks and drainage basins can be defined as preparation for defining a complete hydrologic analysis.

Related Topics

- Computing Basin Data
- Delineation with DEMs
- Defining Basins

TIN Options

The TIN Options dialog controls setting for several of the TIN creation and editing functions.



Vertex Options

There are two different modes for creating vertices, digitizing and non-digitizing modes. In digitizing mode all of the options for triangulation of new points, confirmation of z values etc. are disabled and the z values of newly created vertices are determined by the z value entered in the z edit field of the edit window. Newly created vertices are not triangulated into a TIN so triangulation of vertices should be done once digitizing vertices is completed. See creating a TIN from a scanned image for more information on how digitizing mode can be used to create a TIN from a contour map.

In non-digitizing mode there are several options that control what happens to vertices and triangles of a TIN when new vertices are created:

- If the Retriangulate after deleting is checked, the region surrounding the vertex will be retriangulated as each vertex is deleted. Otherwise, the triangles adjacent to the vertex are simply deleted.
- If the check box entitled Adjust boundary to include exterior vertices is selected, the boundary of the TIN will be changed so that the new vertex becomes part of the TIN if a new point is added outside the active TIN. If the new vertex is in the interior of the active TIN, the vertex will be automatically incorporated into the TIN.
- If the check box entitled Confirm z values item is selected, WMS will prompt for a z value every time a new vertex is created.
- If the Interpolate for default z on interior item is checked and a new vertex is entered in the interior of a TIN, the program will linearly interpolate a default z-value from the plane equation defined by the triangle containing the point.

- The default z value edit box displays the z value that will be assigned all subsequent new vertices created with the Create Vertex tool if the Confirm z values check box is not selected.
- If the Extrapolate for default z on exterior item is checked and a new vertex is entered outside the TIN boundary, the program will extrapolate a default z-value by using a gradient based inverse distance weighted interpolation.
- The duplicate vertex tolerance edit box shows the tolerance used for such TIN operations as removing duplicate vertices, insertion of breaklines, and dividing drainage boundaries. You should not need to change this value, but occasionally it becomes the only way to work around sticky numerical problems.
- The maximum filter angle is used when thinning dense vertex lists for more reasonable management. This is particularly useful when processing TINs generated from LIDAR technology.

Triangulation Optimization

There are two different ways of generating an initial triangulation of the xyz vertices for a TIN.

- Angle optimization prioritizes the creation of triangles with as near equal angles as possible.
- Area optimization prioritizes the creation of triangles with as near equal areas as possible.

Triangle Options

The triangle options are used when searching for thin boundary triangles to eliminate after performing an automated triangulation.

Breakline Options

When enforcing breaklines with feature arcs, elevations can be linearly interpolated from the TIN, or derived from the z-values of the feature vertices (of course it is important that z-values are defined for the vertices if this option is selected).

Related Topics

- Removing Duplicates
- Triangulating
- Creating a TIN from a Scanned Image
- Creating Vertices
- Breaklines
- Selecting Boundary Triangles
- Triangulation Optimization
- TIN Interpolation

Outlets

An outlet, by definition in WMS, is a point that defines a confluence. It is the point where a sub basin ends and a routing reach begins. Default outlets are local minima or pits on the interior and stream exit points on the exterior of the TIN. Whenever an outlet point is added, a drainage basin for that outlet is also created. If the outlet corresponds to a stream branching point, then a drainage basin for each stream branch is created. For this reason the default outlets are not always sufficient. Outlet points can be added and deleted in order to define the sub-basins of a watershed.

Converting TIN Vertices/Nodes to Outlets

The **Node ↔ Outlets** command will add all selected vertices as outlet points if they are not already outlet points.

The **Node ↔ Outlets** command will remove selected outlet points if they are already defined as outlets. Outlet points are selected using the Select Vertices tool. When an outlet is deleted, the area or triangles associated with that outlet's drainage basins are reassigned to the next downstream basin.

Related Topics

- Defining Basins
- Streams
- Reservoirs

Drawing Flow Patterns

The **Draw Flow Patterns** command is used to draw a flow path for each triangle in the TIN. While it does not store stream networks and basin boundaries in memory, it aids in the initial understanding of the terrain model and helps identify regions which need editing before the actual creation of outlets, stream networks, and drainage basins takes place. For large TINs, the number of triangles which have their flow paths drawn can be reduced by changing the number of triangle/flowpath value in the Drainage tab of the Display Options dialog. Overland and channel flow are represented by the downhill overland color and downhill channel color as specified in the Drainage tab of the Display Options dialog. If a basin is selected prior to issuing the command then flow paths will only be drawn for the triangles that are part of the selected basin.

Related Topics:

- Defining Basins
- Draw Flow Path Tool

Vertices

Creating Vertices

New vertices can be created by selecting the Create Vertices tool from the Tool Palette and clicking in the Graphics Window where the new vertex is to be located. The x and y values of the vertex are determined by the position of the mouse cursor when a click is made. The z value must be entered separately. A default z value and other parameters governing the creation of new vertices can be set by selecting the **TIN Options** command from the **TIN** menu.

Deleting Vertices

Vertices can be deleted by selecting the vertex/vertices to be deleted and hitting the DELETE or BACKSPACE key on the keyboard or by selecting the **Delete** command from the **Edit** menu. If the Confirm deletions option is active, you will be prompted to confirm each deletion. This is helpful in preventing accidental deletions. The Confirm deletions flag can be toggled by selecting the Confirm Deletions item from the General tab of the **Preferences** dialog of the **Edit** menu.

Delete Duplicates

The triangulation algorithm assumes that each of the vertices being triangulated are unique in the xy plane, i.e. no two vertices have the same xy location. When a new set of vertices is imported to WMS, duplicate vertices should be removed by selecting **Vertices | Delete Duplicates** from the *TIN* menu. Otherwise, WMS may abort when the vertices are triangulated. The tolerance for duplicate vertices can be set in the *TIN Options* dialog.

Locked/Unlocked Vertices

Since it is possible to accidentally drag points, selected vertices can be "locked" to prevent them from being dragged or edited (using the Edit Window) by selecting the **Vertices | Lock/Unlock** command from the **TIN** menu. Any number of combinations of vertices can be locked or unlocked.

Locking and unlocking vertices provides a differentiation between points that are hard (measured data) and points that may be soft (interpolated or estimated data).

Selected vertices can be unlocked by selecting the **Vertices | Lock/Unlock** command from the **TIN** menu. The status of each vertex, locked or unlocked, is preserved in the TIN file when TINs are saved to disk. Display options can be changed so that a distinction between unlocked and locked vertices is easily visible.

Smoothing Pits

The **Vertices | Smooth Pits** command of the **TIN** menu adjusts the elevations of pits in order to remove them. For each pit the two next highest (elevations) of adjacent vertices are located and the elevation of the pit is set to the average of these two elevations. Flat triangles should be removed before using this command.

Transform TINs

The **Vertices | Transform** command is used to move TIN vertices according to a specified transformation. It is similar to coordinate conversion, but should not be used in place of it. Rather it is most useful if you need to transform a set of vertices that were not originally defined in a standard coordinate system so that they align with a standard coordinate system.

When this command is executed, the Transform dialog opens, in which the transformation type and appropriate parameters can be entered. The following transformation types are available:

Scale

Scaling factors for the x, y, and/or z directions are entered. To prevent scaling a specific direction, the default value of 1.0 should be used.

Translate

Translation values for the x, y, and/or z directions are entered. To prevent translation in a specific direction, the default value of 0.0 should be used.

Datum Conversion

Datum conversion can be thought of as converting depths to elevations or elevations to depths. A constant water surface elevation (WSE) value is defined for the conversion process. Note that this is different than simply translating or scaling the z-coordinate. This transformation is governed by the following simple equation:

new z = WSE - old z

Rotate

When rotation is selected, the set of options on the right side of the dialog become available to define the center of rotation. If the Specified Point option is used, then the x- and y- coordinates of the center of rotation needs to be entered. Otherwise, you must click in the graphics window at the Point or on the Node about which the rotation should occur (this is done after clicking the OK button from the Nodes Transform dialog). The rotation is computed counter-clockwise by the angle around the center of rotation.

Related Topics

- Edit Window
- Editing XYZ Coordinates
- Selecting Objects
- Delete
- Trimming TINs
- Filtering TINs
- Creating a TIN from a Scanned Image
- TIN Options
- Coordinate Conversions

Smooth Dataset

Sometimes, you may want to smooth elevation data or another dataset associated with a TIN. This may be because you want to remove flat areas or other anomalies in your elevation dataset. When you smooth a dataset on the TIN, WMS runs a single iteration with a maximum change in elevation of 0.5 units and a filter ratio of 0.8. See the help for smoothing DEMs for more information about these variables.

There is also an option to smooth pits. This function automatically swaps edges and and adjusts coordinates to remove low spots on your TIN.

Related Topics

- Smoothing DEMs
- Smoothing Pits

5.3. Terrain Data Module

Terrain Data Module

The Terrain Data module is where you can import, edit, and prepare digital terrain data (DEMs, and TINs) for hydrologic and hydraulic modeling. Several tools for thinning, smoothing, clipping, and editing the data are available.

Flood plain delineation using scattered data sets and TINs is also handled from the terrain data module.

Specific tools used in the terrain data module can be viewed in the Terrain Data Tools article.

Related Topics

- DEMs
- TINs
- Floodplain Delineation
- Terrain Data Tools
- Project Explorer Contents for Terrain Data Module
- Scattered Data
- Drainage Delineation

Terrain Data Tools

The following feature object tools are in the dynamic portion of the *Tool Palette* when the Terrain Data module is activated.

Select Vertices

The **Select Vertices** tool is used to select vertices for operations such as deletion, or to drag a vertex to a new location. The coordinates of selected vertices can also be edited using the *Edit Window*. This same tool allows for selection of outlet points on the TIN.

Select Triangles

The **Select Triangles** tool is used to select triangles for operations such as deletion. In addition to the standard multi-selection options, another type of multi-selection is available with this tool. By holding down the *CTRL* key while dragging the cursor, a selection line can be entered. All triangles intersected by the line are added to the selection list.

Create Breakline

The **Create Breakline** tool is used to select one or more strings of vertices. Vertex strings are used for operations such as adding breaklines to the TIN or selecting a string of vertices which will be used to create a stream.

The procedure for creating breaklines is somewhat different than the normal selection procedure. Strings are selected as follows:

- Click on the starting vertex for the breakline. The vertex selected will be highlighted in red.
- Click on any subsequent vertices you would like to be part of the string (vertices do not have to be next to each other). The vertices selected are now connected by a solid red line.
- To remove the last vertex from a string, press the *BACKSPACE* key. To abort entering a vertex string, press the *ESC* key. To end a vertex string, press *RETURN* or double click on the last vertex in the string. Another vertex string can then be selected.

Swap Triangle Edges

The **Swap Triangle Edges** tool swaps the edges of two adjacent triangles and is usually used to make local adjustments to a TIN. To use the tool, simply click on any triangle edge.

Add Vertices

The **Add Vertices** tool is used to manually add vertices to a TIN. It can only be used in plan view. When this tool is selected, clicking on a point within the Graphics Window will place a new vertex at that point. What happens to the vertex after it is added (whether and how it is triangulated into the TIN) depends on the settings in the *Vertex Options* dialog under the *TINs* menu. These settings can easily be used to digitize elevation data from scanned images of contour data (see Vertex Options).

Create Triangle

The **Create Triangle** tool is used to manually create new triangles. Triangles are normally created by triangulating a set of points automatically. However, this tool is useful for localized editing and refining a TIN. To use the **Create Triangle** tool you may either:

- Select three vertices that will form the triangle. The vertices can be selected in either clockwise or counter-clockwise order.
- Drag a box around three vertices that will form the triangle.

The *ESC* key can be used to abort the creation of a triangle once you have started selecting vertices.

Select DEM Points

The **Select DEM Points** tool is used to select a region of DEM points to make active or inactive. Coordinates of DEM points may not be edited. When selecting a group of DEM points a rectangle or polygon around the points is displayed rather than trying to identify individual DEM points. To deselect a group of DEM points that have already been selected, click anywhere in the graphics window. Multiple groups of DEM points may be selected by holding down the *SHIFT* key while dragging a rectangle around the second group.

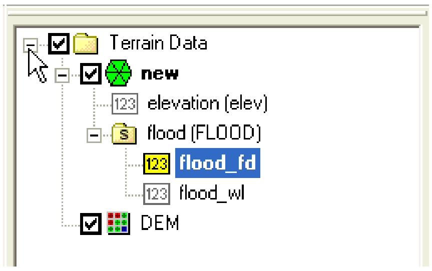
The Feature Object and Drawing Tools are also available from the terrain data module.

Related Topics

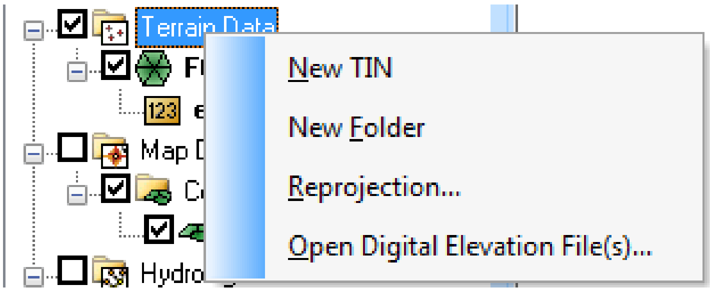
Tool Palettes

Project Explorer Contents for Terrain Data Module

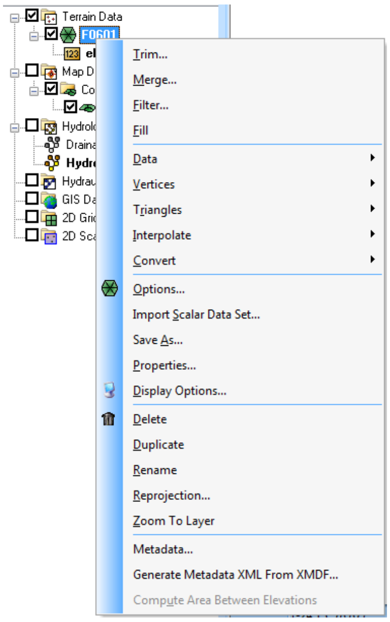
In the Terrain Data module, TINs and DEMs are listed in the *Project Explorer*. A toggle is next to each object; the toggle controls the visibility. The display is automatically updated when the toggle is checked or unchecked.



Right-clicking on the main Terrain Data folder allows you to create a **New folder** or **TIN**. Because TINs can be created by digitizing in WMS you can create a new, blank TIN. However, DEMs are only created by opening a DEM file type. If neither a TIN nor a DEM exists, then you can also perform a **Coordinate Conversion** on any existing terrain data. There is also an option to open a digital elevation file. Opening a digital elevation file using this option will open an elevation grid as a WMS DEM.

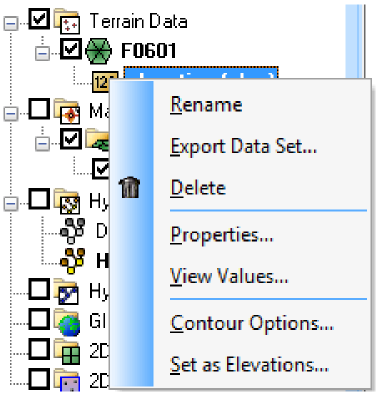


Right-clicking on a TIN allows you to select any of the TIN processing or visualization options in WMS.

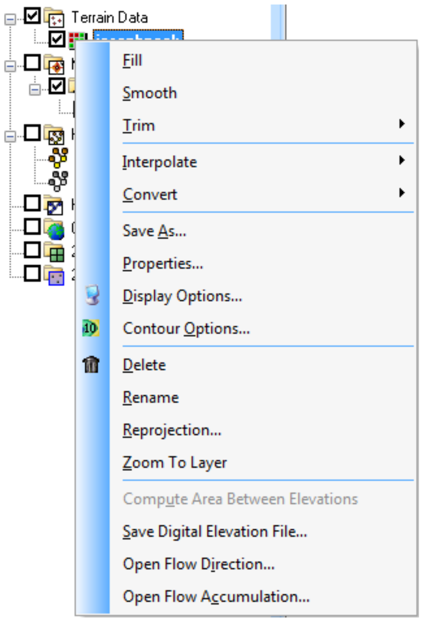


A TIN by default has a single elevation dataset, but additional datasets can be created, such as flood depth and water surface elevation datasets when delineating a flood plain. For more information on datasets see Datasets.

Right clicking on a dataset of a TIN allows you to **Delete**, **Export**, **Rename**, view **Properties**, **View values**, set the dataset as elevations, or set the dataset contour options.



Right clicking on a DEM allows you to select any of the DEM processing or visualization options in WMS. All DEMs are created by either reading a file or converting from another data object, such as a TIN or from raster GIS data. If multiple DEMs exist, these DEMs can be selected and merged together into a single DEM. There is only one active DEM. This active DEM is used for watershed delineation and for all other processes that require elevation data in WMS.



Related Topics

- Project Explorer Overview
- Coordinate Conversions

5.4. Drainage Module

Drainage Module

A primary use of WMS is to automatically delineate watershed, stream, and sub-basin boundaries from digital elevation sources such as TINs and DEMs. The drainage module includes all of the commands necessary to perform the automated delineations in preparation of running one of the hydrologic models.

While delineation can be performed using either DEMs or TINs, it is generally easier to use DEMs. In general the DEM method is simpler and more repeatable because of the uniform nature of a DEM data structure. When inadequate elevation resolution is available (i.e. many urban ares) then the feature object method of delineation should be considered.

Some basic guidelines might be:

1. DEMs are better for larger, rural watersheds
2. TINs might be more appropriate for smaller, urban areas with adequate resolution
3. The feature object method works well if no elevation data exist, or if the resolution of the elevation data is not adequate.

Since a DEM or TIN delineation can be converted to feature objects you may want to consider modifying an initial delineation with an elevation source manually with the feature objects. However, you must realize that some of the parameters computed from the elevation data will be compromised when manual adjustments are made with the feature objects. Be careful and understand what/why you are making modifications.

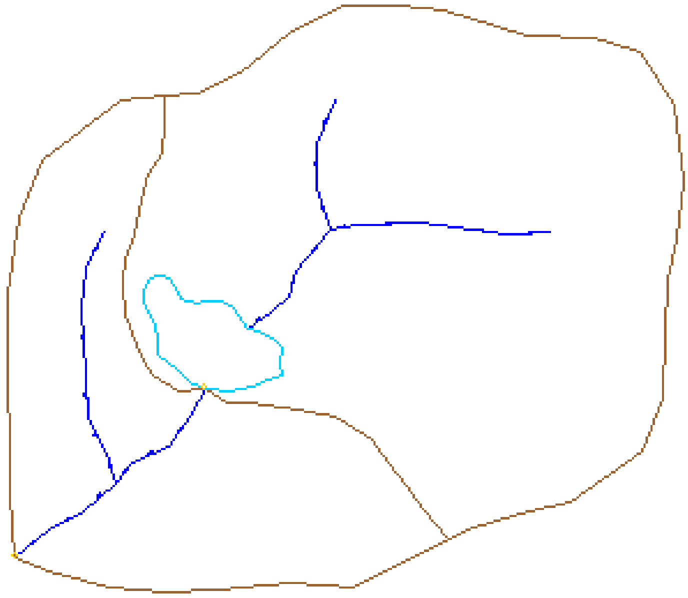
Related Topics

- Watershed Delineation with DEMs
- Watershed Delineation with TINs
- Watershed Delienation with Feature Objects

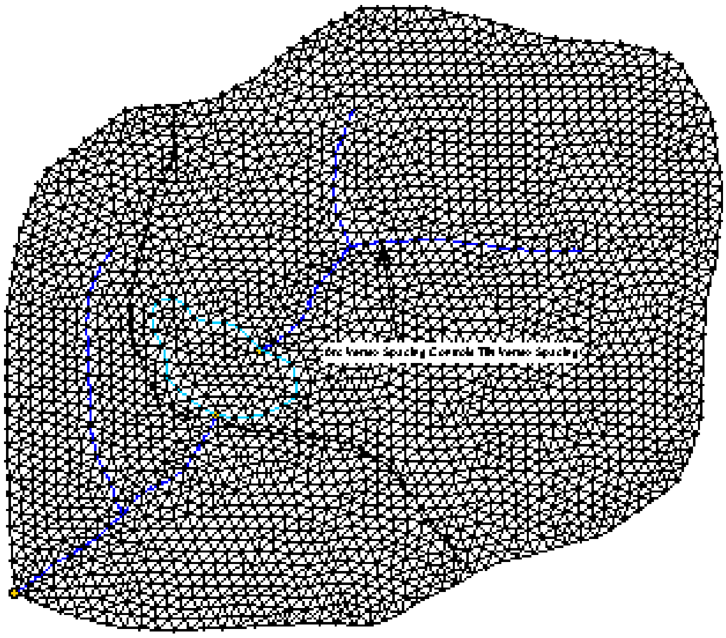
Drainage

The Drainage Coverage is the primary coverage used by WMS. Whenever you start a new WMS session an "empty" coverage is created and assigned the drainage coverage type. If you begin creating points, arcs, and polygons, by default they will belong to this drainage coverage. Most of the work you do will be centered around the drainage coverage which has two different purposes as outlined in the following paragraphs:

1. To develop a hydrologic model directly from feature objects, or GIS vector data. The points, arcs, and polygons in the drainage coverage are tied directly to the hydrologic modeling tree. When a stream arc is created, the most downstream node of the stream arc is converted to an outlet node and an outlet is added on the hydrologic modeling tree. A drainage basin is also added on the tree for every upstream arc from an outlet node. If a drainage polygon is created and a stream is located in the boundaries of that drainage polygon, the drainage polygon is tied into the tree as a drainage basin. An example of a drainage coverage used for the purpose of creating a hydrologic model is shown in the figure below.



2. To use as a "conceptual" model when creating a TIN from a background elevation data source and feature objects for the purpose of automated watershed delineation. WMS can use TINs for performing watershed delineation. However, when triangulating a set of scattered xyz data, it is difficult if not impossible to ensure that triangle edges conform to streams, ridges and other drainage features. Drainage coverage feature objects can be used to create a TIN that conforms to the feature objects. The distribution of TIN vertices will approximate the distribution of feature arc vertices in the conceptual model.



In either case the same set of attributes can be defined, but are used for a different purpose. The following sections outline the different attributes available for points, arcs, and polygons in the drainage coverage and describe the differences when used to create hydrologic models or TINs.

Drainage Point/Node Types

Drainage points can have either generic or outlet attributes. Generic points have no attributes and can be used for drawing purposes. Outlet points are directly linked to the tree. By default, the most downstream node of a stream arc is an outlet point and its attributes cannot be changed. If additional points along a stream are converted to outlet points, the hydrologic modeling tree is updated when these outlet points are added. Furthermore, if outlet points are converted back to generic points, the hydrologic modeling tree is also updated.

When creating a TIN from feature objects, outlet points on the feature objects are converted to outlet points on the TIN. If points are imported but not used by any arcs they can be removed using the **Delete Isolated Points** command.

Drainage Arc Types

Generic arcs have no attributes and are used when developing drainage boundaries or when establishing the boundary polygon for creating a TIN from feature objects.

Stream arcs should be used to define stream reaches hydrologic models. The direction of a stream arc is critical and in WMS, the direction of flow is opposite the direction the arc is created. In other words you should create stream arcs from downstream to upstream (the "from" node being the downstream node and the "to" node being the upstream node). If an arc is designated as a stream, then an outlet point is added at the most downstream node of the stream arc, and the hydrologic modeling tree is updated to include this outlet point. When a stream arc is created, a drainage basin is added on the hydrologic modeling tree for each upstream arc emanating from an outlet point. If the stream arcs are being used to create a TIN, a breakline is forced along the arc (triangle edges are enforced along the arc). Furthermore, a stream along the affected triangle edges is automatically created for the TIN.

Lake arcs should be used to trace around the boundary of a lake. Stream arcs can be attached up and/or downstream of the lake. If no downstream stream arc is attached to the lake then a node on the lake will automatically be defined as the outlet of the watershed and a drainage basin will be created for the lake/stream combination. If a downstream arc is defined from the lake, then the downstream most point on the stream will, by default, be designated as the watershed outlet. However, any node on a lake arc can be assigned an outlet if you want to define a sub-basin outlet at the lake. If a lake polygon exists when using feature objects to create a TIN, then the lake arcs will be enforced in the TIN as breaklines.

Ridge arcs should only be used when creating a TIN from feature objects and are used to designate any other (besides boundary, stream, and lake) linear segment you wish to have enforced as a breakline in the resulting TIN. They have no effect when creating the hydrologic model directly from the feature objects.

Drainage Polygon Types

To link a polygon to the hydrologic modeling tree in the map module, the polygon must be a Drainage boundary polygon type. Also, the polygon must have a stream inside the polygon. Each stream has a drainage basin assigned to it. This drainage basin will also be assigned to the polygon containing the stream, provided the polygon is a Drainage boundary polygon. Because most of the time you will be working want to use drainage polygons WMS uses the following defaults when building polygons in a drainage coverage:

1. If the arcs that enclose a polygonal region contain at least one generic arc type the resulting polygon will be assigned a drainage boundary type.
2. If all the arcs that enclose a polygonal region are lake arcs then the resulting polygon will be a lake polygon.

The generic polygon type is really only used as an intermediate polygon type when importing data from another source.

When creating a TIN, it is only necessary to have a single bounding polygon (of either Generic or Drainage boundary type) to define the extent of the area that will be converted to a TIN. If you have multiple sub-basins, WMS will ignore sub-basin boundary arcs interior to the watershed (you must define the sub-basins boundaries from the TIN).

Related Topics

- Feature Objects
- Coverages
- Feature Object Guidelines

Drainage Data to Feature Objects

In order to provide a way to export watersheds delineated from TIN data sources to GIS they must first be converted to feature objects. This can be done using the **Drainage Data → Feature Objects** command. When performing this operation the TIN is deleted and the drainage boundary and stream network are converted to feature polygons and feature arcs. The **Export** command from the *File* menu can then be used to save the feature objects to a shapefile so that the information can easily be transferred to a GIS. All of the drainage data computed/entered for the TIN basins and junctions (areas, lag times, slopes, and other hydrologic modeling parameters) are automatically transferred during the conversion process.

Related Topics:

- TIN Boundary to Polygon
- TIN to DEM
- TIN to Scatter Points
- TIN Contours to Feature Objects

Drainage Tools

Contour Labels

The **Contour Label** tool manually places numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contouring options are changed, until they are deleted using the *Contour Label Options* dialog, or until the Graphics Window is refreshed. Contour labels can also be deleted with this tool by holding down the *SHIFT* key while clicking on the labels. This tool can only be used when the TIN is in plan view.

Select Drainage Unit or Basin

The **Select Drainage Unit or Basin** tool is used to select basins which can then be either merged together or split. In addition to selecting basins, this tool can be used to select one of the basin icons.

Move Basin Labels

Computed drainage data may be displayed for each basin. However, when there are many basins, the screen can become cluttered with data. The **Move Basin Labels** tool allows basin data to be placed at a position other than the centroid, which is the default location. When moving a label, click in the desired basin and while holding down the mouse button, drag the cursor to the desired position on the screen and then release the button. An arrow will be drawn from the final position to the point first clicked in the basin.

Flow Path

The **Flow Path** tool allows the flow paths for specified points to be drawn. When this tool is active, clicking in the graphics window at a location on a TIN, or a DEM after TOPAZ data are computed will cause a flow path to be initiated from that point and followed "downstream" until a pit or local minima is reached, or until the path leaves the TIN/DEM. This tool can be very useful in checking portions of an edited TIN before stream and basin definition is completed.

The length and slope of overland and stream flow is displayed in the help window each time a new path is drawn. This can be helpful in obtaining parameters used to compute lag times with some empirical formulas. Stream distances are shown only after a stream has been created. In other words, channel flow is not counted in the stream distance unless a "stream" has been created along the channel.

Flow paths initiated from the centroid of each triangle or DEM cell can be displayed using the **Draw Flow Patterns** command in the *TIN* or *DEM* menu within the drainage module.

Other Drainage Tools

The remainder of the drainage tools are also found in the Terrain Data module tools, and the Map module tools.

Note: One difference is that the create feature point tool always creates an outlet point when used in the drainage module.

Project Explorer Contents for Drainage Module

The Drainage Module is used for computing watershed drainage attributes and for delineating watersheds and their sub-basins on TINs and DEMs. When WMS computes watershed drainage attributes or determines flow directions, the active TIN or DEM is used. If you select a TIN or a DEM, the Terrain Data Module is set as the active module. To perform drainage computations on a TIN or a DEM, change the active module to the Drainage Module by selecting the Drainage Module.

In the Project Explorer, the Drainage Module has the same menus and commands as the Terrain Data module.

Right-clicking on the main Drainage Module folder allows you to create a **New folder** or **TIN**. Because TINs can be created by digitizing in WMS you can create a new, blank TIN. However, DEMs are only created by opening a DEM file type. There is also an option to open a digital elevation file. Opening a digital elevation file using this option will open an elevation grid as a WMS DEM. Right-clicking on a TIN allows you to select any of the TIN processing or visualization options in WMS.

A TIN by default has a single elevation dataset, but additional datasets can be created, such as flood depth and water surface elevation datasets when delineating a flood plain. For more information on datasets see Datasets.

Right clicking on a dataset of a TIN allows you to **Delete**, **Export**, **Rename**, view **Properties**, **View values**, set the dataset as elevations, or set the dataset contour options.

Right clicking on a DEM allows you to select any of the DEM processing or visualization options in WMS. All DEMs are created by either reading a file or converting from another data object, such as a TIN or from raster GIS data. If multiple DEMs exist, these DEMs can be selected and merged together into a single DEM. There is only one active DEM. This active DEM is used for watershed delineation and for all other processes that require elevation data in WMS.

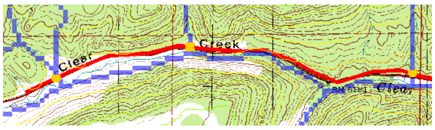
Refer to the Terrain Data Module information page for more information about managing TINs and DEMs and their attributes in WMS.

Related Topics

- Project Explorer Overview
- Project Explorer for Terrain Module

Arcs to Outlets

The **Arcs→Outlets** command in the *DEM* menu of the Drainage module will automatically place a new outlet feature point at all intersections of a selected arc (required input to the command) and the specified flow accumulation value. The default flow accumulation threshold will correspond to the current display options setting and represents the area upstream. This is particularly useful if you wish to establish outlets for a series of drainages that a highway or other important line feature will cross as illustrated in the figure below.



Outlets created with the **Arcs→Outlets** command

The arc should exist in a drainage coverage or the nodes created by the intersection will be generic nodes rather than drainage outlets.

Related Topics

- Drainage Coverage
- Nodes to Outlets
- Watershed Delineation with DEMs

5.5. Map Module

Map Module

The Map module provides a suite of tools for defining watershed data in a GIS and then using the information to directly create and manage hydrologic and hydraulic models, or as a support utility for data development with either TINs or DEMs. Results of watershed and floodplain delineations can also be saved in the map module and converted to GIS data layers for export.

Land Use and Soil type layers can be created using feature objects in the Map module and then used to compute curve numbers or map other important modeling parameters. Other layers are also used for computing time of concentration or lag time, cutting cross sections, mapping NFF regions, mapping rainfall and other parameters for the LA County modified rational (MODRAT) model, and streams for 2D analysis using GSSHA.

A rough boundary and stream network can also be used to generate a TIN or two-dimensional finite difference grid that conforms precisely to streams and other important hydrologic features. Feature objects can also be used to create polygonal boundaries of soil type or land use to aid in the computation of curve numbers for hydrologic analysis.

Images can be used to provide a background display of a region, or draped over a TIN or grid as a texture map. They can be imported from TIFF files or created from within WMS by capturing the screen.

Within the Map module there are several other tools which can be helpful in either setting up models or presentation of results to a client. Tools for reading and writing of CAD files, and text annotation are part of this module.

Related Topics

- Drawing Objects
- Feature Objects
- Images
- CAD Data
- Data Acquisition

Map Properties

The **Map Properties** dialog is used when ArcObjects is enabled to specify the coordinate system that you wish to display/map features from the ArcGIS data layer. An ArcGIS data layer should have a currently defined coordinate system associated with it. If the coordinate system is geographic (latitude/longitude), then ArcObjects is able to "guess" correctly at the projection. Using the coordinate system as defined in the Map Properties you can specify the coordinate system you wish to use to display features/rasters in WMS. While this does not change the actual geometry of the layer, it will display in the WMS window according to this projection and any data mapped to WMS coverages will be mapped into the coordinate system specified by the Map Properties.

Related Topics:

- Coordinate Systems of GIS Layers
- Mapping to Feature Objects

Map Tools

The following feature object tools are in the dynamic portion of the Tool Palette when the Map module is activated.

Select Feature Objects

The **Select Feature Objects** is an intelligent tool that can be used in place of many other Select tools. It can be used as a sort of generic selection tool to select arcs, polygons, points, etc. without having to change the tool. This tool can be used to select via a single-click, a double-click, or by dragging a selection box.

Select Feature Points/Nodes

The **Select Feature Points/Nodes** tool is used to select existing points or nodes. A selected point/node can be deleted, moved to a new location, or operated on by one of the commands in the Feature Objects menu. The coordinates of selected points/nodes can also be edited using the Edit Window. Double clicking on a point or node with this tool brings up the Point or Node Attribute dialog.

Select Feature Vertex

The **Select Feature Vertex** tool is used to select vertices on an arc. Once selected, a vertex can be deleted, moved to a new location, or operated on by one of the commands in the Feature Objects menu. The coordinates of selected vertex can also be edited using the Edit Window

Select Feature Arc

The **Select Feature Arc** tool is used to select arcs for operations such as deletion, redistribution of vertices, or building polygons. Double clicking on an arc with this tool brings up the Arc Attribute dialog.

Create Feature Point

The **Create Feature Point** tool is used to interactively create new points using the cursor. A new point is created for each location the cursor is clicked on in the Graphics Window. A background drawing grid can be turned on using the Grid Options command in the Display menu to aid in the placement of points.

Create Feature Vertex

The **Create Feature Vertex** tool is used to interactively create new vertices along an existing arc, to add more detail. A new vertex is created for each location the cursor is clicked on in the Graphics Window, that it is within 10 screen pixels of an existing arc. Once the vertex is created, it can be repositioned with the Select Vertex tool.

Create Feature Arc

The **Create Feature Arc** tool is used to interactively create new arcs. An arc is created by clicking once on the location where the arc is to begin, clicking once to define the location of each of the vertices in the interior of the arc, and double-clicking at the location of the end node of the arc.

As arcs are created, it is often necessary for the beginning or ending node of the arc to coincide with an existing node. If you click on an existing node (within a given pixel tolerance) when beginning or ending an arc, that node is used to define the arc node as opposed to creating a new node. Also, if you click on a vertex of another arc while creating an arc, that vertex is converted to a node and the node is used in the new arc. If an existing point is clicked on while creating an arc, the point is converted to a vertex, unless it is the beginning or ending location of an arc, in which case it is converted to a node.

While creating an arc, it is not uncommon to make a mistake by clicking on the wrong location. In such cases, hitting the BACKSPACE key backs up the arc by one vertex. The ESCAPE key can also be used to abort the entire arc creation process at any time.

The new arc type is determined from the Feature Arc Type dialog, accessed from the Attributes command in the Feature Objects menu. The Feature Object Type dialog that comes up when selecting the Attributes command is determined by the currently active tool. For example if the Select Arc or Create Arc tools are active the Feature Arc Type dialog comes up, whereas if the Select Polygon tool is active the Feature Polygon Type dialog comes up.

When creating stream arcs, the points/vertices must be connected from downstream to upstream as the arc is entered. Stream arcs are drawn with an arrow pointing in the downstream direction and can be reversed after creation using the Reverse Directions command.

Select Feature Polygon

The **Select Feature Polygon** tool is used to select previously created polygons for operations such as deletion, assigning attributes, etc. A polygon is selected by clicking anywhere in the interior of the polygon. Double-clicking on a polygon with this tool brings up the Polygon Attributes dialog.

Select Feature Line Branch

The **Select Feature Line Branch** tool can be used to automatically select all arcs of a branch without having to select each individually. This is particularly useful when locally redistributing vertices along a stream branch. A branch is selected by selecting any arc in the branch. WMS searches down the stream from the selected arc until the next branching node is encountered, and then adds all arcs upstream from that node to the list of selected arcs. While it is intended that it be used for a set of arcs that represent a stream network, in fact it will work for any set of connected arcs.

Related Topics

- Tool Palettes
- Map Module

Mapping Tables

The mapping table is different depending on whether composite curve numbers or runoff coefficients are being computed. A mapping table must be imported using the **Import** button prior to computing the composite curve numbers. The radio group above the mapping window specifies whether the land use or soil type mapping table is displayed. For a land use or soil type coverage the mapping table may be set up interactively by selecting each polygon and assigning the appropriate parameters. Such files can then be exported and later imported. For grid attributes, you can define a land use or soil type table by defining a land use or soil type coverage and then using the **Attributes** command from the **Feature Objects** menu (in the Map module) to define individual polygon attributes that constitute the table. See Land Use Coverages and Soil Type Coverages for information on assigning parameters to land use and soil type polygon coverages. See File Formats for the different mapping parameters.

Composite CN Table

Because land use tables with corresponding CNs vary from text to text and agency to agency, WMS supports a user-definable method for relating land use to CNs. This is done through a simple table file that is imported prior to computing the composite CN. The file scsland.tbl is an example of such a file and was created from a table given in the Handbook of Hydrology. This file can be edited to supply your own values or a new table with the same format can be created. See Land Use Files to view the format of the table.

Runoff Coefficient Table

The runoff coefficient table must be defined to relate soil type IDs to runoff coefficient values. A very simplified file named soiltype.tbl is provided with the distribution of WMS. This file can be edited to supply your own values or a new table with the same format can be created. See Soil Type Runoff Coefficient Files to view the format of the table.

Related Topics

- Compute GIS Attributes
- Land Use
- Soil Type

Conceptual Model

A conceptual model is a map-based representation of your hydrologic or hydraulic model using map module objects (feature objects) such as points, nodes, arcs, and polygons. To define and compute lumped-parameter model data, WMS normally converts a map module representation of your model to a schematic or tree-based representation of your model in either the hydrologic modeling or river module. If you are creating a conceptual model of *hydrologic* data (basins, streams, and outlet points), the schematic representation is created automatically. If you are creating a conceptual model of *hydraulic* data, the schematic representation is created by selecting the Map→1D Schematic menu item available in the special menu associated with the hydraulic modeling coverage (This command is available in 1D Hydraulic Schematic and Storm Drain coverages). After selecting the **Map→1D Schematic** command, cross sections, reaches, links, and/or nodes are displayed as symbols in the river module and their properties can be edited in that module.

As an example, a SWMM hydraulic model with arcs representing links and nodes in a SWMM model and the symbols corresponding to links and nodes in the hydraulic schematic is shown below:

Mapping to Feature Objects

While future versions of WMS may be able to process some commands directly from the GIS data layers, currently you must map all features you wish to use as part of model development to feature objects in a map coverage. One way to do this is to import an entire shapefile directly to map coverage (this is the only way available in previous versions), but often the extents of the GIS data layer are much larger (i.e. an entire state) and so it may be more efficient to select only those GIS features (points, lines, polygons) that overlay your study area and map those to feature objects in a map coverage.

A mapping wizard guides you through the process of converting your GIS data layer features to feature objects in a map coverage. Before beginning the mapping process you should first go to the map module and make sure that the currently active coverage is the coverage you wish to map GIS data layer features to. For example, the default coverage in WMS is a drainage coverage and so if you are about to map soil polygons you will want to create a new coverage and make sure that it has an attribute set of Soil Type. After making sure you will be mapping to the correct coverage select the polygons which overlay your study area and you wish to map (this is done with the selection tool(s) in the GIS module). If you wish to map all features, you can choose the **Select All** command from the **Edit Window**, or just move to the **Mapping** command (you will be prompted if you want to convert all features since none are selected).

If you have ArcObjects enabled, you will see that the **ArcObjects->Feature Objects** command is activated, whereas if you do not have ArcObjects enabled, you will see that the **Shapes->Feature Objects** command is activated. After choosing the appropriate mapping command you will see the mapping wizard shown below. This wizard will guide you through the rest of the process. The first dialog in the mapping wizard contains instructions and marks the beginning point of mapping for selected features. The first of two steps is to map the attribute fields of the features to attributes used by WMS. WMS recognizes some attribute names as commonly used for certain attributes and maps them automatically (i.e. HYDRGRP for hydrologic soil groups and LU_CODE for land use ID's).

The second step marks the end of the wizard, and after selecting Finish, all selected features will be converted to feature objects within the active coverage. Attributes of mapped fields will be saved accordingly as attributes of the feature objects.

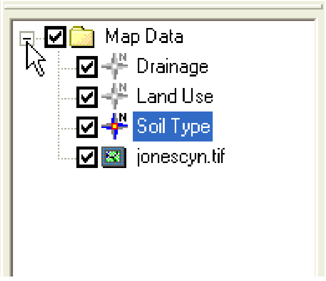
Related Topics

- Feature Objects
- Project Explorer
- Importing Shapefiles

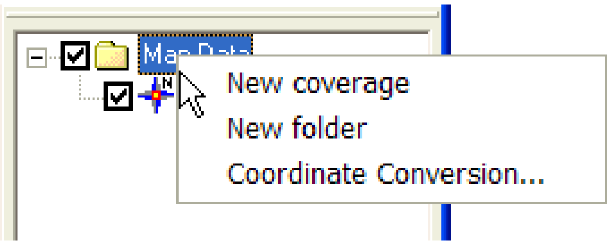
Project Explorer Contents for Map Module

In the Map Module the coverages and images are listed in the Project Explorer. At this point CAD data are not supported in the Project Explorer, but it is anticipated that in future versions the CAD data will also be controlled from the Project Explorer.

A toggle is next to each object; the toggle controls the visibility. The display is automatically updated when the toggle is checked or unchecked. The active coverage is set in the Project Explorer by selecting it. The active coverage is indicated in the Project Explorer display with a color map module icon, while the inactive coverages are gray.

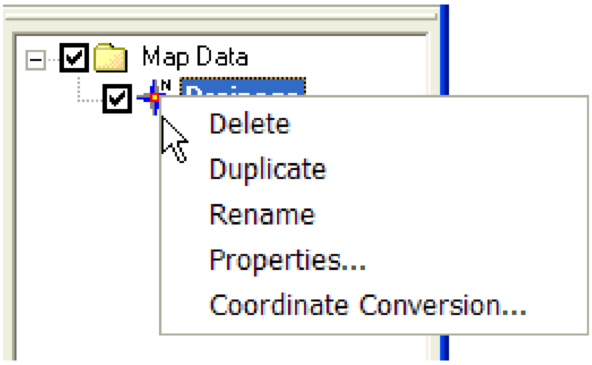


Right clicking on the Map Data folder allows you to create a **New coverage** or **New folder** to organize coverages in, or perform **Coordinate Conversions** on all of the coverages.

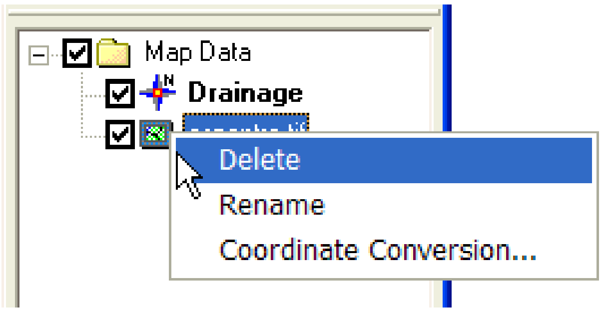


The Project Explorer replaces the Coverage Options dialog in previous versions of WMS. Click here to learn about coverages in the Project Explorer.

Right clicking on a coverage allows you to **Delete**, **Duplicate**, **Rename**, set/view the **Properties** of the coverage, or perform a **Coordinate Conversion**. Properties include the name and coverage type.



Right clicking on an image allows you to Delete, Rename, or perform a Coordinate Conversion (all images are created only by opening an image file).



Related Topics

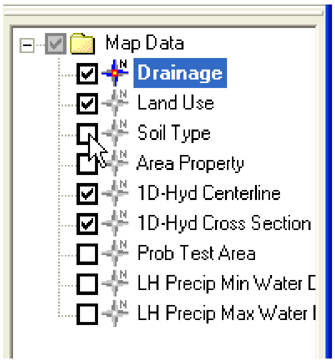
- Project Explorer Overview
- Coordinate Conversions

5.5.a. Coverages

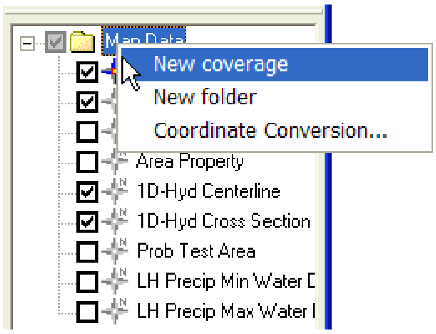
Coverages

Feature objects in the Map module are grouped into coverages. A coverage is similar to a layer or level in a CAD drawing, or a theme or layer in ArcView®. Each coverage represents a particular set of information. For example, one coverage could be used to define drainage boundaries and another coverage could be used to define land use or soil zones. These objects could not be included in a single coverage since polygons within a coverage are not allowed to overlap.

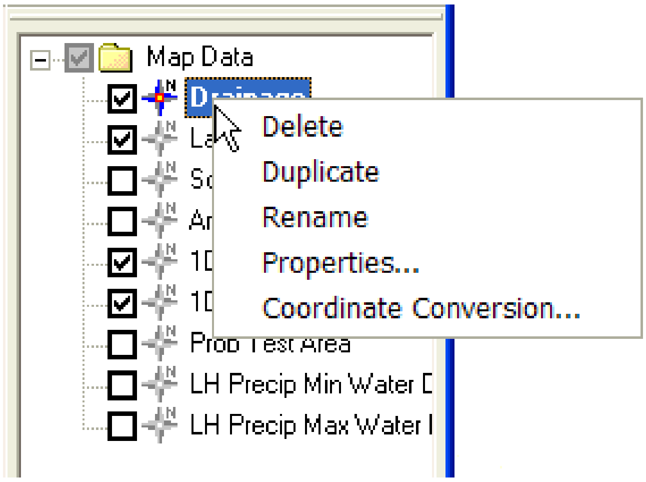
Coverages are managed using the *Project Explorer*. Coverages can be organized into folders and moved once they are created. When WMS is first launched the default coverage is a drainage coverage. When multiple coverages are created, one coverage is designated the "active" coverage. New feature objects are always added to the active coverage and only objects in the active coverage can be edited. The figure below shows several coverages in the *Project Explorer*. The active coverage is displayed with a color icon and bold text. A coverage is made the active coverage by selecting it from the *Project Explorer*. In some cases it is useful to hide some or all of the coverages. The visibility of a coverage is controlled using the check box next to the coverage in the *Project Explorer*.



A new coverage can be created by right-clicking on a folder and selecting the **New Coverage** command in the pop-up menu.



Right clicking on a coverage bring up a menu with the following options: Delete, Duplicate, Rename, and Properties. The **Delete**, **Duplicate**, **Rename**, **Properties**, and **Coordinate Conversion** commands are self explanatory.



The **Properties** command brings up the Coverage Properties dialog. The coverage properties dialog allows you to specify what the coverage type is, its default elevation (mostly used for displaying and overlaying) and change the name (although the name of the coverage can also be changed from the Project Explorer in the same fashion as changing a name in a typical Windows Explorer.

Coverage Types

Each coverage is assigned a coverage type that controls which set of attributes are associated with the coverage. The appropriate type for a coverage depends on the intended use of the coverage. The available types are as follows:

- 1D-HYD Centerline
- 1D-HYD Cross-section
- Area Property
- CE-QUAL-W2 Branch
- CE-QUAL-W2 Observations (Unused-can be used as a generic coverage)
- CE-QUAL-W2 Segments
- Drainage
- Flood Barrier
- **Flood Extent** – A flood extent coverage is created from a flood delineation dataset and contains polygons showing inundated limits or differences in inundation limits between two different flood plain delineation scenarios. See Flood Extent for more information.
- General
- Land Use
- **MODRAT DPA Zone** – This coverage type is used to delineate the Debris Production Area used in a MODRAT simulation.
- MODRAT Tc
- NSS Region
- Rain Gage
- **Rainfall Zone** – The rainfall zone coverage is used to map MODRAT rainfall zones for the purpose of developing a MODRAT simulations. These zones correspond to the lettered zones (A-K) and are not a part of the aerial distributed rainfall grid simulations.
- **Runoff Coefficient** – The Runoff Coefficient coverage is used in the same was as the land use coverage, except that rather than defining a land use type a floating point runoff coefficient can be entered for each polygon. Composite runoff coefficients (C values for the rational method or CN values for the SCS method) can then be computed using the **Compute GIS Attributes** command in the *Calculators* menu of the Hydrologic Modeling module.
- Soil

- Storm Drain
- Time Computation

Related Topics

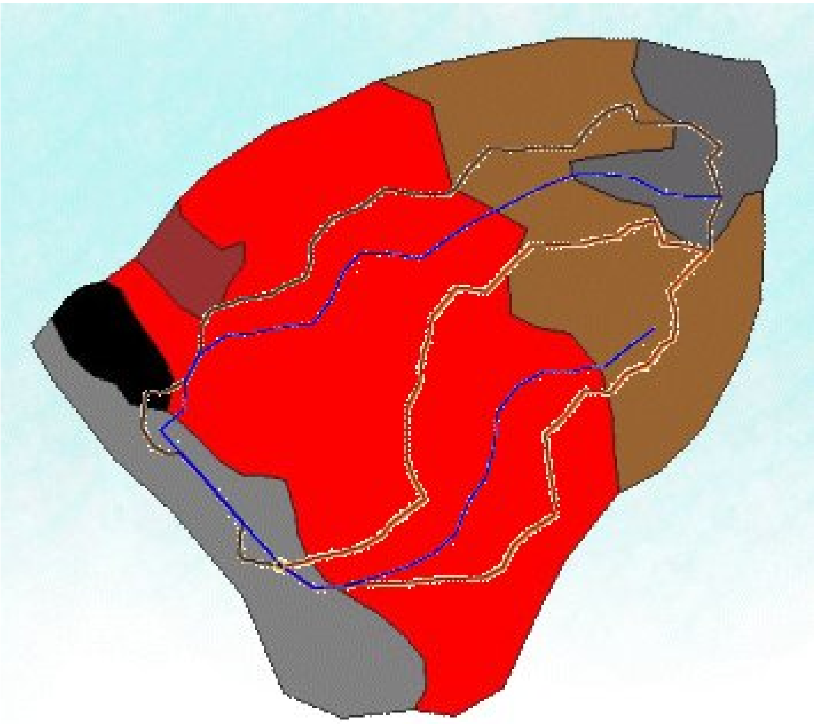
- Feature Objects

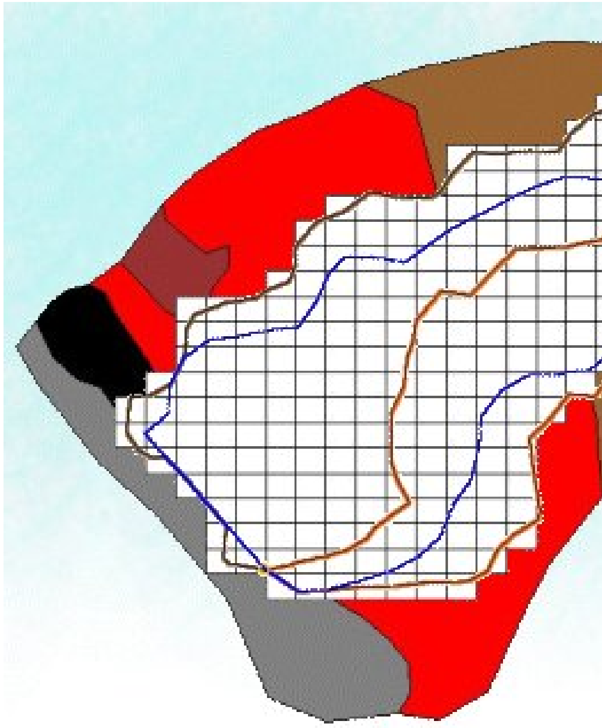
Coverage Overlays

Overlaying two coverages is one of the most basic and powerful GIS operations for manipulating spatial data. WMS is a geographic information system (GIS) for hydrologic modeling, and to the extent that it performs "GIS" operations like overlay, it is used to establish specific hydrologic parameters like curve numbers (derived by overlaying a land use and soil coverage with the drainage coverage). However, there are a number of other useful overlays that may apply to hydrologic modeling that are not programmed into WMS. The **Compute Coverage Overlay** command in the **Calculators** menu allows you to overlay any two polygon coverages (such as land use and drainage boundaries) and derive information from them. For example you may want to know the percentage of a certain land use type (i.e. lakes, forest, pavement, etc.) covering your basins. These values are often needed for computations using the National Streamflow Statistics (NSS) program of regional regression equations.

Overlay areas (or percentages) are computed from the **Compute Coverage Overlay** dialog by first specifying the two coverages. The Input coverage is the coverage you are overlaying and want to determine values for. This is typically the drainage coverage because we usually want to know the area of land use or soil "overlaying" the basins and not the area of basins overlaying soils. You then specify the overlay coverage. If you wish to only use the selected polygons from the input coverage then check the toggle box.

WMS does not perform a true vector overlay where lines from one coverage are intersected with the other and then a new set of polygons created. Instead a temporary or moving grid is placed over the two coverages and then each grid cell is sampled to see which polygons from the input and overlay coverages that it lies in as illustrated in the three figures below. The smaller the computation step the more accurate, but more time consuming the overlay calculations will be. WMS tries to estimate a reasonable size based on the overall area covered by the coverages, but you may need to adjust the computation step at times.





The Compute Overlay button performs the overlay calculations and the summary report (according to the options) is given in the text window of the dialog.

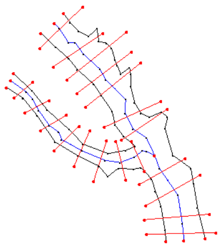
Related Topics

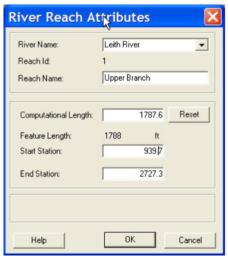
- Hydrologic/Hydraulic Calculators
- Compute GIS Attributes
- NSS

1D-HYD Centerline

The 1D-Hydraulic Centerline coverage has three possible attribute types: general, centerline, and bank. If the arc is a general arc type then it does not participate in the building of a hydraulic centerline and only provides additional visual detail to the model. A bank arc is used to mark left and right bank points for any cross sections that are automatically extracted from at digital terrain model.

A centerline arc provides the backbone of the hydraulic model definition. It's direction should be from upstream to downstream as this is the way HEC-RAS commonly views the river. This automatically defines which is the left bank and which is the right bank (think of standing up river and looking downstream when determining left and right). A centerline has as attributes the river reach properties as defined in the dialog shown below. The river reach properties include: a River Name, a Reach ID (internally assigned and not editable), a Reach Name, a Computational Length for the river (generally equal to the length but this could be different in order to account for additional sinuosity), and a start and end station.



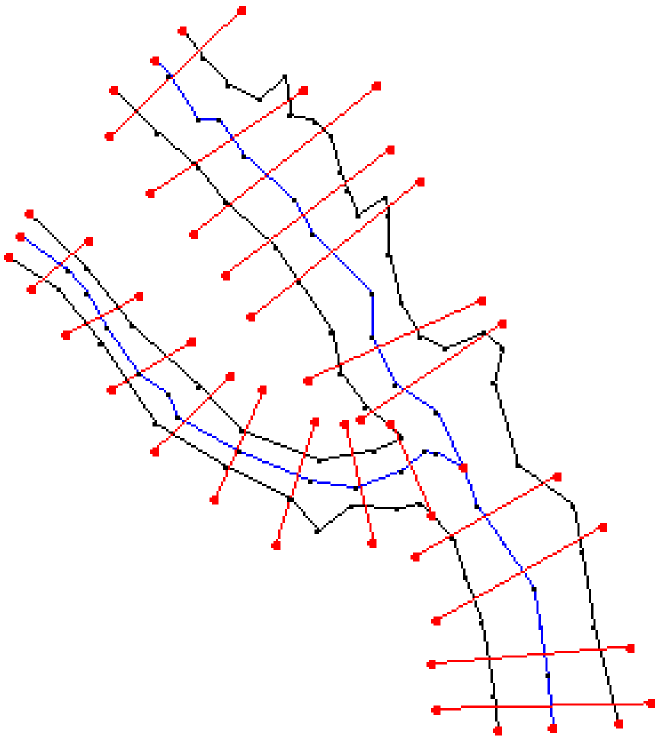


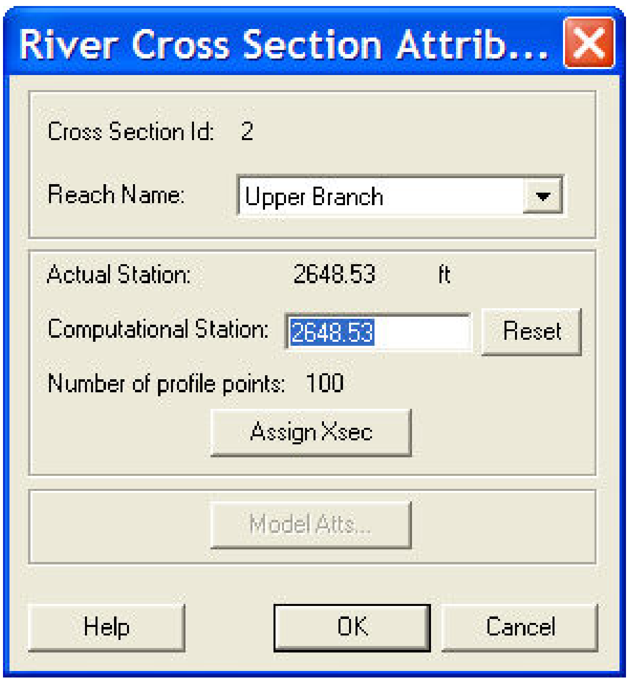
Related Topics

- Cross Section Coverage
- Area Property Coverage
- 1D Hydraulic Modeling (HEC-RAS)

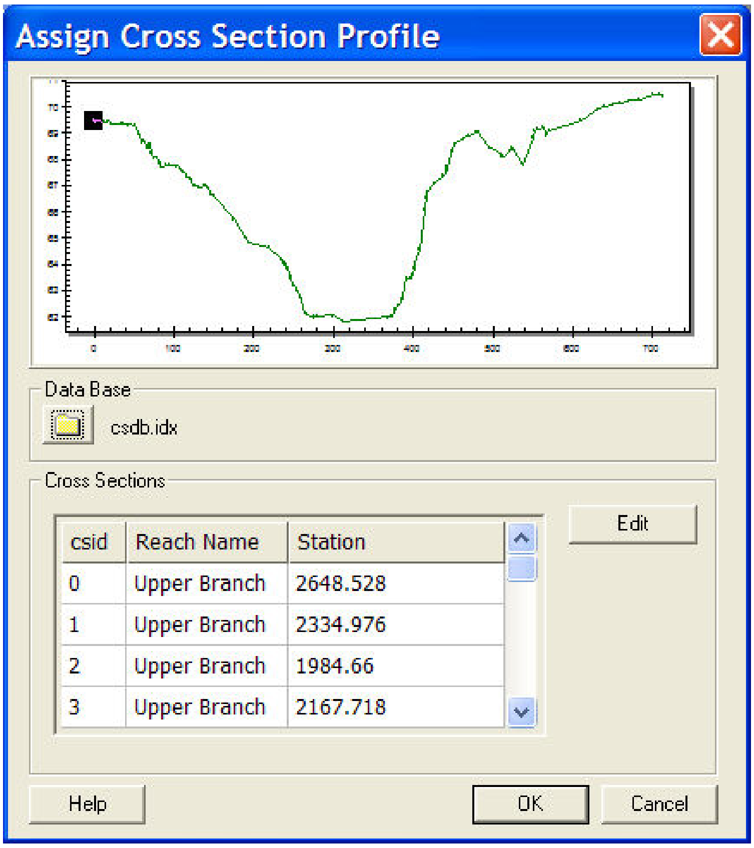
1D-HYD Cross-section

The 1D-Hydraulic Cross-section coverage is used to identify the cross section stations in the hydraulic model, and can also be used to automatically cut a cross section from an underlying digital terrain model. The attributes of a cross-section feature arc is the cross-section itself, along with the other parameters that define its topology in the model and include: a cross-section ID (internally assigned), the reach name (inherited from the centerline arc it intersects), the station (inherited from the centerline), and any specific model attributes. The 1D-Hydraulic coverage is used in conjunction with the cross sections and digital terrain model in order to determine the thalweg position, or lowest point in the cross section (from the centerline arc) and the left and right bank points (from the bank arcs).





A cross-section is assigned automatically when cutting the cross sections, or can be assigned manually (imported from a file or entered directly) using the cross section editor.



See the help for editing cross sections to learn more about how cross sections are managed and edited.

Related Topics

- 1D-Hydraulic Centerline Coverage
- Area Property Coverage
- Editing Cross Sections
- 1D Hydraulic Modeling (HEC-RAS)

Area Property Coverage

An area property coverage is very similar to a land use coverage or a soil coverage (and often are defined by converting one of these coverages to an area property) and is used to map Manning's roughness values to the cross-sections of a hydraulic model. Each polygon in the area property coverage is assigned a material ID. Material IDs are then mapped as line properties for the cross-sections when cutting from a digital terrain model. As part of the hydraulic model the material IDs are assigned unique Manning's roughness values (in the case of HEC-RAS), or other similar properties that may used in other models.

In conjunction with the Area Property coverage *Materials* should be created that correspond to the different unique IDs that are assigned to the polygons. Materials can be given a name and a color for display.

Material Properties

WMS uses an Area Property coverage to map the roughness line properties to segments of a cross section. The cross sections store the material property ID's and in order for WMS to correctly associate a material ID with an actual roughness they must be defined using the *Material Properties* dialog.

Related Topics

- Materials
- 1D-Hydraulic Centerline Coverage
- 1D-Hydraulic Cross-section Coverage
- Cutting Cross-sections
- Hydraulic Modeling

Flood Barrier

A flood barrier coverage represents natural or artificial barriers that are not represented explicitly in the elevations of a TIN. The flood plain delineation process can then use these barriers while interpolating water surface elevations across a TIN surface. The resultant flood depth will be closer to reality rather than a mere interpolation. For example, an embankment or a road is not always represented in a TIN.

Create Flood Barrier Coverage

This option allows you to incorporate flood barriers through a coverage representing natural or artificial barriers that are not represented explicitly in the elevations of a TIN. The floodplain delineation process considers these barriers during water level interpolation. The resulting flood depth will be closer to reality, rather than a mere interpolation. For example, an embankment or a road is not always represented in a DTM. In order to delineate a floodplain properly, these barriers must be considered in a hydraulic model as well as in the floodplain delineation process. This option not only incorporates such existing barriers in the process, it also provides the flexibility so that they can be considered proposed structures and evaluate the “what if” scenarios.

Note: This is not equivalent to actually running a hydraulic model with the "proposed" embankment or structure, but will limit the flood plain delineation from proceeding beyond, or at least force calculated flow paths to go around.

Related Topics

- Coverages
- Overview of Flood Plain Delineation
- Flow Paths and Barrier Coverages
- Delineate Flood Plain

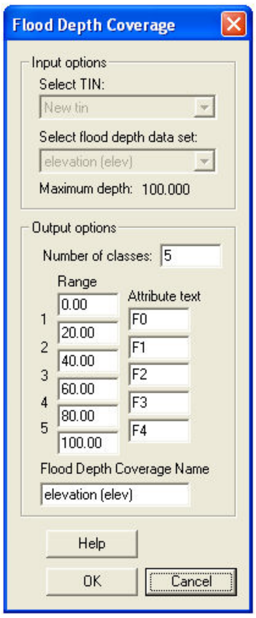
Flood Extent

A flood extent coverage is created from a flood delineation dataset and contains polygons showing inundated limits or differences in inundation limits between two different flood plain delineation scenarios. Three different types of flood extent coverages can be created: Flood Depth Maps, Impact Maps, and Extent Maps.

Flood Depth Map

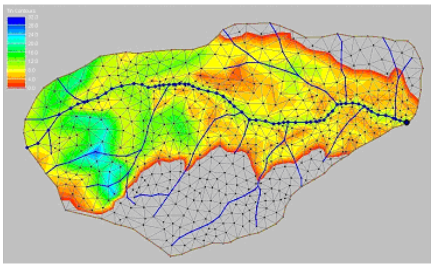
The classified flood depth coverage (**Conversion | Flood→Depth Map** command in the *Flood* menu) shows the variation of flood using different classes. Each class represents a range of flood depths (or water surface elevations) specified by the user. On the other hand the flood extent coverage contains the land-water boundary showing the extent of a floodplain. These coverages can be exported from WMS for reporting or use with Geographic Information Systems (GIS) to perform other flood management tasks.

The *Flood Depth Coverage*, dialog shown below, allows you to specify the options for creating your depth map.

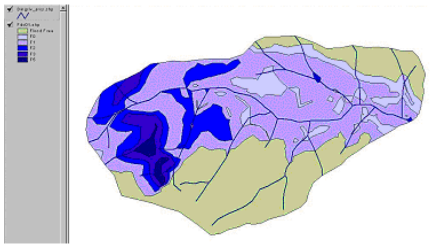


You must first select the TIN (if more than one TIN is present) and the dataset you wish to create the coverage for (this would typically be a water surface or flood depth dataset computed from the flood delineation process, but you will note that the elevation data set may be used as well). The first range will be from 0.0 to the value specified and the second range from the values between 1 and 2. The attribute field can be used to store a keyword or text that will be exported as an attribute of a shape file when exported and should be something that uniquely identifies the given range.

An example of a flood depth map is shown below:



Delineated flood plain with depth contours shown.



Classified Flood Depth Map exported as a shapefile to ArcView GIS.

Flood Impact Map

The floodplain delineation tools allow you to compare two different flooding scenarios and generate a flood impact coverage. A flood impact coverage shows increase and decrease in flooding from two different delineations using different classifications. Like other coverages, this can also be exported from WMS for reporting or other flood management purposes.

An impact map is created (**Conversion | Flood→Impact Map** command in the *Flood* menu) by showing classified increases and decreases between two separate delineations. You can specify what the increments in change are for increased and decreased (up to 5 each) flooding between two calculated datasets. A coverage of polygons is then generated for each classification. A keyword or text string can be stored with each polygon classification in order to better identify it when exported to a shape file for use in Geographic Information Systems (GIS) analysis.

Flood Extent Map

The **Conversion | Flood→Extent Coverage** command in the *Flood* menu allows you to create a polygon in a flood extent coverage that defines the inundated area as determined in the flood depth dataset. The default inundation limit is 0.0, but you can specify any positive depth value as the inundation limit and WMS will generate a polygon that conforms to that limit.

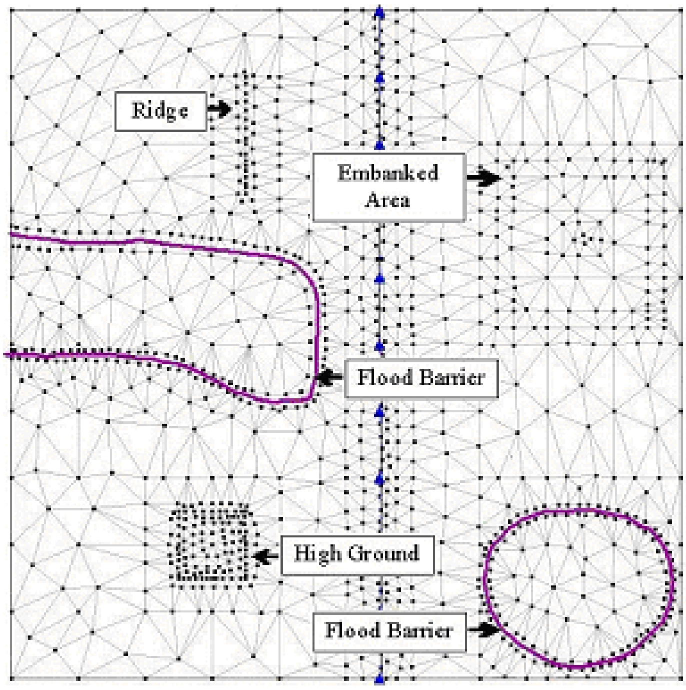
To create the flood extent map you must specify the flood depth and water surface elevation datasets along with the inundation limit. You can optionally have WMS force or stamp in the flood extent coverage to the TIN. This will insure that you have TIN vertices with depth values at exactly 0.0. This is not the case in the original data sets computed by the flood plain delineation algorithm. The 0.0 (or any other depth contour value) value is likely interpolated along a triangle edge between two vertices. Stamping the flood extent coverage into the TIN is particularly useful for confining the display of water surface elevation values to be within the flood extent region. While the inundation limit for flood depth is a constant value (0.0), the value of water surface elevation at the inundation limit is not. By stamping in the flood extent coverage and updating the associated data sets water surface elevation contours can be controlled since WMS will set vertices outside of the flood extent coverage to be inactive and those inside to be active.

Related Topics

- Coverages
- Overview of Flood Plain Delineation
- Delineate Flood Plain

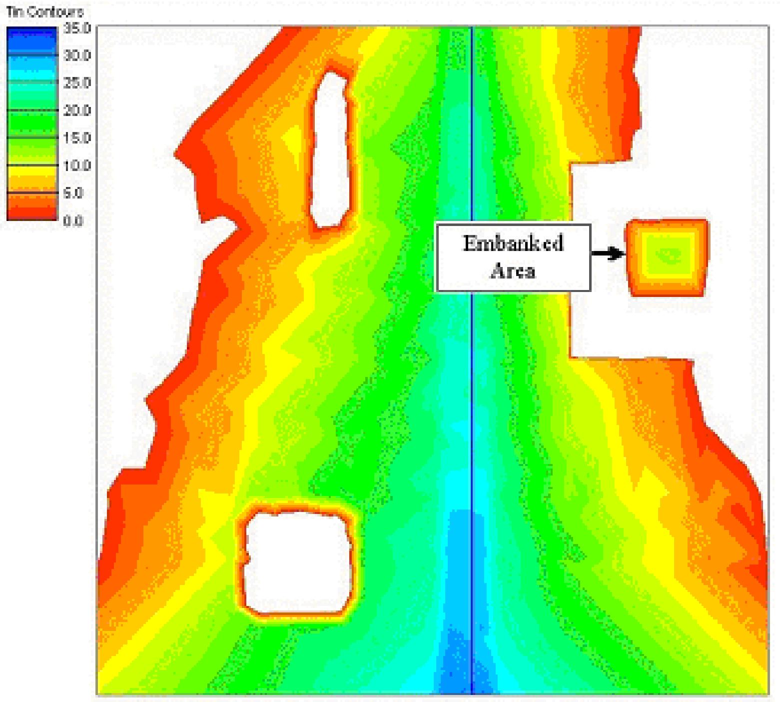
Flow Paths and Barrier Coverages

Figure (a) shows natural high ground and embankment in a TIN and the location of water levels and flood barriers used in the floodplain delineation. This and the subsequent figures demonstrate the following options: 1) floodplain delineation without considering the flow path option, 2) floodplain delineation considering the flow path option, and 3) floodplain delineation considering flood barriers.



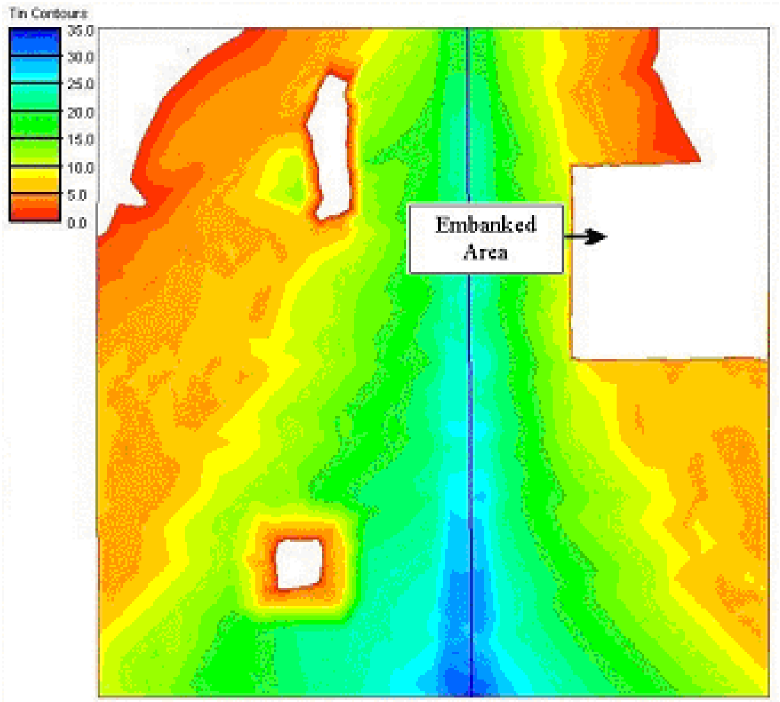
(a) Sample TIN for illustrating flood plain delineation options.

The flow path option in the floodplain delineation process ensures that the sources of water (i.e. the water levels) and the areas flooded are hydraulically connected. This is an important option because if not applied, the process may interpolate water levels while ignoring obstructions between the water levels and the point of interpolation. The effect of such interpolation is shown in Figure (b) where the floodplain is delineated without considering flow paths. The figure shows two flooded areas separated from each other by a natural embankment. Considering the water levels, which are located outside the embankment and lower than the elevation of the embankment, the area inside the embankment should not be flooded. Therefore, it is obvious that the flooding inside the embankment is the effect of the interpolation done by the process without checking hydraulic connectivity.



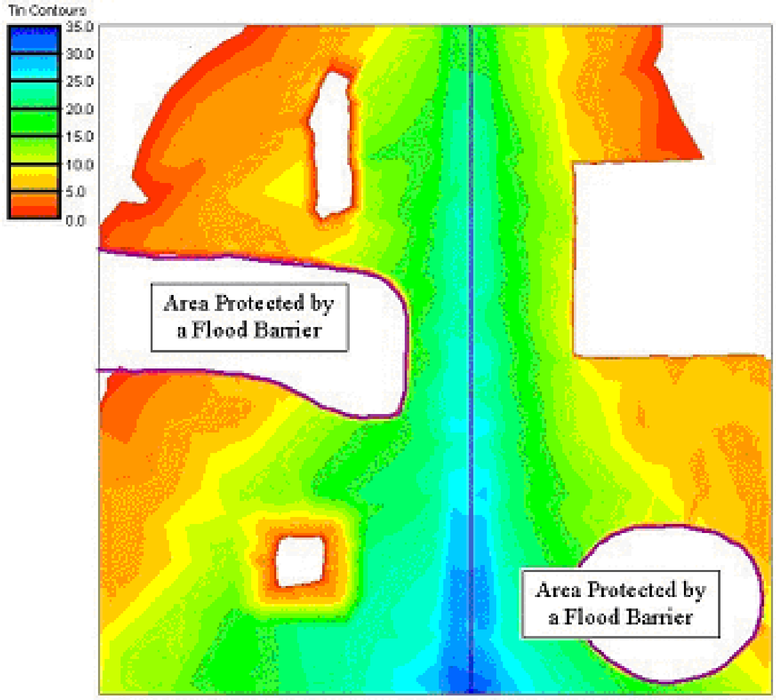
(b) Flood plain delineation without consideration of flow paths

The more realistic flooding scenario is simulated when flow paths are considered as shown in Figure (c). In this case the process first checks for hydraulic connectivity before performing any interpolation at any location inside the embankment. Since the water is obstructed by the embankment the process does not find any flow paths. As a result the area inside the embankment remains flood free.



(c) Flood plain delineation using flow paths.

Not all natural and artificial flood obstructions are well represented by digital terrain models. The effect of these obstructions cannot be simulated with the flow path option alone. The use of flood barrier coverages along with the flow path option provides a way to address this issue in the floodplain delineation process. The effect of using flood barriers in the floodplain delineation is demonstrated in Figure (d), which shows an area on each side of the river that is protected by a floodwall. Since the elevations in the TIN do not show the presence of the floodwalls, a flood barrier coverage is used to simulate the effect in the floodplain delineation. While checking hydraulic connectivity, the process recognizes those barriers and therefore, reports the areas inside the barrier as flood free.



(d) Flood plain delineation using a flood barrier coverage.

Related Topics

- Overview of Flood Plain Delineation
- Delineate Flood Plain
- Flood Barrier

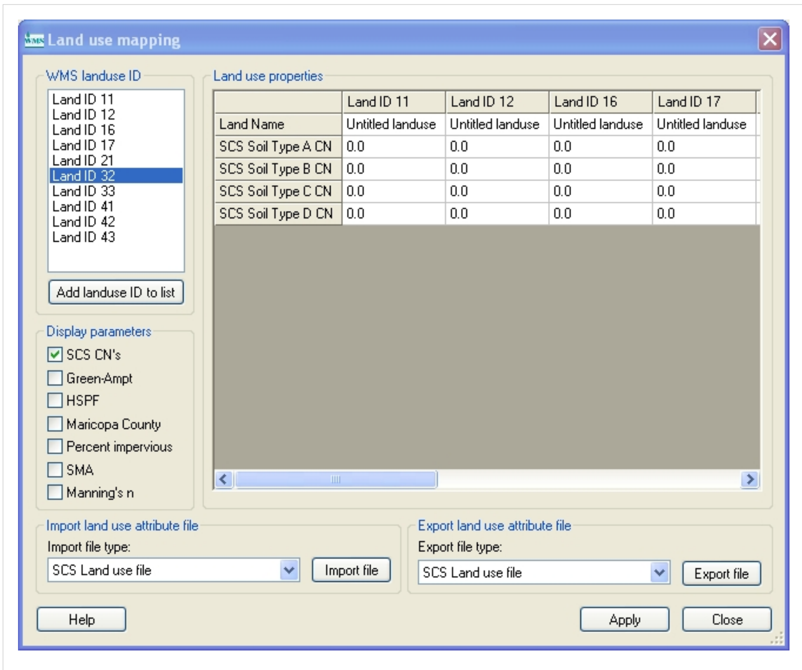
General

Sometimes you may be unsure which coverage attributes to use, especially when importing a layer from another source (GIS, CAD). The general coverage allows you to create/edit points/nodes, arcs, and polygons without defining attributes, or worrying about special rules associated the different coverages and attributes. For examples WMS requires that stream and time computation arcs be defined from downstream to upstream. If you import data from another source you have no guarantee of the ordering, and if you import them to a Drainage Coverage type it may cause problems. Therefore, you can import them to a general coverage, make appropriate edits (streams can be reordered by selecting the downstream most point and choosing the **Reorder Streams** command from the **Feature Objects** menu), and then convert the coverage to a drainage type.

Related Topics

- Feature Objects
- Coverages

Land Use Coverage



A Land Use coverage in WMS has a different purpose depending on the model and/or model parameters you wish to assign to basins using land use as an indicator. The following is a list of operations that can be done using land use coverages, and the parameters that must be linked to land use ID's:

1. SCS (NRCS) Curve Numbers for hydrologic soil groups (requires the use of a soil coverage as well).
2. Soil type A CN, soil type B CN, soil type C CN, and soil type D CN.
3. Green & Ampt parameters for infiltration modeling in HEC-1.
4. Initial abstraction, percent impervious, percent vegetation cover.
5. HSPF Pervious and Impervious land segments and other parameters.
6. Canopy and surface storage for SMA losses.

You indicate which application(s) you wish to use the land use coverage for by selecting the appropriate toggle boxes in the *Display parameters* (lower left) section of the *Land Use Mapping* dialog. You may only define parameter values for the applications selected.

Each land use polygon will have a land use ID associated with it (a single integer number). In order to perform the correct mapping you will need to link the appropriate land use variables to each land use ID. WMS allows you to do this in one of two ways. First of all, with the land use coverage active, you can open the *Land Use Mapping* dialog from the *Feature Objects* menu using the **Attributes** command, and then create new IDs and enter parameters for each ID. Secondly, you can enter the data in a text file and then import it from within the same dialog.

Parameter values for land use IDs are defined by selecting the ID in the WMS landuse ID text window and the parameter from the selected land use properties text window, and then entering the value in the edit field.

If you choose to define all the data manually using the *Land Use Mapping* dialog, you will want to export the data to a file so that you will not need to reenter the data for future models using the same land use parameter definitions.

Once the land use parameters have been defined, land use ID’s can be assigned to polygons by first selecting the desired polygon(s) and then choosing the **Attributes** command from the *Feature Object* menu (or by double-clicking on the polygon) and choosing the appropriate ID from the WMS land use ID text window.

Mapping files may also be imported/exported so that if you constantly use the same landuse id attributes you do not need to enter the values for each new model.

Once land use IDs have been assigned to polygons and parameters linked to the land use IDs, model parameters can be computed using the **Compute Composite Runoff Coeff/CN…** command from the *Calculators* menu in the Hydrologic Modeling module .

Related Topics

- Feature Objects
- Coverages
- Computing Composite Curve Numbers
- Obtaining Land Use Data

MODRAT Tc Coverage Type

Use this type of coverage to define time of concentration arcs for a MODRAT simulation.

To change the attributes for feature arcs, select the arc(s) and click **Attributes** from the *Feature Objects* menu, or double-click on the arc. Feature arcs created in the Modrat Tc coverage can have the following conveyance types.

- street
- 1/2 street
- pipe
- channel
- mountain channel
- valley channel
- triangular channel
- overland
- none

Related Topics

- Feature Objects
- Coverages
- Overview of MODRAT
- Rainfall Zone

NSS Region Coverage

An NSS Region coverage can be generated so that regions can automatically be mapped to a drainage coverage when performing hydrologic analysis using the NSS regression equations. Individual state maps can be found at the USGS website: http://water.usgs.gov/software/NSS/(Look at the bottom of the page and select your state). If you want to digitize the state, then download the image from the USGS website and save the picture as either a *.tif or *.jpg image (many times the default will be a *.gif image) using any type of imaging software program. Load the *.tif, or *.jpg image into WMS and register according to the latitude and longitude coordinates found on the map, and then convert the coordinates to the coordinate system you will be delineating your watershed in. With the image in the background you can easily digitize the lines and build the polygons of the coverage. Finally assign the attributes (state and region) to the polygons and save it as a map file. This NSS coverage file can then be used for any watershed within the given state.

Related Topics

- NSS Model
- Image Registration
- Coordinate Conversions

Rain Gage

The rain gage coverage in WMS is designed for use with HEC-1, HEC-HMS, and GSSHA. Attributes can only be assigned to points in the rain gage coverage. WMS automatically computes and displays Thiessen polygons for all of the gages defined on a rain gage coverage according to their x,y coordinates.

HEC Gages

Manage the rain gage data required by HEC hydrologic models, including the x,y gage location, storm total precipitation values, and the temporal distribution. When gages are present, weights are assigned to each basin from the Thiessen polygon network as the basin data are computed. See Weighted Average Precipitation for instructions on how to use rain gages in HEC-1 to compute a weighted average precipitation.

GSSHA Gages

Choose one of the following types of precipitation data to apply to all gages on the coverage:

- GAGES – rainfall accumulation (mm) over the last time period
- RADAR – rainfall rate (mm/hr) for the last time interval
- RATES – rainfall rate (mm/hr) for the next time interval
- ACCUM – cumulative amount of rainfall up until that time period (mm)

Enter the name, x,y locationand precipitation values corresponding to the precipitation type for each gage. When entering data some guidelines to be aware of include:

- Precipitation types (GAGES, RADAR, RATES, and ACCUM) cannot change within a storm event (on a rain gage coverage)
- The time interval can be any value, but there must be a rainfall value at each regular time interval.

Related Topics

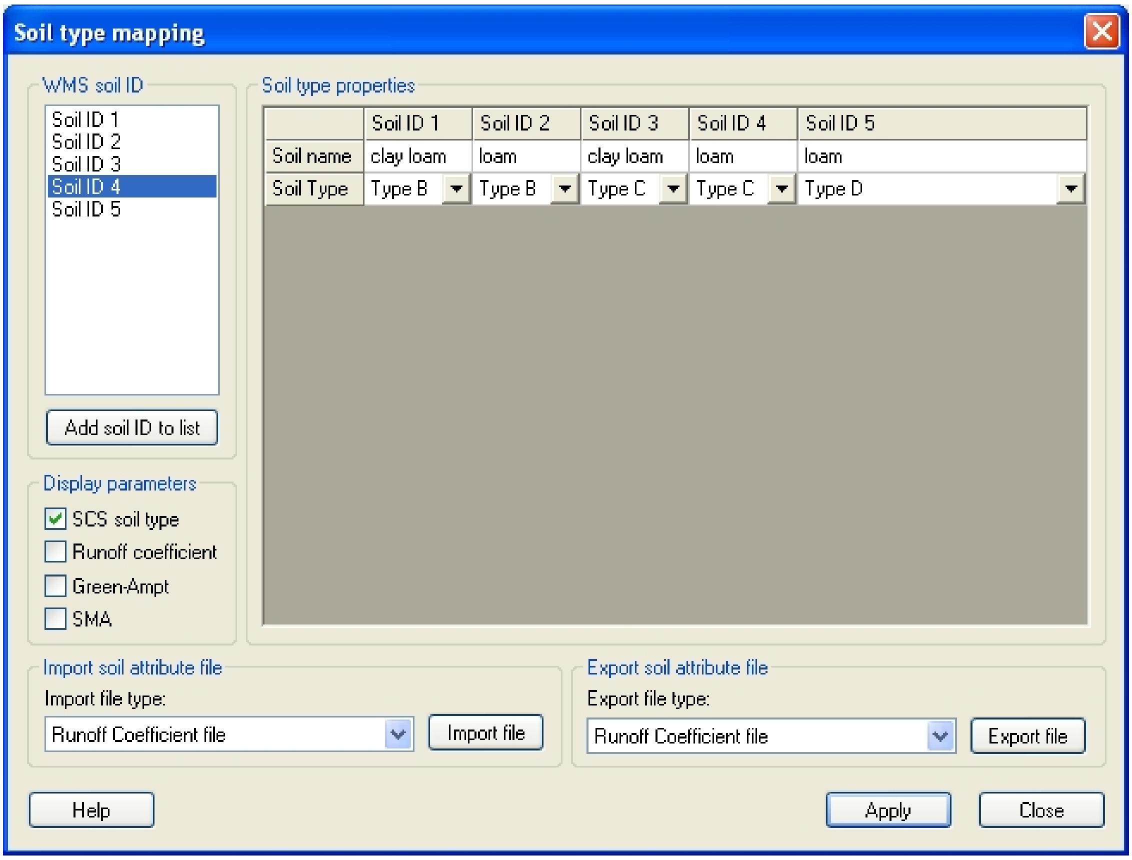
- Coverages
- Feature Objects
- GSSHA Precipitation
- HEC-1 Gages
- HEC-HMS Gages

Soil

The soil type coverage is similar to the land use coverage in that it can be used to map different model parameters (related to soil type) from polygonal coverages (usually imported from a GIS). The following is a list of operations that can be done using soil type coverages and the parameters that must be linked to soil IDs:

1. Hydrologic soil group to map SCS Curve Numbers (requires the use of a land use coverage as well).
2. Soil Type Number (must be = 0 for type A, 1 for type B, 2 for type C and 3 for type D).
3. Runoff coefficients for the Rational Method (C in Q = CiA).
4. Runoff coefficient.
5. Green & Ampt parameters for infiltration modeling in HEC-1.
6. Soil texture (soil name) to map SMA loss parameters.
7. Hydraulic conductivity, percent impervious, percent effective.

You indicate which application(s) you wish to use the soil type coverage for by selecting the appropriate toggle boxes in the *Display parameters* (lower left) section of the *Soil Type Mapping* dialog (shown below). You may only define parameter values for the applications selected.



Each soil type polygon will have a soil type ID associated with it (a single integer number). In order to perform the correct mapping you will need to link the appropriate soil type variables to each ID. WMS allows you to do this in one of two ways. First of all, with the soil type coverage active, you can open the *Soil Type Mapping* dialog from the *Feature Objects* menu using the **Attributes** command, and then create new IDs and enter parameters for each ID. Secondly, you can enter the data in a text file and then import it from within the same dialog.

Parameter values for soil type IDs are defined by selecting the ID in the WMS soil type ID text window and the parameter from the selected soil type properties text window, and then entering the value in the edit field.

If you choose to define all the data manually using the *Soil Type Mapping* dialog, you will want to export the data to a file so that you will not need to reenter the data for future models using the same soil type parameter definitions.

Once the soil type parameters have been defined, soil type IDs can be assigned to polygons by first selecting the desired polygon(s) and then choosing the **Attributes** command from the *Feature Object* menu (or by double-clicking on the polygon) and choosing the appropriate ID from the WMS soil ID text window.

Once soil type IDs have been assigned to polygons and parameters linked to the soil type IDs, model parameters can be computed using the **Compute GIS Attributes** command from the *Calculators* menu in the Hydrologic Modeling module.

Related Topics

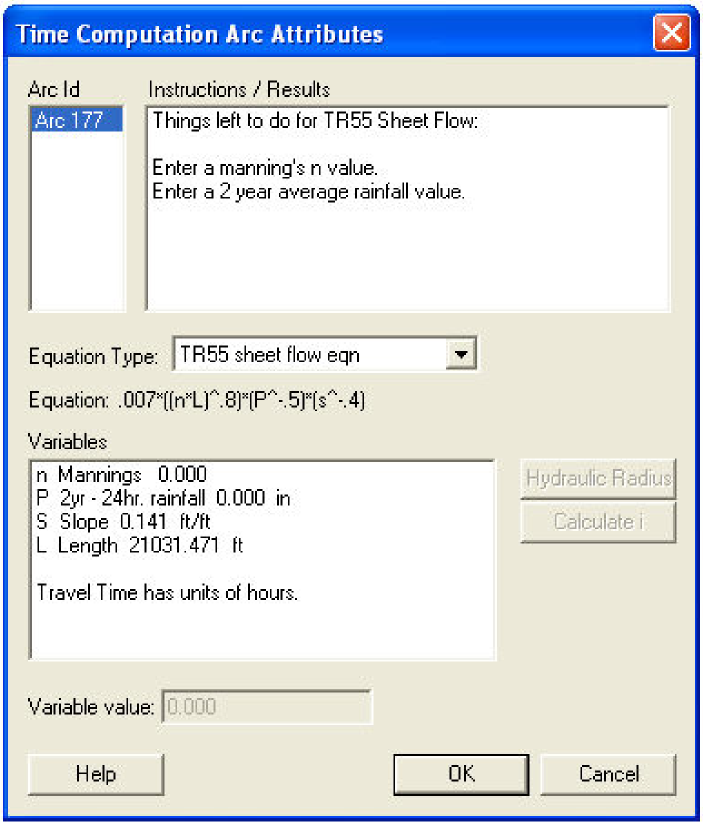
- Feature Objects
- Coverages
- Computing Composite Curve Numbers
- Obtaining Soil Type Data

Time Computation

The Time Computation coverage type allows you to define equations along arcs for computing flow path travel times within a basin. In this way times of concentration or lag times can be determined using standard overland, and channel flow equations such as those used by the FHWA, TR-55, or any other user-defined equation. Since most equations used for travel time are functions of flow path length and slope, WMS automatically determines the length of the arc, and if a TIN or DEM is available the slope, and makes them available for use in an equation.

Time of concentration arcs are similar to stream arcs as defined in the drainage coverage in that their direction is important. When creating time computation arcs you should always define them from “downstream” to “upstream” in the same way that drainage arcs are defined.

Time computation arcs are the only type of arcs in the Time Computation coverage. Each arc is assigned a time of travel equation. Equations may be selected from a library of equations or as a user-defined equation. The predefined equations in WMS include TR55, FHWA, and Maricopa County. For example, TR55 includes the standard equations for sheet flow, shallow concentrated flow, and open channel flow. The dialog shown is used to assign the arc attributes and corresponding values for the parameters of the equation so that a travel time for the arc can be determined.



Time of concentration or lag time for a basin is determined by summing the travel times of all time computation arcs within a basin. A summary of how this is automated within WMS and further information on defining equations and computing travel times for time computation arcs is given in the hydrologic calculators section.

There are no point/node or polygon attributes in Time Computation coverages.

Time Computation Arcs

The **Node → Flow arcs** command in the **Feature Objects** menu allows you to automatically develop time computation arcs by tracing the flow path across a TIN or DEM and saving an arc representing the path so that an equation can be assigned to it. You can investigate where the arc(s) will be created using the flow path tool and selecting points on the TIN or DEM since the same functions for flow are used to create the arcs. The process requires the following three steps:

1. Create a feature points at locations where the flow path segments should begin.
2. Make sure any feature points you wish to use to create time computation arcs are selected.
3. Choose the **Node→Flow arcs** command.

You will be asked if you want to create one continuous arc or multiple arcs. If you respond with one continuous arc it will create an arc representing the flow path from the selected feature point to the next downstream outlet. If you choose to create multiple arcs it will break the arc into separate arcs if the flow path encounters a stream.

Flow Paths to Time Computation Arcs

When using the **Node → Flow Arcs** command in the **Feature Objects** menu, only the portions of the stream that are part of the flow path from the selected point to the outlet get converted to time computation arcs. If you wish to compute the lag time between consecutive outlet points then you will need to convert the remaining stream portions to time computation arcs. This is the purpose of the **Streams → Flow Arcs** command. Not all streams are converted when using this command, only those stream segments that connect outlets. This command works for either TIN or

DEM feature object stream segments.

Flow Path Arcs

To compute the time of concentration in a sub-basin, the longest flow path in each basin must be defined. Once a watershed has been defined and broken into sub-basins, flow path arcs (Time Computation arcs) may be defined for each sub-basin. These arcs must be created in a Time Computation Coverage; the Coverages dialog in WMS includes the option to create this type of coverage. The new coverage should be created and activated before creating flow path arcs.

When a Time Computation Coverage is active, $T_c$arcs may be created using the Create Feature Arc tool in the Map Module of WMS. These paths are created by simply pointing and clicking along the desired path, and double-clicking to end the arc. The $T_c$arcs created should start near the basin outlet and follow the longest flow path in the basin. This path must often be determined from knowledge of the area or visual inspection of a map or photograph of the area. However, WMS can automatically create flow paths if elevation data exists and is imported into WMS. By creating a node at the furthest point in the basin, the user can direct WMS to define the flow path to the outlet when the **Node to Flow Path** command is issued from the **Feature Objects** menu.

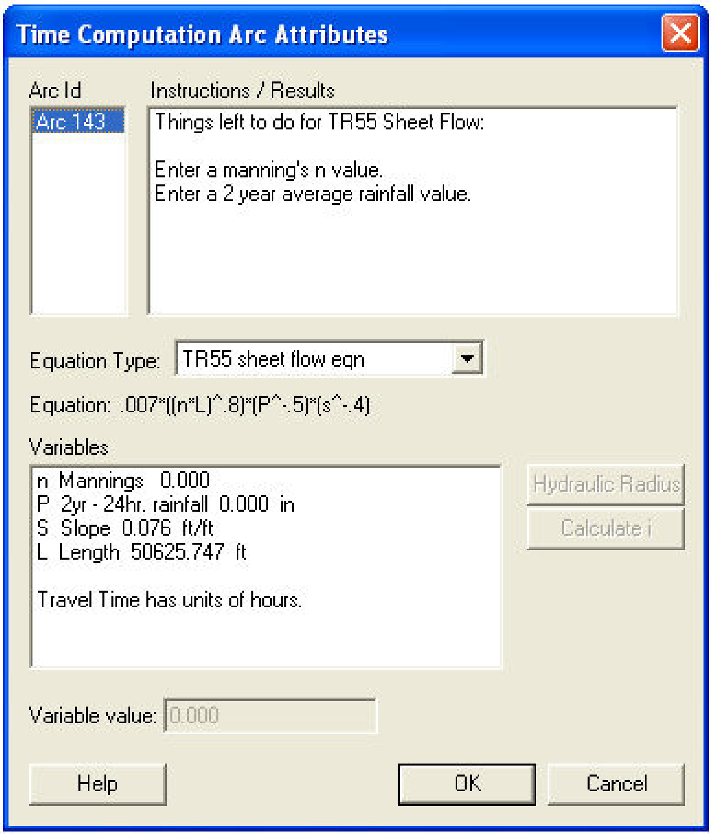
NOTE: the flow path arc for each basin must not cross the basin boundary at any point. Such a crossing will result in errors.

Automated T_cCalculations

WMS will compute $T_c$for all sub-basins using the LACDPW $T_c$regression equation and the parameters defined along time computation arcs (on the Time Computation coverage) using the **Compute** $T_c$command in the **MODRAT** menu. Error messages are displayed and $T_c$will not be computed for any sub-basin if any of the required parameters are missing. This automates the process of selecting the time computation arc, verifying the input parameters, and computing $T_c$for each sub-basin.

Assigning Equation to Arcs

The primary attribute for a time computation arc is the equation that will be used to compute travel time for the flow path segment represented by the arc. In addition to the equation, the length, slope, and travel time for the arc are also stored. Length is defaulted from the length of the arc and a slope will be determined for the arc if there is an underlying TIN or DEM. However, you can edit either of these values if you want. For example you may decide that the actual flow path is somewhat more sinuous than the arc represents and decide to increase the length. The equation and appropriate variables are defined for an arc using the *Time Computation Arcs Attributes* dialog shown below.



This dialog is used to set all values and compute travel times for selected arcs. It can be accessed by selecting an arc and then choosing the **Attributes** command from the *Feature Objects* menu in the Map module, by double clicking on the arc when in the map module, or when using the *Travel Time Computation* dialog. If multiple arcs are selected the ArcID window displays the ID of all selected arcs and you may edit the equations / variables of any selected arc by choosing it from this window. While the selected arc is highlighted in the Graphics Window, you may find it useful to toggle on the display of arc IDs from the *Feature Object Display Options* dialog.

When using the *Time Computation Arc Attributes* dialog the Instructions / Results window will let you know which variables need to be entered before a time computation can be made, and when all variables are defined it will display the computed travel time of the selected arc.

Editing Equation Variables

Equation variables from the currently selected arc are displayed in the variables window of the *Time Computation Attributes* dialog. Variables such as length and slope will generally have defaulted values, however other variables such as Manning's roughness coefficient will need to be entered before a travel time for the arc can be computed. Variables are edited by selecting the variable you want to modify from the text window and then setting the value in the adjacent edit field. The Instructions/Results window will let you know when variables have not been defined and the travel time for the arc once all variables have reasonable values.

Assigning the Regression Equation

Once a flow path (time computation) arc has been created for each sub-basin, the LACDPW TC regression equation must be assigned to each arc. This can be done in two ways:

1. After creating the $T_c$arc, double-click on it with the *Select Feature Arc* tool. The *Time Computation Arc Attributes* dialog will appear.
2. Select a sub-basin and bring up the *Edit MODRAT Parameters* dialog. Then click on the **Compute Regression** $T_c$button. This will invoke the *Travel Time Computation* dialog; the flow path arc can be selected and assigned an equation by selecting the **Edit Arcs** button. This button will invoke the *Time Computation Arc Attributes* dialog.

Once in the *Time Computation Arc Attributes* dialog, choose the LACDPW TC equation. The necessary variables will be shown in the window at the bottom of the dialog. If you have computed the watershed data and assigned rainfall, soil type, and percent impervious, all the variables will be assigned the appropriate values. If not, you can select the variables and enter a value manually.

NOTE: Once you have selected the LACDPW TC equation for one $T_c$arc in your model, it becomes the default for all other arcs.

Regression T_cArc Computations

To produce accurate runoff results, MODRAT must have data to allow routing of flows through basins and reaches. One of the most important parameters needed for these operations is the time of concentration. Differences in the time of concentration can have marked effects on hydrograph peaks and shapes (temporal runoff distribution), especially when hydrographs from various basins are combined.

As a general rule, the time of concentration calculations should be done after all other parameters have been assigned to each sub-basin of your MODRAT model. The reason for this is that the methods used to compute $T_c$use rainfall depth or zone, soil type, and percent impervious as variables. If these parameters have been assigned to each sub-basin, WMS will automatically plug those numbers into the equation to compute $T_c$for the sub-basin.

Once the LACDPW TC equation has been assigned to an arc and all variables defined, WMS will compute the T_c for the arc and display it in the Time Computation Arc Attributes dialog. The computation of the time of concentration requires data from the soil file (lasoilx.dat) to be present. If this file is not present in that folder, you will be prompted to designate the location of the soil file.

To assure the $T_c$is assigned to the MODRAT model:

1. Select a sub-basin and bring up the *Edit MODRAT Attributes* dialog.
2. Click the **Compute Regression** T_c' button.
3. Ensure that a $T_c$has been computed in the *Travel Time Computation* dialog; edit the $T_c$arc data if necessary.
4. Click OK and the $T_c$will be assigned to the sub-basin.
5. Repeat for each sub-basin.

Compute Regression T_c

The LACDPW has developed a regression equation to compute time of concentration for the MODRAT program. Use of these equations in WMS is done through the Time of Concentration Calculator. The basic steps to using this calculator are:

1. Define flow paths with *Time Computation Arcs*
2. Choose the regression equation to assign to the flow path
3. Enter needed parameters
4. Compute T_c

Related Topics

- Feature Objects
- Coverages
- Computing Lag Times and Time of Concentration
- Time of Concentration
- Travel Times from Map Data
- Editing Equation Variables

5.5.b. Feature Objects

Feature Objects

Feature objects in WMS have been patterned after Geographic Information Systems (GIS) objects and include points, nodes, arcs, and polygons. Feature objects can be grouped together into coverages, each coverage defining a particular set of information. The use of feature objects is determined by the coverage, or attribute set, to which they belong, but can be separated into three important categories:

1. As a means of defining basin polygons and stream networks of pre-delineated watersheds (typically this data would be imported as a shapefile from ArcView or ArcInfo where the basin delineation and attribution has already taken place)
2. To define a conceptual model or layout of features in the watershed, such as its rough boundaries and streams. This conceptual model is then used to aid in the construction of a TIN or DEM processing for delineating watershed and sub-basin boundaries
3. Soil, land use, rainfall, or other data which can be used to define important hydrologic modeling parameters such as curve number (CN) or rainfall zone.

Related Topics

- Drawing Objects
- Images
- Shapefiles
- Coverages
- Map Module
- Feature Object Guidelines
- Build Polygons
- Creating TINs
- Creating Watershed Models
- GIS Module

Feature Object Guidelines

GIS vector data includes points, lines, and polygons that are used in WMS to represent basins, streams, and key points such as outlets or culverts. In WMS we refer to this GIS data as Feature Objects, and tools for using them are included in the Map Module. Feature object data can be used by itself to create a watershed model for hydrologic analysis or as a companion in the development of watershed models with TINs and DEMs.

Many times it is not practical to obtain digital elevation data and perform an automated watershed characterization prior to setting up a hydrologic model. Watershed and sub-basin boundaries may already be known and stored as part of a GIS or CAD database, or it may be straight-forward to trace an existing map to define streams and basins. With WMS, properly structured hydrologic models can be created automatically from points, lines, and polygons. Since these data are often already developed and stored in a GIS, importing from ARC/INFO® and ArcView®, or DXF files is easily done.

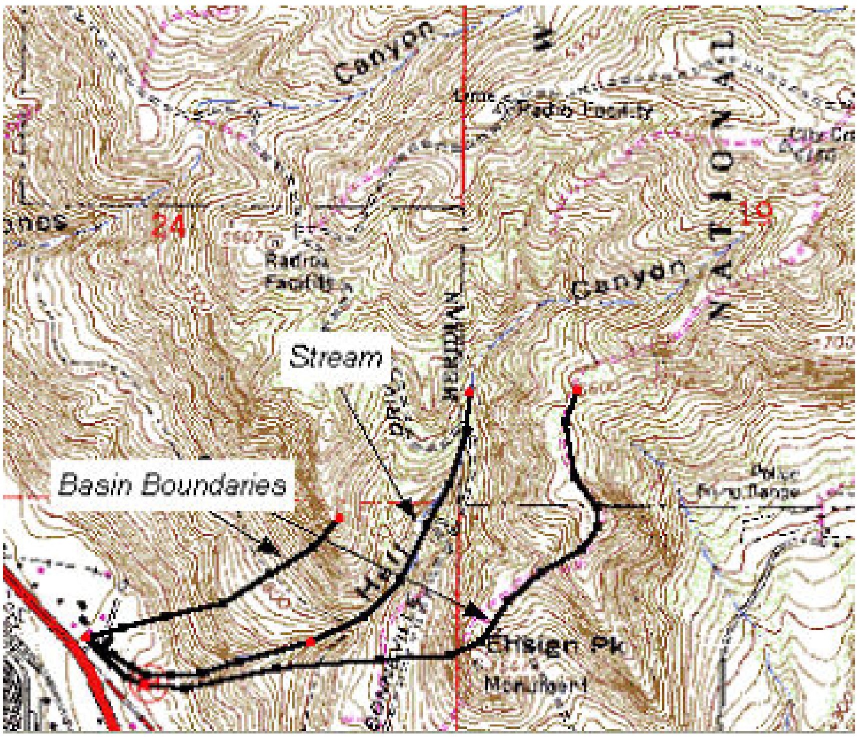
The following are the basic steps taken to create watershed models from GIS data within WMS.

1. Obtain a Map or Already Developed GIS or CAD Data

The first step is to obtain a map that defines the streams and basins which will be modeled. If such a map already exists digitally as a CAD drawing or as part of a GIS database then it can be imported directly and the next step can be skipped.

2. Digitize the Map

The map can then be digitized using a tablet and standard digitizing software outside of WMS and then imported as a CAD or GIS file, or it can be created using "heads up," or "on-screen digitizing" inside of WMS. In order to do heads up digitizing you will need one of two things: 1) digital elevation data that can be contoured by WMS, or 2) a scanned tiff image that can be read into WMS and used as a background map.



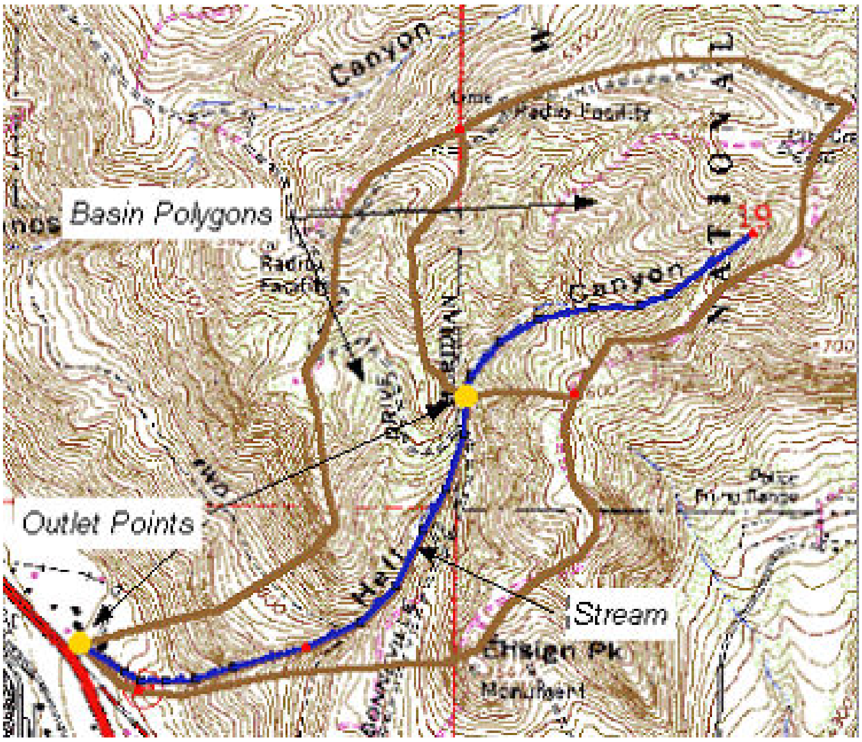
3. Construct Feature Object Topology

The points and lines must be assigned the right attribute types, and the polygons for sub-basins constructed from the lines.

If you use data already developed in a GIS then you may have to do some editing. This will depend on how well the data being imported matches with the required data for watershed model development. In WMS three primary layers as illustrated below : 1) A point layer representing the watershed outlet and any sub-basin outlet or confluence points, 2) a line layer representing a stream network, and 3) a polygon layer representing watershed boundaries.

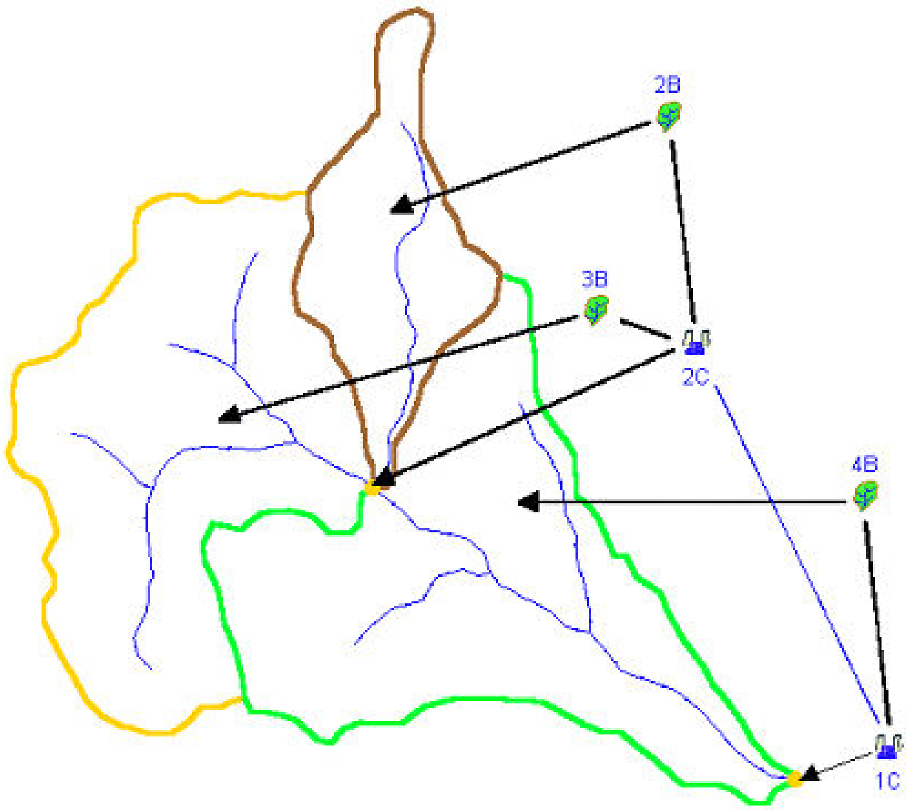
If all three layers exist then construction of the watershed model topology can proceed, but if one or more of the layers are absent, they must be created manually from within WMS. For example if you only had a file that contained sub-basin boundaries, you would need to digitize the stream network and define the outlet locations of the sub-basins.

An important point to remember in WMS is that lines used to define a stream network have direction. For each line (arc) there is a beginning and an ending node and “flow” along the line is defined in this direction. When interactively creating lines in WMS you should always create streams from downstream to upstream. If you import a set of lines that has been previously created by another program you may discover that the order does not match what is required by WMS. The Reorder Streams command in the **Feature Objects** menu (Map module) can be used to properly order the incoming lines so that they can be converted to streams.



4. Define the Hydrologic Model

Once the watershed model representation has been created, data defining a specific hydrologic model can be entered through a series of user friendly dialogs. Since WMS allows for all hydrologic modeling input to be defined separate from any digital terrain data, it is not required that the watershed model developed with feature objects be to scale. Area and length parameters can simply be manually defined using the model interface. The figure below shows how a topologic model is automatically created from point, line, and polygon feature object data.



Related Topics

- Map Module
- DEM Guidelines
- TIN Guidelines
- Data Acquisition

Feature Objects Menu

Cleaning

The **Clean** command in the *Feature Objects* menu is used to clean up feature object data. Specifically, it prompts for a snapping tolerance and minimum dangling arc length, and then uses these parameters to do the following:

- A check is made to see if any nodes are within tolerance of other nodes. If so, the nodes are snapped together.
- A check is made to see if any arcs intersect. If so a node is created at the intersection and the arcs are split.
- A check is made for dangling arcs (arcs with one end not connected to another arc) with a minimum length. If any are found they are deleted.

All objects of the active coverage will be cleaned.

Alternatively, a check is made to see if any arcs, vertices, nodes, or points are selected in the active coverage. If any intersecting arcs are selected, you have the option of intersecting only the selected arcs. If any points, nodes, or vertices are selected, you snap the selected points, nodes, and vertices to a snapping point that is selected after the Clean Options dialog is closed.

Build Polygons

Just defining a series of arcs that form a closed loop, or polygon, does not create a polygon. Polygons are created from arcs only after the **Build Polygons** command in the **Feature Objects** menu is used. Feature polygons can be created in one of two ways:

1. If there are no selected arcs when the **Build Polygons** command is chosen, polygons are created for all arcs of the active coverage that form closed loops, or polygons. The one exception being that if the active coverage is a “drainage” type coverage, stream arcs are not used to create polygons. If you want a stream arc to also form a basin boundary then you must build polygons according to method two below.
2. If there are selected arcs when then **Build Polygons** command is chosen then polygons are only created for closed loops or polygons formed by the set of selected arcs.

By either method, the new polygon inherits the current default polygon type, unless in the drainage cover and then the new polygon will be defaulted to a lake polygon if all it's arcs are lake arcs and a drainage polygon otherwise.

Attributes

The attributes of feature objects depend on the type of feature selected (point, line, or polygon) and the coverage to which it belongs. Many coverages are polygonal only (i.e. land use, soil, area property) and so the only kind of feature object that has attributes are the polygons. Other coverages, like cross sections or time computation arcs only have attributes for the lines. When you choose the **Attributes** command in the **Feature Objects** menu the attributes dialog of the currently selected feature object will appear. You can also invoke the attributes dialog by double clicking on the desire feature object.

See the coverages listing to read more about the kinds of attributes that feature objects have for each coverage type.

Redistribute

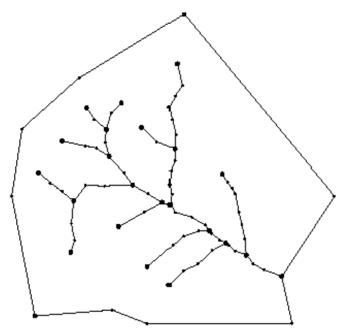
Vertices along arcs can be redistributed at either a higher or lower density using the **Redistribute** command in the **Feature Objects** menu. The vertex density along arcs determines the density of TIN vertices when issuing the Create TIN command from the Feature Objects menu. Vertices are redistributed along all selected arcs using a uniform or cubic spline method. Arcs can be selected one at a time using the Select Arc tool, the **Select All** command in the **Edit** menu (the Select Arc tool must be active), the Select Branch tool, or the Select Network tool. Once the arcs are selected the method of redistribution can be chosen from the Redistribute dialog.

Uniform Subdivision

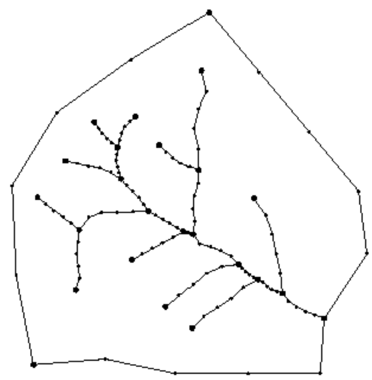
If the Subdivide each end uniformly options is specified then either a number of intervals, or a specified spacing can be given to determine how points are redistributed along the selected arcs. If the specified spacing is greater than the length between adjacent arc vertices, the vertices are moved to reflect this larger spacing.

Spline Redistribution

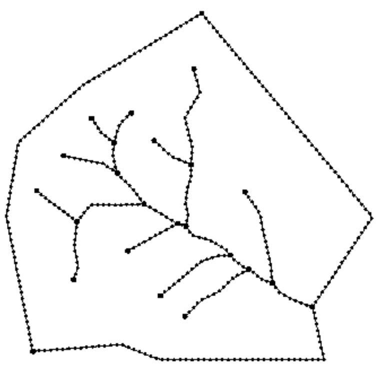
If the Redistribute along a cubic spline option is specified vertices between arcs are redistributed by creating a series of splines from the vertices of selected arcs and then redistributing a new set of vertices at the specified distance. If the flag for preserving points with an angle greater than the specified angle is checked then vertices at such locations will remain after the redistribution. This method can be used to create vertices at a smaller density (specify a larger target spacing) as well as a higher density.



Arcs before redistribution



Arcs with a constant interval of 5.



Arcs with a constant spacing of 100.

Arcs with a splined redistribution

Reorder Streams

The **Reorder Streams** command in the **Feature Objects** menu is used to ensure that the direction of all stream arcs are consistently defined from a selected outlet node. When creating stream arcs in WMS a check is made to ensure that points are created from downstream to upstream, so that the first node in the arc is the downstream node. This makes it impossible to create streams in WMS that do not follow this definition. However, when arcs are imported it is possible to create streams where the downstream node is second rather than first for some arcs but not others. All arcs can be consistently ordered by selecting an outlet node (a node attached to only one arc) and then choosing the **Reorder Streams** command.

Reverse Directions

The **Reverse Directions** command can be used to reverse the direction of selected arcs. This is used for stream type arcs where the direction of connectivity is important. Stream arcs must always be defined by connecting point from downstream to upstream. If an error was made when creating the points this command can be used to correct it. An arrow is drawn on all arcs from upstream to downstream and can be used to verify that directions are correct.

Vertex to Node

The **Vertex ↔ Node** command in the **Feature Objects** menu can be used to create a node from a vertex or a vertex from a node (providing the node is connected to only two arcs). Vertices only define the geometry of the arc whereas nodes define the beginning and ending of an arc. Therefore when converting a vertex to a node the arc to which the vertex belongs is split into two separate arcs at the selected vertex. Likewise, when a node is converted to a vertex the two arcs attached to the node are merged into a single arc (if more than two arcs are attached to a node it cannot be converted to a vertex).

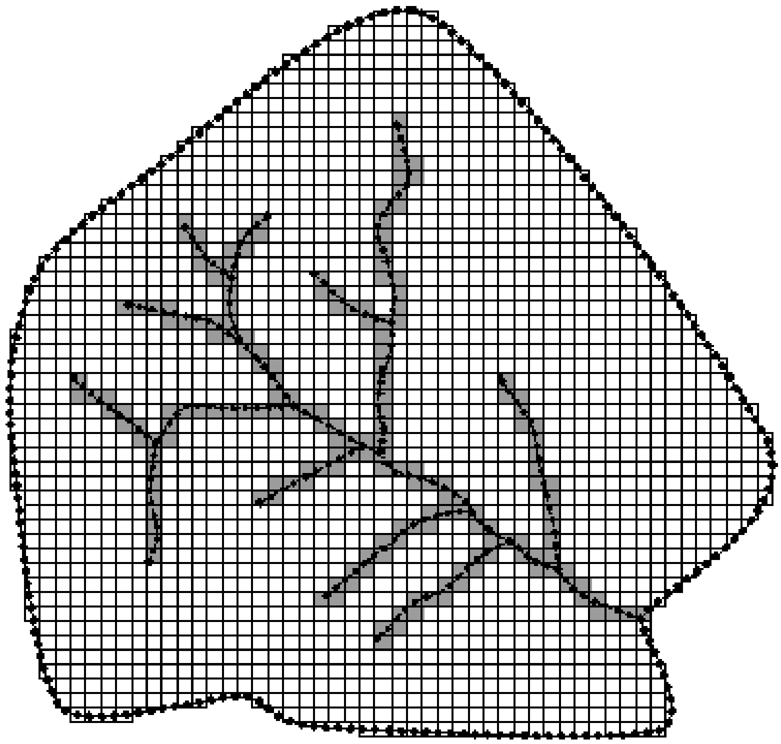
When converting a vertex to a drainage outlet for a DEM the vertex is automatically converted to a node first and then the attribute type is set to drainage outlet.

Related Topics

- Feature Objects
- Coverages
- Creating Watershed Models
- Creating TINs

Creating Grids

A grid can be created from a feature polygon using the **Create Grid** command in the **Feature Objects** menu. Active and inactive cells are determined from the boundary polygon. A rectangular grid is created that encompasses the bounds of the boundary polygon and cells outside the polygon are assigned an inactive status. Either an existing TIN or a DEM can be used as a background elevation map when interpolating z values for the elevation data set of the grid.



Related Topics

- Creating TINs
- Grids
- GSSHA Modeling

5.6. Hydrologic Modeling Module

Hydrologic Modeling Module

Hydrologic analysis is typically done using lumped parameter models such as HEC-1. The Tree module provides a graphical interface to HEC-1, TR-20, HSPF, TR-55, Rational Method, the National Flood Frequency (NFF), and other programs. In the absence of terrain data, topological or tree representations of a watershed can be created. Then all necessary input data to run one of the supported models can be defined using a series of user-friendly dialogs. This module is used for interfacing to hydrologic models and for the construction of topologic watershed models in the absence of digital terrain data.

This module is the primary module available for the public domain version.

Related Topics

- HEC-1
- HEC-HMS
- TR-20
- TR-55
- NFF
- Rational Method
- HSPF
- MODRAT (LA County Modified Rational Method)

Hydrologic Modeling Tools

The following tools are in the dynamic portion of the Tool Palette. These tools are available when the Hydrologic Modeling module is activated. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window with the cursor depends on the current tool. The tools are for selection and interactive editing of a topologic tree.

Select Outlet

The **Select Outlet** tool is used to select outlets for operations such as assigning routing or diversion data, creation of new outlets and basins, or deletion.

Select Basin

The **Select Basin** tool is used to select basins for operations such as assigning loss, unit hydrograph, precipitation, and other basin data, as well as deletion. It can be used to select basins from either the TIN or Tree representation of the watershed and behaves identically to the Select Basin tools from the Drainage module.

Select Diversion

The **Select Diversion** tool is used to select diversions for entering/editing diversion data or deletion. Diversions are displayed on the topologic tree only.

Select Hydrograph

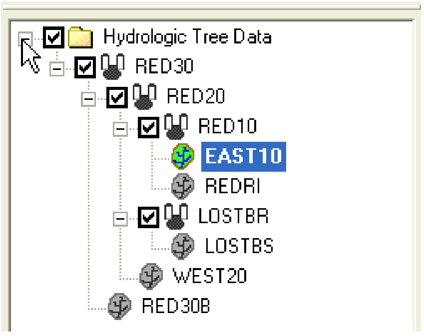
The **Select Hydrograph** tool is used to select hydrographs which can then be displayed in the Hydrograph Window. Multi-selection operations are available with this tool so that hydrographs from different locations can be overlaid.

Related Topics

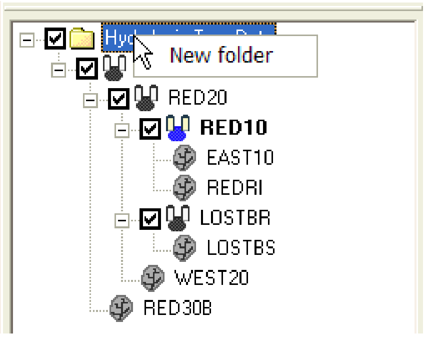
- Tool Palettes
- Hydrologic Modeling Module

Project Explorer Contents for Hydrologic Modeling Module

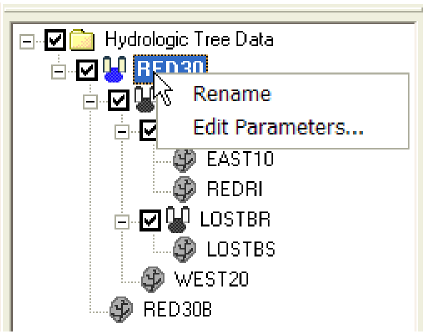
In the hydrologic modeling module the Project Explorer displays the identical structure as the topologic tree providing an alternate hierarchical view of your hydrologic model (diversions are not displayed in the Project Explorer). A check box controlling the display of an entire folder (model) and outlet points is displayed to the left of the icon. If you uncheck an outlet then the display of that outlet, it's basins and everything "upstream" of the outlet will not be displayed in the tree display of the graphics window.



Right clicking on the main Hydrologic Tree Data folder allows you to create a New folder (at this point a topologic tree for hydrologic model cannot be created directly from the Project Explorer).

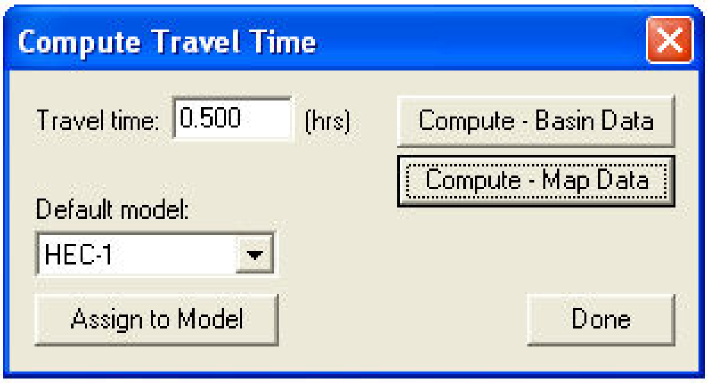


Right clicking on a basin or outlet icon allows you to **Rename** that basin (remember that HEC-1 names should be six characters or less) or **Edit Parameters** for the currently active model. Selecting a basin or outlet will cause it to be selected in the topologic tree of the graphics window.

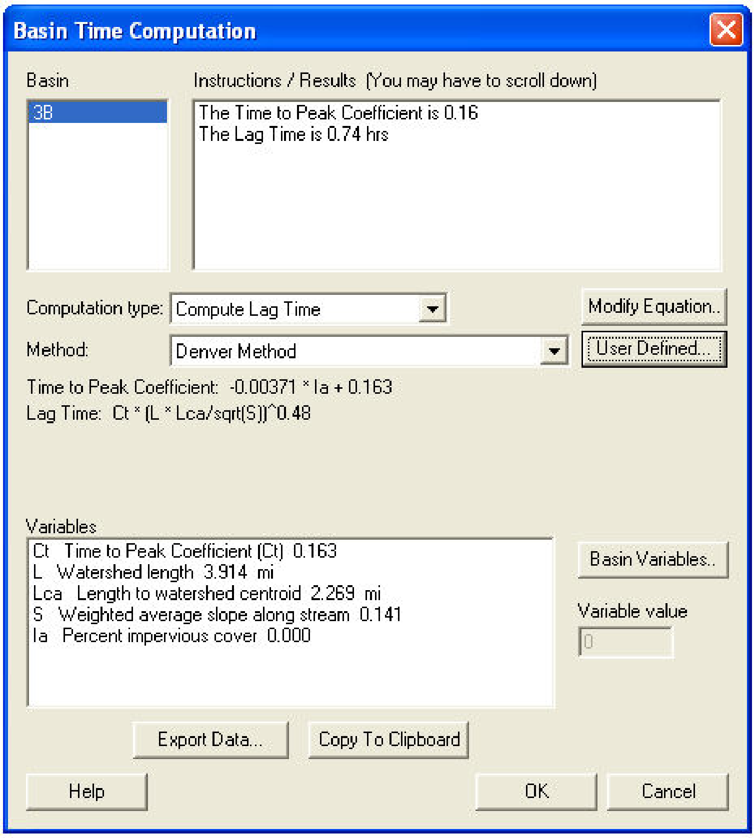


Combining Arc Travel Times

After defining equations and variables for individual flow path segments (arcs), the second step in computing a basin time of concentration (or lag time) is to sum the travel times of all arcs within a basin or between outlets. Selecting the basin or outlet and choosing the **Compute Travel Time** option from the **Calculators** menu in the Hydrologic Modeling module allows you to do this. This brings up the Compute Travel Time dialog. This option is also available directly from within many of the hydrologic model parameter dialogs where time of concentration, lag time, or routing travel time is needed.

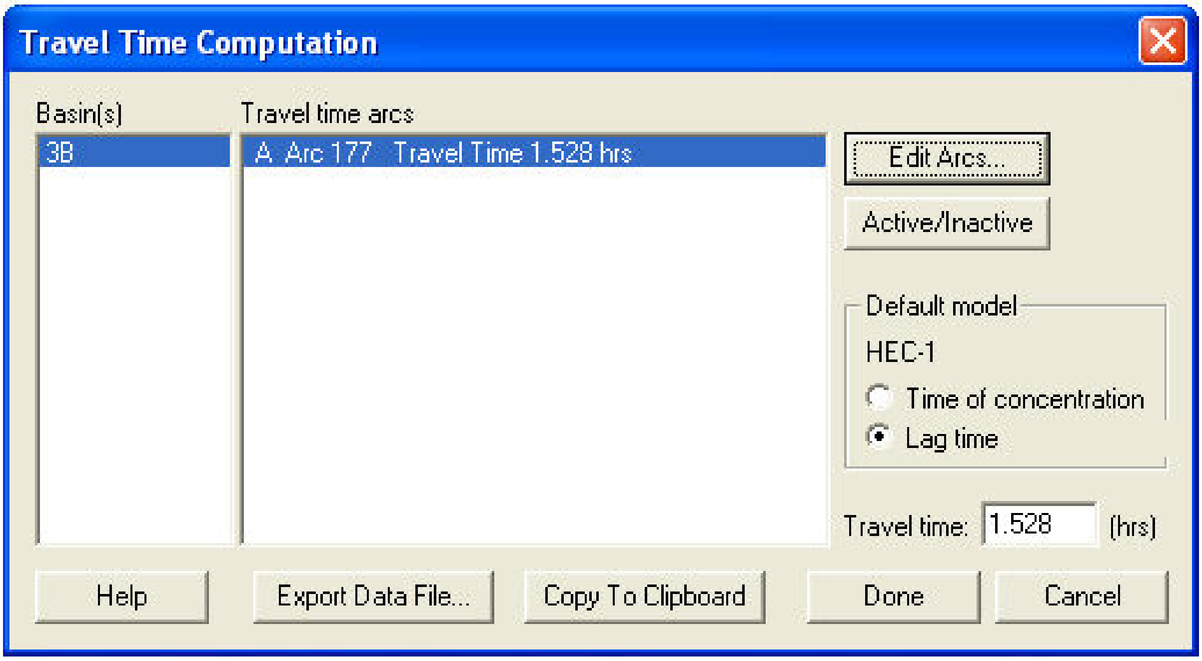


Selecting the Compute - Basin Data button brings up a dialog similar to the Assigning Equation to Arcs dialog where time computation types and methods can be selected. This dialog has a Modify Equation and a User Defined option where you can change parameters for various equations.



The Travel Time Computation dialog is accessed by selecting the Compute &ndsah; Map Data button from the Compute Travel Time dialog. When it is accessed, you will see the list of time computation arcs that lie within the currently selected basin if a basin is selected or between the selected outlet and the next downstream outlet if an outlet is selected. If you have already defined the equation and necessary variables, the travel time for each arc will be displayed and the total travel time for all arcs will be displayed in the travel time edit field. You can accept the computed value for travel time, or you may edit the value (override computed value) as may be appropriate. If you want to change the equation definitions or variable values for any/all of the arcs you can select the Edit Arcs button. This will bring up the Time Computation Arc Attributes dialog which is used for editing travel time equations and variables as described in the previous section.

The Default Model button can be selected to determine which set of model parameters is to be assigned the computed time of concentration or travel time.



In the Travel Time Computation Dialog, the Export Data File and Copy To Clipboard buttons are used to create a text report that summarizes the equation, variables, and computed time of concentration or lag time for the basin. Exporting the data will create a text file and allows you to either append to an existing file (so that a single report for multiple basins can be created) or create a new file. Copying to the clipboard places the report text on the Windows clipboard so that it is available for pasting into other documents.

Related Topics

- Travel Times from Map Data
- Assigning Equation to Arcs

Elevation Discharge Relationship

Discharge data for the basin/reservoir can be entered either by supplying an elevation vs. discharge pairs, or by defining any number and combination of spillways (weirs), outlets (orifices), and standpipes (weir-orifice combinations). The **Elevation Discharge Input** dialog is used to set up the discharge data.

If the Known-Discharge option is chosen you will need to enter a series of Elevation and Discharge values (you need the same number of values in each series) to define the relationship.

If the Discharge Structures option is chosen you can add any number of weirs, outlets, and standpipes along with their individual parameters. WMS will then compute an elevation discharge relationship with an appropriate elevation step and display it in the detention basin calculator. When defining a weir the parameters can be chosen from the available weirs in the weir calculator by selecting the **Weir Calculator** button.

If an HY-8 Culvert coverage is defined in the map module, you can use the HY-8 Culvert button in the Elevation Discharge Input dialog to define an HY-8 culvert crossing. The elevation-discharge computations for this crossing are then assigned to the WMS Elevation Discharge relationship. The hydrograph is routed based on the HY-8 computed elevation-discharge relationship. One important thing to remember when using the HY-8 computed elevation discharge relationship is that HY-8 only computes headwater elevations for the range of discharges you specify. You should be sure to specify a range of discharges in the HY-8 analysis that corresponds to the input hydrograph discharges.

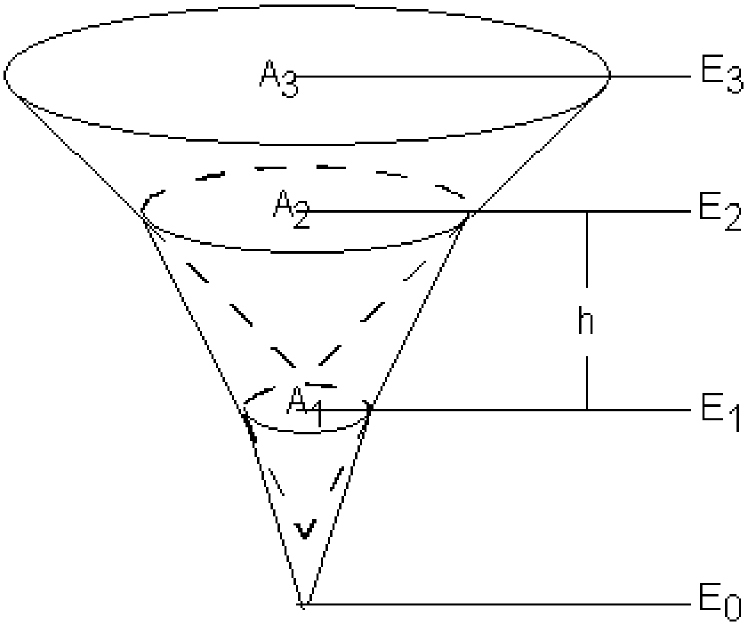
Related Topics

- Detention Basin Calculator
- Storage Capacity Curves

Storage Capacity Curves

There are three different methods for defining storage capacity: volume vs. elevation, area vs. elevation, and known geometry.

In all three cases a relationship between elevation and volume will be computed. For the volume vs. elevation option this is explicitly defined. If area vs. elevation is specified, then a corresponding volume for each elevation is computed using the conic method. The conic method is illustrated below.



The volume between incremental areas $A_1$and $A_2$is computed using the following equation:

$$\Delta V12 = \frac{h}{3}(A_1 + A_2 + \sqrt{A_1 A_2})$$

where:

$\Delta V12$ – The volume between areas $A_1$and A_2.

A_i – surface area i.

h – vertical distance $(E_2 - E_1)$between surface areas $A_1$and A_2.

E_i – elevation of surface area i.

The same equation is used to compute the volume between each adjacent set of surface areas, with the bottom area assumed to be 0. A TIN can be used to automatically create and store for use in the detention basin calculator the elevation-volume relationship.

If the basin geometry option is chosen then an elevation vs. volume relationship is computed directly from the geometry defined for the basin.

Related Topics

- Detention Basin Calculator
- Elevation Discharge Relationship

Computing Area Between Elevations

The **Compute Area Between Elevations** command is useful for determining areas in different elevation zones as part of a snow melt analysis. This operation can also be done when defining snow melt parameters for HEC-1. Model units are assumed to be either in feet or meters and subsequent areas are converted to square miles or square kilometers according to the metric flag set in the HEC-1 Job Control dialog.

This same procedure is also useful for determining storage capacity curves.

Related Topics:

- Snow Melt Simulations
- Storage Capacity Curves

5.7. River/Hydraulic Modeling Module

Hydraulic Modeling Introduction

The primary purpose of the hydraulic modeling interface within WMS is to process digital terrain and map data (TINs and coverages) to build the basic geometry necessary for a 1D Hydraulic Model. Much of the information for developing models with these tools is described in the information on River Tools in the Map module.

The general process for developing a model consists of the following steps:

1. Prepare a background digital terrain model that represents the river channel bathymetry and surrounding flood plain with enough detail to substantiate the modeling objectives.
2. Develop a 1D-Hydraulic Centerline coverage including the centerline and bank arcs.
3. Create the cross section arcs at important/required locations along the section of river being modeled.
4. An Area Property coverage can be used to map roughness values to line properties on the cross sections
5. Extract cross sections from the TIN and establish the 1D Model
6. Export the GIS data and finish defining HEC-RAS (or other model).

It is possible to establish the hydraulic model with extracting cross section information from a TIN. Cross sections which have already been surveyed can be used by assigning them to an arc. This, along with geo-referencing the data is done using the cross section editor from the *River Tools* menu in the Map module (when River Tools is the active model).

Default Hydraulic Model

The commands in the *Hydraulic* menu are used in defining any of the supported hydraulic models supported by WMS. The default model is controlled using the drop-down combo box in the *Edit Window*. Whenever you select a command for another model, the default model will be updated to specify that model.

Related Topics

- River Tools
- Using TINs
- 1D-Hydraulic Centerline Coverage
- Cross Section Coverage
- Area Property Coverage
- Editing Cross Sections
- Extracting Cross Sections
- Mapping Conceptual Models to River Schematics
- Recompute All Stations
- Exporting the GIS File

Hydraulic Modeling Tools

The following hydraulic modeling tools are found in the River module:

 Select River Cross Section

The **Select River Cross Section** tool is used to select a cross section and edit the associated parameters from within the hydraulic modeling module. It is equivalent to editing the attributes of a cross section feature line from within a *1D Hydraulic Cross Section* coverage in the map module.

 Select River Reach

The **Select River Reach** tool is used to select a reach and edit the reach data from within the hydraulic modeling module. It is equivalent to editing the attributes of a centerline from a *1D Hydraulic Centerline* coverage in the map module.

Related Links

- Tool Paletted

Hydraulic Schematic

A **hydraulic schematic** is a schematic tree-based representation of a map-based conceptual model. It represents a single hydraulic model. It is displayed as a node under the Hydraulic schematic data in the project explorer and is the key data type in the river module.

Hydraulic Toolbox 1.0

Executable and Library Downloads

- Download Hydraulic Toolbox 1.0 Installation (.zip) [1]
- Download C++ Library Redistributable files (.zip) [2]

Reference Papers

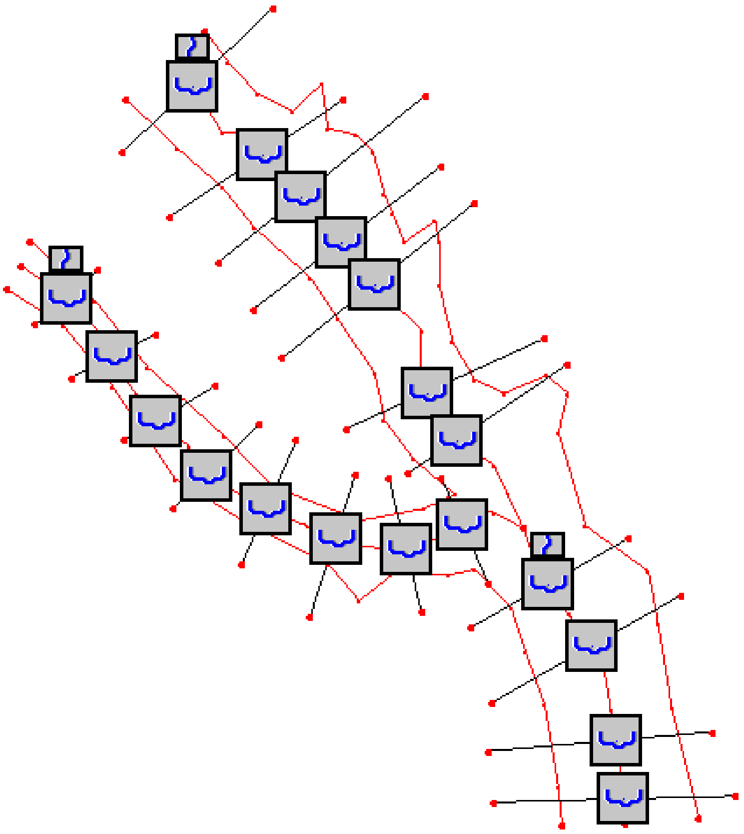
- Main WMS page

References

[1] http://wms.aquaveo.com/Hydraulic%20Toolbox%201.0.0.6%20Setup.exe
[2] http://utils.aquaveo.com/vcredist_x86.zip

Mapping the Conceptual Model to a River Schematic

WMS uses a conceptual model (coverages of centerlines, and cross-sections) to define the hydraulic model, but at some point this conceptual model must be mapped to an equivalent topologic model representation for a Hydraulic model. HEC-RAS for example is defined as reaches and cross sections. Each reach and section has appropriate stationing defined from the spatial nature of the conceptual model. When choosing this command (**Map → 1D Schematic** command in the *River Tools* menu), a schematic of the river is made for the appropriate model. The example below shows the reach (small boxes at the beginning of each reach) and section icons of the schematic for an HEC-RAS model. In the Hydraulic Modeling module the schematic is generally used as the view of the model.



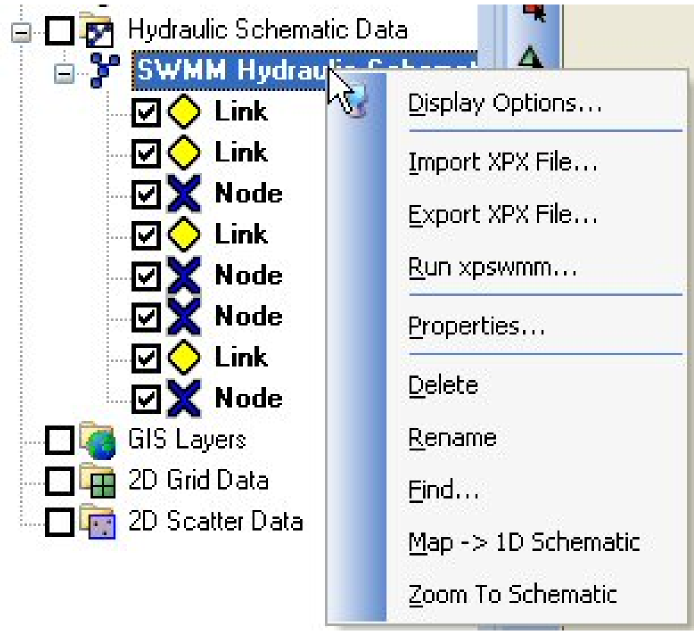
Related Topics

- Hydraulic Modeling Introduction
- 1-D Hydraulic Centerline
- 1-D Hydraulic Cross-sections

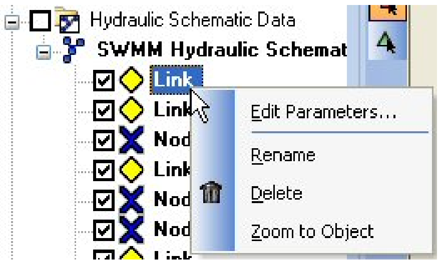
Project Explorer Contents for River Module

The project explorer contents for the river module include hydraulic schematics, cross sections, reaches, links, and nodes. Cross sections and reaches are used with HEC-RAS and SMPBDK models. Links and nodes are used in SWMM models.

Right-clicking on a hydraulic schematic gives you several options. The available options changes depending on the type of model you are working with. For example, right-clicking on a SWMM schematic brings up the following menu:



You can right-click on cross sections, reaches, links, or nodes either from the project explorer window or from the graphics window. Right-clicking on these objects brings up a menu similar to the following:



If you make a change to the map data (arcs) used to generate the hydraulic schematic, just re-generate the schematic from the by right-clicking on the hydraulic schematic and selecting the Map→1D Schematic menu item.

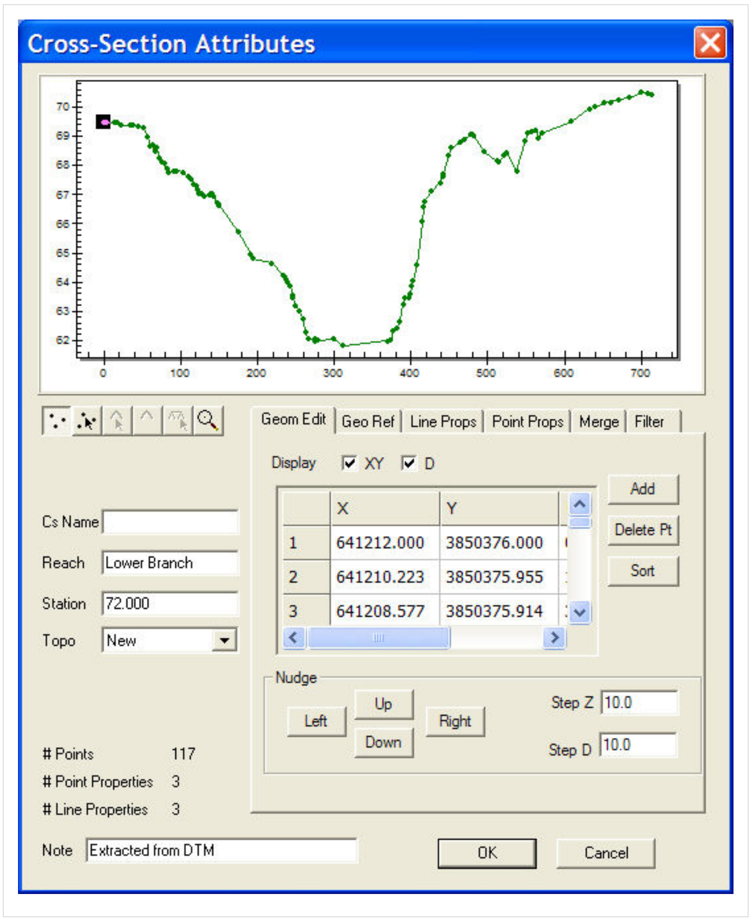
Editing Cross Sections

For the new 1D Hydraulic Cross Section coverage, the cross section geometry is stored in text database file on disk. When extracting cross sections they are saved to a new (or existing) database file. However, extraction of cross sections from digital terrain models is not the only way that they can be created, nor is extraction always the only thing that needs to be done. For example, other ways cross sections can be entered for use include importing from a spreadsheet, or entering manually. In such cases, and many times after extraction from a digital terrain model, there are edits that must be performed in order to prepare the cross sections for hydraulic modeling.

You can edit cross sections in one of three ways:

1. You can assign a cross section from a database by double clicking on an arc in a 1D Hydraulic Cross Section coverage. After assigning the cross section you can also enter the editor for that cross section.
2. You can open a cross section database for editing (or create a new database) using the Manage Cross Sections command.
3. You can also open an existing cross section database using the Open command in the *File* menu.

The operations described in the following paragraphs can be done using the cross section editor shown in the figure below.



General Properties

In order to identify information about the cross section in the database a name (not required), a reach, a station, and the name of the topographic data used to extract the cross section (if applicable) can be defined. A note about the cross section can also be defined. Not all of these attributes are critical for the development of a hydraulic model, but they are useful in managing the cross section within a database.

Editing Geometry

Cross section points can be added, or values edited when the *Geom Edit* tab of the editor is active. XY values are available when the actual 3D position of each point on the cross section is known. The more traditional D-Z pairs define the distance from the starting point and a corresponding elevation.

Geo-Referencing

Geo-referencing information provides the spatial (xy) location of the cross section and included geometry. This information is inherent in the 3D coordinates, when extracting cross sections from a digital terrain model. However, if the cross section geometry is taken from a survey then the actual xyz coordinates of the points may not be known. In order to use the data within WMS for flood plain delineation, a proper geo-reference must be provided.

A cross section can have one of the following georeferencing definitions:

- All points specified (extracted cross sections will be of this type)
- Use two points (e.g. the coordinates of the beginning and ending location along the cross section defined)
- Use one point and angle (e.g the centerline location is known and some angle relative to it defined)
- No georeferencing defined.

The geo-referencing is defined from the *Geo Ref* tab in the cross section editor.

Line Properties

Line properties define segments of material properties along the cross section. When using an area property coverage during extraction from a digital terrain model these properties are automatically marked and defined. However, they can also be established manually from within the *Line Props* tab in the cross section editor.

Point Properties

Point properties include thalweg, left bank, and right bank (other properties can be defined but are not mapped/saved to hydraulic models from within WMS) locations. When using a centerline and bank line arcs from a 1D Hydraulic Centerline coverage during extraction these points are marked. You can also have WMS "Auto Mark" these points by looking for the lowest elevation (thalweg), and appropriate breaks in elevation/slope (banks). Point properties are edited from within the *Point Props* tab in the cross section editor.

Merging

It is possible to combine a surveyed cross section with a section extracted from a terrain model for the flood plain (e.g. the terrain model does not contain enough detail to define the cross section of the river) using the tools in the *Merge* tab in the cross section editor. Two different cross sections can be merged, with rules for locations and precedence defined in order to create a new cross section.

Filtering

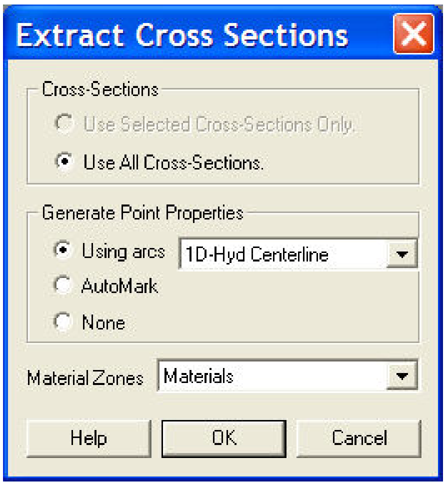
It may be that there are more points defining the cross section than are necessary (or that the hydraulic model is capable of processing). The *Filter* tab in the cross section editor allows you to specify rules for filtering "insignificant" points along the cross section. This can be particularly important when extracting cross sections from a dense digital terrain model.

Related Topics

- Managing Cross Section Databases
- Extracting Cross Sections
- 1D Hydraulic Cross Section Coverage
- 1D Hydraulic Centerline Coverage
- Cross Section Interpolation
- Interpolation of Results
- Hydraulic Modeling

Extract Cross Sections

The **Extract Cross Sections** command in the *River Tools* menu uses the cross section arcs and a digital terrain model (TINs are the only source that can currently be used) to extract the elevations at vertices of the feature arc cross-sections, or at the intersection points with the triangles.



Cross sections for individual arcs may be extracted by selecting the arc(s) before choosing the **Extract Cross Sections** command. If no cross sections are selected then the *Use All Cross-Sections* option is used.

Point properties (thalweg, left bank, right bank) can be defined from a 1D-Hydraulic Centerline coverage, or by AutoMark. The *AutoMark* option will examine the elevations of the extracted cross sections and try to infer the thalweg (low point) and the left and right bank points (change of slope) automatically.

Line properties can be determined from an area property coverage by intersecting the cross section arcs with the area property polygons and marking them in the cross section database.

Cross Section Database

When extracting the cross sections you will be prompted for the name of a cross section database file. WMS stores all of the cross section information in a text database file. The cross section database can also be edited independently using the **Cross Section Editor** tools. Extracting cross sections with feature arcs is only way to generate cross-section information, they also can be imported from spreadsheet files (cut and paste), or entered manually.

Related Topics

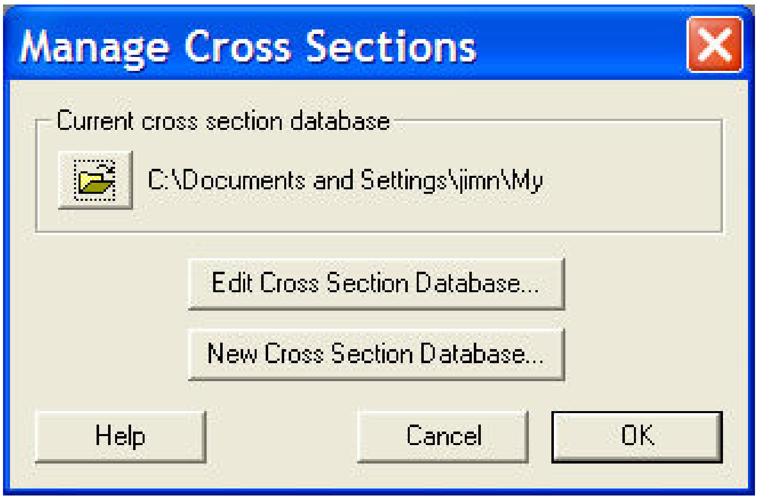
- Editing Cross Sections
- 1D-Hydraulic Centerline
- 1D-Hydraulic Cross-section
- Area Property

Managing Cross Sections

For the new 1D Hydraulic Cross Section coverage, the cross section geometry is stored in text database file on disk. When extracting cross sections they are saved to a new (or existing) database file. This database was the basis for the development of the cross section data in the ArcHydro data model. Cross sections in the database can be used for the development of hydraulic models such as HEC-RAS.

Extracting cross sections from a TIN is not the only way cross section geometry can be created. It is also possible to enter surveyed cross sections, however in order to use them in WMS for flood plain delineation, or modeling of any kind they must be georeferenced (tied to a geographic location). Further cross sections can be edited, merged, etc.

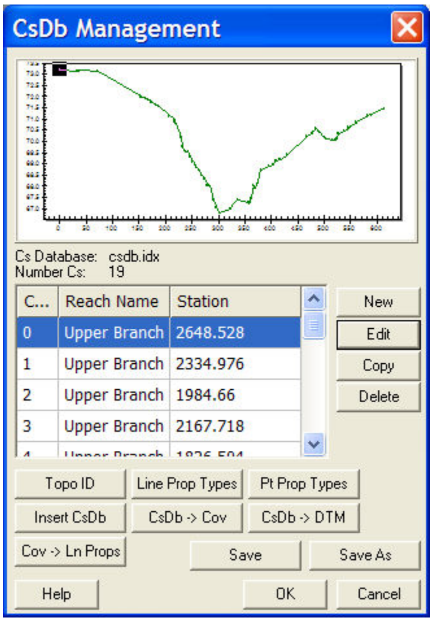
The **Manage Cross Sections** command in the *River Tools* menu allows you to create a new database or open an existing database to add geometries, edit existing ones, and provide proper georeferencing information. It is also possible to open a cross section database using the **Open** command from the *File* menu.



Cross Section Database Definition

When setting up a new database the following attributes can be defined as shown in the *CsDb* dialog below:

- Topo ID – a topographic identifier and description that identifies where the cross section database was derived from. You should create a new Topo ID for each database.
- Line Prop Types – By default WMS uses only a Material ID, but other properties could be defined for general use (they will not immediately be used by supported hydraulic models).
- Point Prop Types – By default WMS uses thalweg, left bank, and right bank but other point properties could be defined for general use.



The cross section database management dialog also allows you to create a new cross section, edit/copy/delete an existing cross section, insert an entire database (merge databases together), convert a cross section database to a coverage (of course the georeferencing of cross sections must be provided for the cross section to be included in the coverage), create a digital terrain model form the cross section geometry, and converting the coverage to line properties.

Related Topics

- Edit Cross Section Database
- Extract Cross Sections
- River Tools Overview (Hydraulic Modeling in WMS)

River Tools

The *River Tools* is not necessarily a model, but provides the tools and commands necessary to build 1D Hydraulic modeling data from feature object coverages. The River Tools primarily support the HEC-RAS model, but will be used to support the FHWA BriStars model as well as additional hydraulic models that will be supported in future versions.

The River Tools provide the ability to define a hydraulic model using a 1D-Hydraulic Centerline coverage and a 1D-Hydraulic Cross-section coverage. The layout of the feature objects defining the centerline and cross sections establishes the direction, the stationing, and the topology (connectivity between cross-sections) of a hydraulic model. Further, using these two coverages cross sections may be automatically extracted from a digital terrain model and then edited, merged, or combined with other cross section information to provide the geometric basis of the model. An area property coverage can also be used to map materials (Manning's roughness coefficients) to the cross section based on some type of aerial distinction (land use or soils) that may be available. The diagram below illustrates how these coverages are used to establish a hydraulic model.

The river tools also allow you to interpolate cross sections to establish more cross section information in between surveyed or extracted cross sections.

Results data from HEC-RAS and other hydraulic models can be read back in and used to perform a flood plain delineation. The flood plain delineation algorithm in WMS works better with a denser set of resulting water surface elevation points and so there are river tools that allow a water surface elevation computed at a cross section to be interpolated (copied since it will be the same value) along cross section arcs, or along a centerline.

Related Topics

- 1D-Hydraulic Centerline
- 1D-Hydraulic Cross-section
- Area Property
- Extracting Cross Sections
- Hydraulic Modeling
- Flood Plain Delineation

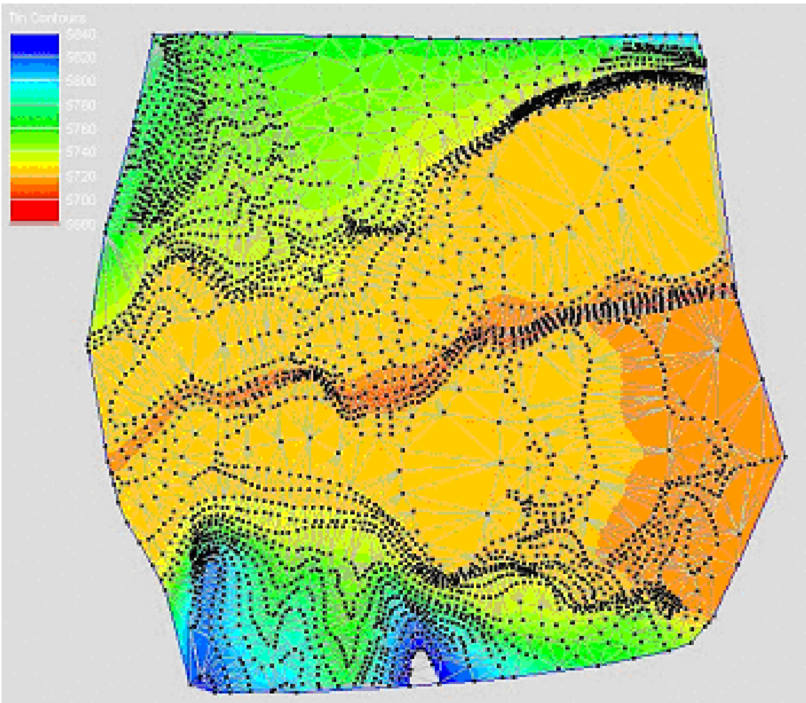
5.7.a. Flood Plain Delineation

Overview of Flood Plain Delineation

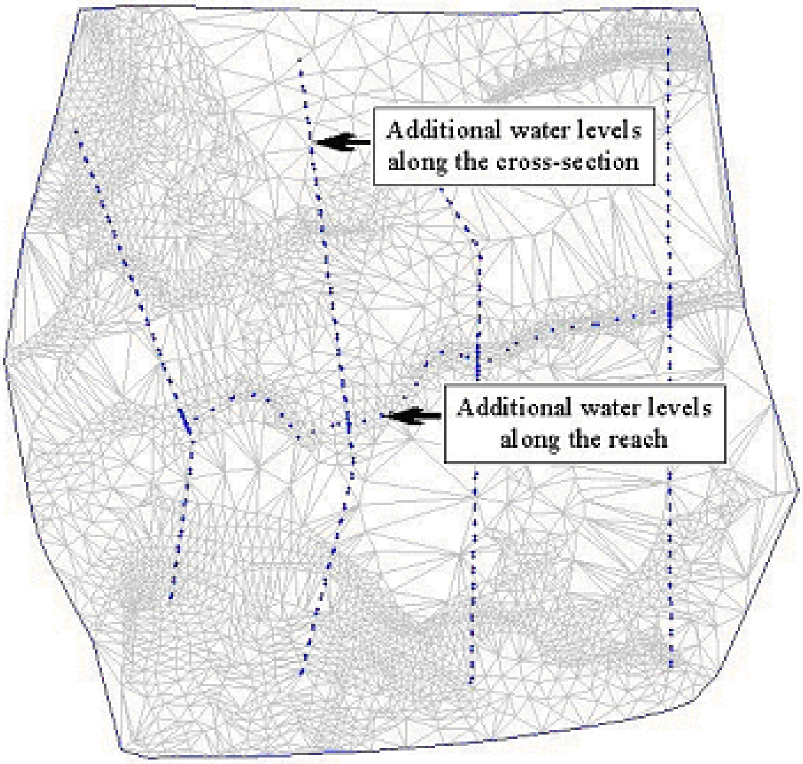
In addition to stream network and drainage basin delineation, WMS can also be used to perform floodplain delineation. Water levels simulated by a river hydraulic model or collected from different sources are read from a text file as a scatter dataset (see preparing stage data for more help). A smooth water surface is constructed by interpolating water levels at TIN vertices. User specified flood barriers such as embankments, roads, etc are also considered during this process. This surface is then intersected with the triangles in TIN representing the ground elevations, and the resulting set of edges defines the floodplain.

The basic steps to performing a flood plain delineation in WMS include:

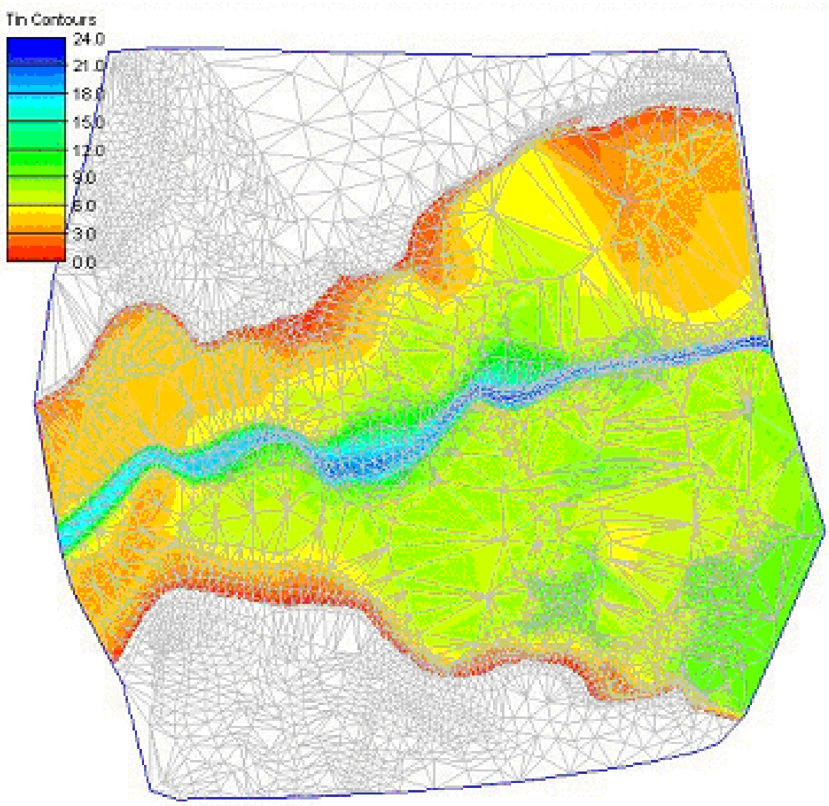
- Prepare a triangulated irregular network (TIN) surface of the area where the delineation is to be performed. This can be done by reading scattered elevation, converting from a DEM, or digitizing a contour map.



- Prepare your water surface elevation data. Water elevations data consists of a series of surface water elevations points defined as x, y, z (where z is the elevation of the water surface). Such points could be the results of a hydraulic model simulation, calculated in the WMS channel calculator, or retrieved from a known gaging station. They are stored as a scatter dataset.



- Select the appropriate options for delineating the flood plain, including the possibility of using a barrier coverage, and then delineate the flood.



- The result of the flood plain delineation will be a new dataset of water surface elevations and/or inundation depths. These datasets can be used to display contours on the TIN and converted to a series of output coverages (maps), including a flood depth map and impact maps derived from two separate delineations.

Stochastic Modeling

The flood plain delineation tools are connected with the HEC-1 hydrologic model and HEC-RAS hydraulic model to perform a series of floodplains based on the results of a series of model runs where rainfall, CN, and Manning's n are varied stochastically within a range of valid results.

Related Topics

- Differences From Earlier Versions
- Delineate Flood Plain
- Preparing Stage Data
- Stochastic Modeling
- Hydraulic Modeling
- Interpolation Options for Floodplain Delineation
- Simplified Dam-Break Analysis
- Flood Barrier

Differences From Earlier Versions (Version 6.0 and earlier)

The new method differs from the previous method in several aspects. The locations of water levels and their section criteria for interpolation are more flexible than the previous method. Ability to incorporate user defined flood barriers as coverage provides an excellent opportunity to overcome the limitations inherent in digital terrain models. It also becomes useful in evaluating “what if” or post project scenarios. The new method provides several options to present flood depth data that are not available in the older method. In addition to conceptual and computational differences between two methods, you will also notice following changes while using the new method:

- Water levels are read as a scatter dataset as opposed to flood stages at TIN vertices.
- The method does not require “streams” in the TIN.
- Multiple events or water level time series can be read as oppose to a single event. User can choose an event while delineating floodplain.
- User can specify flood barriers as features in the flood barrier coverage and the new method incorporates those features during flood depth computation.
- Computed flood depths are stored as TIN dataset and saved along with the TIN.
- Multiple flood depth datasets can be created in a TIN from multiple events.
- In addition to displaying flood depth as contours, this method can also create flood extent and classified flood depth coverage.
- It is now possible to compare two different flooding scenarios by creating a flood impact coverage.
- Finally flood extent, classified flood depth, and flood impact coverages can be exported as shapefiles for reporting or other flood management purposes.

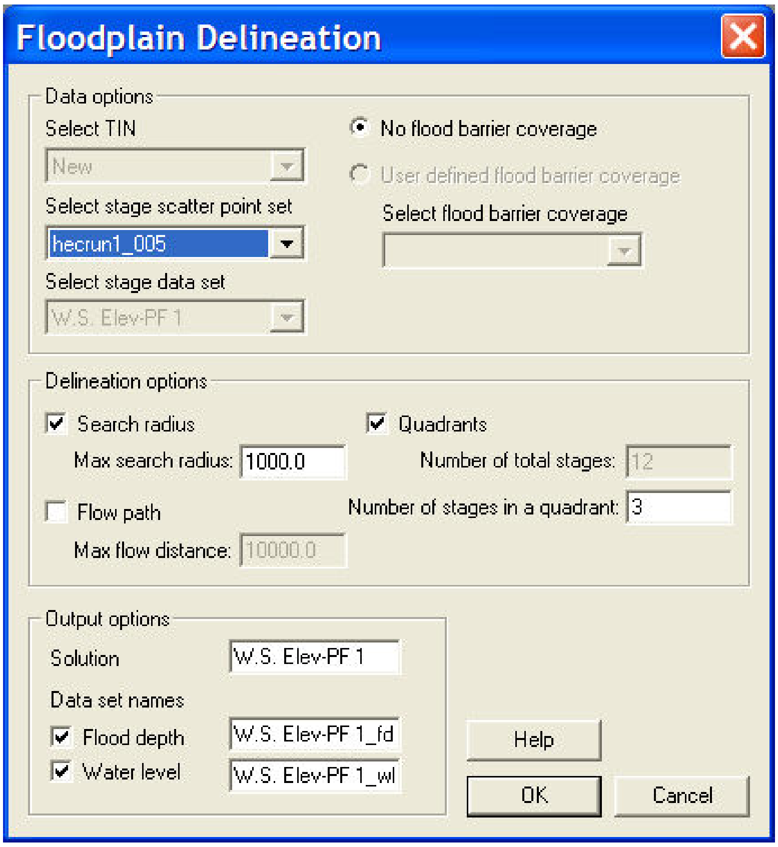
Related Topics

- Overview of Flood Plain Delineation

Delineate Flood Plain

There are several options that must be defined in the *Floodplain Delineation* dialog shown in order to delineate a flood plain in WMS. Each of these options are explained and described in detail below.

Select a TIN



Most of the time you will be working with a single TIN and therefore this option will be dimmed out. However, it is possible to have more than one TIN in WMS and in such cases you will need to specify the TIN on which the flood plain delineation will be performed.

Select the Stage Scatter and Dataset

Water surface elevation data used to derive the floodplain are imported and processed in WMS as scatter datasets. In addition to the scatter set (the xy locations of water surface elevation points), a particular dataset must also be specified. It is possible to have a single scatter set (locations of water elevations) with multiple datasets representing the water levels themselves. This will often occur when multiple scenarios of a hydraulic model are run. See the information on preparing stage files for more information about creating scatter datasets for floodplain delineation.

Select a Flood Barrier Coverage

A flood barrier coverage allows you to incorporate flood barriers representing natural or artificial barriers that are not represented explicitly by elevations in a TIN. The floodplain delineation process considers these barriers during water level interpolation. The resulting flood depth become closer to reality rather than a mere interpolation. For example, an embankment or a road is not always represented in a TIN.

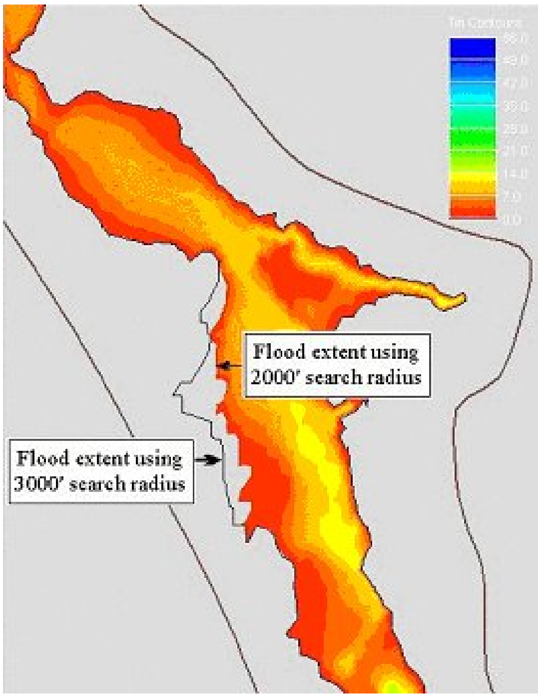
In order to delineate the floodplain properly, these barriers must be considered in a hydraulic model as well as in floodplain delineation process. This option not only incorporates such existing barriers in the process, it also provides the flexibility to professionals so that they can consider proposed structures and evaluate the “what if” scenarios.

Maximum Search Radius

Water surface elevations are determined for each TIN vertex by using interpolation from a set of "nearest" water surface elevations in the stage scattered dataset. The maximum search radius defines the limiting distance that will be used when collecting the nearest stage scatter points. If the *Use Flow Paths* option is turned on then the "radius" distance is the flow distance path, whereas if the option is turned off it is the straight line (as the crow flies) distance.

The floodplain delineation process offers several options for selecting water levels that are used in the interpolation. As expected, these options and the values of the parameters used in the process affect the resulting floodplain delineation. Therefore, care must be taken in selecting these options and appropriate values.

The effect of search radius and flow distance are demonstrated in the figure shown below. The figure shows the flooded areas delineated using a 2,000 foot search radius and flow distance in the shades of colors. The line represents the extent of flooding delineated using a 3,000 foot search radius and flow distance. For most places along the river these two flood extents coincide except in the west side of the middle portion. In that area a 3,000 foot search radius and flow distance resulted in more flooding than the 2,000 foot search radius and flow distance. This indicates the earlier the process could not compute flooding in that area because of the 2,000 foot limit. The water levels that could flood that area were discarded because they were outside of the 2,000 foot search radius or flow distance. To avoid this kind of problem, floodplains should be delineated by increasing the search radius and the flow distance until the flood extents stop changing. The final extent would then be the extent determined by the topography not by the search radius and flow distance.



Flooded areas delineated using a 2,000 foot search radius

Using Flow Paths

The flow path option in the floodplain delineation process ensures that the sources of water, i.e. the water levels, and the areas flooded (TIN vertices for which water surface elevation is interpolated) are hydraulically connected. This is an important option because if not applied, the process may interpolate water levels while ignoring obstructions between the water levels and the point of interpolation.

Quadrants

The quadrant option ensures the water levels are selected for interpolation from different directions instead of being biased by a particular direction. When using this option water elevations used for interpolation are selected equally from the four primary quadrants surround the point of interpolation. If no water levels, or an insufficient number, are found in a quadrant the process proceeds using that many fewer water levels for interpolation.

Number of Stages

A number of stages used for interpolation must be defined. If the quadrant option is turned on then you will need to specify the number of closest stage points to find within each quadrant. Without the quadrant option then the total number of nearest stages are specified. It is not required that the number specified be found, interpolation will proceed as long as one possible water elevation scatter point is found to meet the specified criteria. For example if 2 stages from each quadrant is specified, and one quadrant has zero possible choices and another quadrant only 1, then 5 points will be used in interpolation.

Resulting Datasets

The floodplain delineation tool generates two different types of datasets that can be used for contouring and further analysis with the TIN. First of all a dataset of water surface elevations at each TIN vertex contained in the flood plain is calculated and stored. Secondly TIN elevations are subtracted from the water surface elevations to create a flood depth dataset. Both, or either of the data sets can be specified for calculation. The data set(s) are TIN datasets and are managed by selecting the active dataset for the TIN.

Related Topics

- Overview of Flood Plain Delineation
- Differences From Earlier Versions
- Preparing Stage Data
- Stochastic Modeling

Preparing Stage Data

There are three basic ways to create stage data for flood plain delineation. The tutorial on floodplain delineation demonstrates all three methods.

Hydraulic Model Data

When running HEC-RAS or other supported hydraulic models, resulting water surface elevations are created as a scatter set for each cross section in the model. Typically the number of scatter points created from the cross sections is insufficient to adequately interpolate a flood plain on the TIN and so these points need to be interpolated along the river center line and cross sections in order to create a scatter set with sufficient points for interpolation. The additional points along cross sections are created with the same value as the first point which does not violate the assumptions of a 1D model like HEC-RAS. Additional points along center lines are created by linearly interpolating from the cross sections (again being consistent with the assumptions of 1D models).

Read a scatter dataset

Scatter sets can be read in as 2D scatter files, or imported using the text wizard if existing data can be created in a spreadsheet or other consistent text file format. The interpolation tools for cross sections and center lines can also be used after a file has been read.

Manually create scatter points by digitizing in WMS

The 2D Scatter module provides a tool for creating new scatter points interactively. Water surface elevation can be entered for as the data value. Generally in such cases you will want to take advantage of a centerline and cross sections to develop a larger dataset for flood plain delineation. For example if you create a 1D-Hydraulic centerline to represent your stream, and optionally cross sections you can interpolate computed values along these feature objects.

Use the Channel Calculator

This is actually similar to the manual method. A new scatter point can be created along a cross section arc at the intersection point of a centerline using the water surface elevation (computed depth plus lowest elevation along the cross section) computed with the channel calculator.

Related Topics

- Read a Stage File
- Importing Text
- Interpolating Hydraulic Model Results
- Floodplain Delineation
- Channel Calculator

5.8. GIS Module

GIS Module

The GIS module has been separated from the Map module in order to define a more integrated and separate approach to linking with GIS data. The GIS module has two separate modes, although the primary functions are available in either mode. The main reason that the GIS data has been separated from the map module is to allow users to handle large files more efficiently when creating hydrologic models. For example the GIS module allows you to import large files and then select and convert to feature objects only the portions that are needed. The conversion to feature objects is both time consuming and potentially memory intensive, and so managing your GIS data in this fashion is more efficient (the equivalent GIS functionality is to clip out just the parts of the data you need for import).

Some of the key functionality available in either mode includes:

- Efficient management of large datasets
- Graphical selection of features
- Mapping of selected features to feature objects in map coverages
- Viewing attribute tables
- Joining additional attribute tables based on a key field (i.e. joining the hydrologic soils group attribute to a STATSGO/SSURGO shapefile).

Using the GIS Module with a License of ArcView®

WMS uses the ArcObjects to incorporate much of the ArcMap functionality directly. You can open any file format (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) that is supported by ArcView® and you use all of the ArcView® Display Symbology properties to render the GIS data. WMS actually uses ArcView® to display the GIS layers and then copies the bitmap generated by ArcView® into WMS.

Using the GIS Module without a License of ArcView®

Most of the same functionality that exists for users with licenses of ArcView® is available to users without. The primary differences are that you can only read layers that are in the shapefile format, and all of the display and symbology available with ArcView® is not available. Points, lines, and polygons are displayed in a single color and not filled. Further some of the queries for selection are not supported without a license to ArcView®.

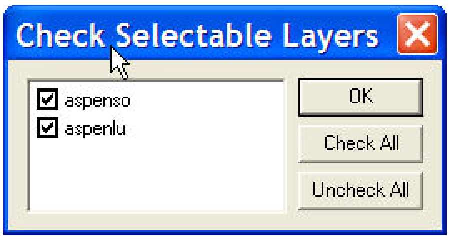
Related Topics

- Project Explorer Contents in the GIS Module
- Map Module
- ArcObjects
- Shapefiles

GIS Tools

Select Features

The **Select Features** tool selects features from GIS layers when ArcObjects has been enabled (the Select Shapes tool is used when ArcObjects is not enabled). All features from selectable layers will be selected as long as the layer is visible. A layer can be turned off from the Project Explorer so that it is not visible, and therefore not selectable. The layers can also be made un-selectable by using the Selectable Layers... command in the Selection menu. The following dialog allows you check or uncheck layers as being selectable.

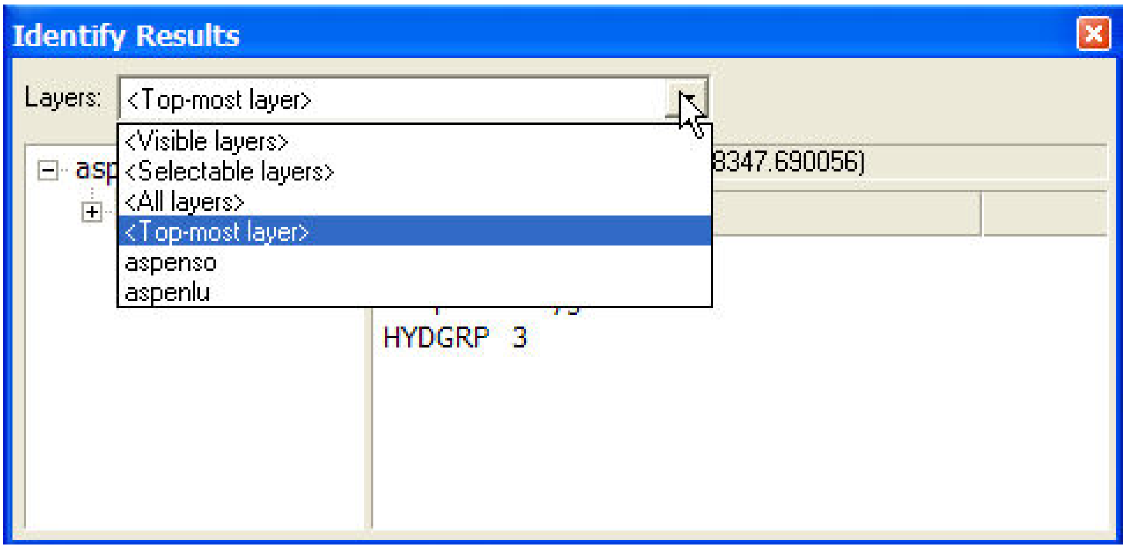


Individual features can be selected to create a new selection, added to the current selection, taken from the current selection, or selected from within features already selected as specified by the Interactive Selection method defined from the Selection menu (by default the selection method is create a new selection).

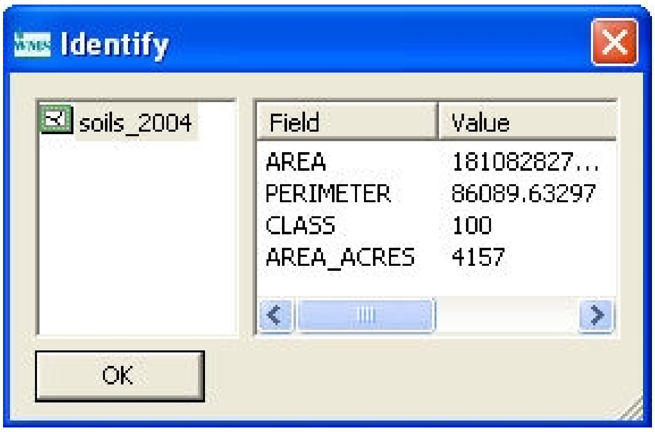
Multiple features may also be selected by dragging a box around all features that you want selected. In such cases all features that have any portion within the selection rectangle will be selected.

Identify

The Identify tool can be used to view the attributes of selected features (points, lines, or polygons in a GIS layer) in a GIS layer. When selecting a feature the attributes are displayed in the Identify Results window. By default, with ArcObjects enabled, only the attributes of the selected feature in the top most layer (as displayed in the WMS Project Explorer) are displayed, but the Layers drop down combo can be changed to All Layers, or a specific layer as shown in the figure below.



With ArcObjects not enabled, attributes of the selected feature in the top most layer (as displayed in the WMS Project Explorer) are displayed, while features from other shapefile layers can also be viewed by selecting them in the dialog as shown in the figure below.



Select Shapes

The Select Shapes tool is used to interactively select shapes when ArcObjects is not enabled. Individual shapes may be selected using this tool, and if you hold the shift key down, newly selected shapes will be added to the selection list. A rectangle can also be dragged and all shapes that have any part within the rectangle will be added to the selection list.

Related Topics

- GIS Module
- Selection with ArcObjects Enabled
- Mapping GIS Features to Feature Objects

Project Explorer Contents for GIS Module

GIS layers are organized and displayed in the GIS module. This module can be used for displaying and converting GIS data, but data in this module is not meant to be used for watershed analysis directly.

There are two general types of GIS layers that can be displayed, and there are various ways of reading GIS data. The two general types of GIS data include raster and vector GIS data. WMS can read either raster or vector GIS data into the GIS module and these data can be converted to various formats for use both inside and outside of WMS.

To understand how data is stored and projected under the GIS module, it is important to understand how WMS reads the various types of raster and vector GIS data.

First, WMS reads and displays point, line, and polygon shapefiles as GIS vector data. These data can be converted to feature objects and WMS has a rich set of options for preserving, modifying, and converting attributes and geometry from shapefiles to feature objects in the map module of WMS.

Second, WMS has tools for reading, displaying, and converting several formats of raster and vector GIS data to other formats. WMS uses the GlobalMapper [1] library of functions to read and convert nearly all the formats that can be read and displayed by GlobalMapper. This functionality is available for all WMS users who have a license that includes the GIS module. The GIS vector data read as a GlobalMapper object can be converted to a WMS point, line, or polygon shapefile type so the data can be converted to feature objects and used in your hydrologic modeling projects. When you select the Get Online Maps button in WMS, WMS loads an online map that is either a standard image, a palette-based raster (such as a land use grid), or an elevation raster (such as a DEM).

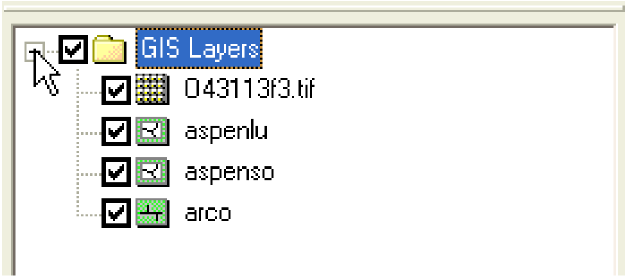
Finally, if you have a current license to ArcMap that includes ArcObjects and you have enabled ArcObjects, WMS can read any data that ArcGIS can read. If ArcObjects is not enabled, WMS can still read several raster and vector data formats through GlobalMapper. However, some formats only supported by ArcObjects cannot be read.

Each type of data has a similar but different right-click menu that allows you to access some of the functions that are available. Some functionality is also available in the GIS module menus. Some menus are only available if ArcObjects has been enabled.

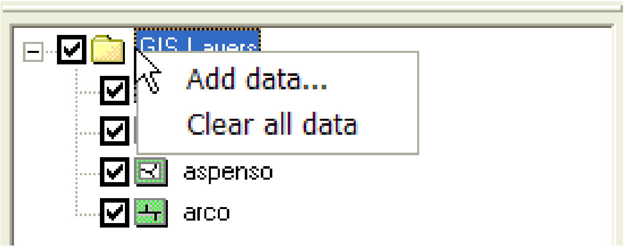
A summary table listing the various types of data that can be read and displayed and some of the functionality available for the various types of data is shown below:

Description	Icon	GIS Data Type	Functionality	Licensing Requirements
WMS Point Shapefiles		GIS Vector Data	• Remove/Rename/Zoom To Layer • Convert to Map Module feature objects • View object and entire shapefile attributes • Join .dbf file with the vector layer using a common ID • Convert to Vector GIS Data (GlobalMapper)	WMS License with Map Module
WMS Line Shapefiles		GIS Vector Data	• Remove/Rename/Zoom To Layer • Convert to Map Module feature objects • View object and entire shapefile attributes • Join .dbf file with the vector layer using a common ID • Convert to Vector GIS Data (GlobalMapper)	WMS License with Map Module
WMS Polygon Shapefiles		GIS Vector Data	• Remove/Rename/Zoom To Layer • Convert to Map Module feature objects • View object and entire shapefile attributes • Join .dbf file with the vector layer using a common ID • Join NRCS data (joins tables specific to NRCS SSURGO data with the shapefile) • Convert to Vector GIS Data (GlobalMapper)	WMS License with Map Module
GIS Vector Data		GIS Vector Data	• Remove/Rename/Set Transparency/Zoom To Extents/Open Containing Folder • Export to various supported formats • Automatic reprojection based on current projection • Convert to WMS point, line, and polygon shapefiles • Convert point or line elevation data to feature objects, 2D scattered data, and TIN formats if fields beginning in "elev", "valdco", or "contour" exist.	WMS License with Map Module
Generic GIS Raster Image		GIS Raster Data	• Remove/Rename/Set Transparency/Zoom To Extents/Open Containing Folder • Export to various supported raster formats • Automatic reprojection based on current projection • Export an image world file • 2 or 3-point registration based on pixel/real-world coordinates • Crop/Uncrop image collars	WMS License with Map Module
Color Palette-based GIS Raster		GIS Raster Data	• Remove/Rename/Set Transparency/Zoom To Extents/Open Containing Folder • Export to various supported raster formats • Automatic reprojection based on current projection • Export an image world file • 2 or 3-point registration based on pixel/real-world coordinates • Crop/Uncrop image collars • Convert to a WMS Soil Type or Land Use Grid • Extract land use attribute information	WMS License with Map Module

Elevation-based GIS Raster		GIS Raster Data	• Remove/Rename/Set Transparency/Zoom To Extents/Open Containing Folder • Export to various supported raster formats • Automatic reprojection based on current projection • Convert to a WMS DEM or Rainfall Grid	WMS License with Map Module
Online Generic GIS Raster Image		GIS Raster Data	• Delete/Rename/Set Transparency • Export to various supported raster formats • Automatic reprojection based on current projection • Dynamic layer-runs in a separate thread and changes with current view	WMS License with Map Module
Online Color Palette-based GIS Raster		GIS Raster Data	• Delete/Rename/Set Transparency • Export to various supported raster formats • Automatic reprojection based on current projection • Convert to a WMS Soil Type or Land Use Grid • Extract land use attribute information • Dynamic layer-runs in a separate thread and changes with current view	WMS License with Map Module
Online Elevation-based GIS Raster		GIS Raster Data	• Delete/Rename/Set Transparency • Export to various supported raster formats • Automatic reprojection based on current projection • Convert to a WMS DEM or Rainfall Grid • Dynamic layer-runs in a separate thread and changes with current view	WMS License with Map Module
ArcObjects Point File		GIS Vector Data	• Delete/Rename/Zoom To Layer • Convert to Map Module feature objects • View file properties • View object and entire shapefile attributes • Set Layer Transparency • Join GIS Table to Layer based on a common ID • Ability to select features by attribute or location	WMS License with Map Module, Current ArcMap/ArcGIS license
ArcObjects Arc File		GIS Vector Data	• Delete/Rename/Zoom To Layer • Convert to Map Module feature objects • View file properties • View object and entire shapefile attributes • Set Layer Transparency • Join GIS Table to Layer based on a common ID • Ability to select features by attribute or location	WMS License with Map Module, Current ArcMap/ArcGIS license
ArcObjects Polygon File		GIS Vector Data	• Delete/Rename/Zoom To Layer • Convert to Map Module feature objects • View file properties • View object and entire shapefile attributes • Set Layer Transparency • Join GIS Table to Layer based on a common ID • Ability to select features by attribute or location	WMS License with Map Module, Current ArcMap/ArcGIS license
ArcObjects Raster File		GIS Raster Data	• Delete/Rename/Zoom To Layer • View file properties • View data • Set Layer Transparency • Write an ArcInfo ASCII DEM file • Convert to a WMS DEM, Land Use Grid, Soil Type Grid, or Rainfall Grid	WMS License with Map Module, Current ArcMap/ArcGIS license that includes the Spatial Analyst
ArcObjects CAD/TIN File		GIS CAD/TIN Data	• Delete/Rename/Zoom To Layer • View data • Set Layer Transparency	WMS License with Map Module, Current ArcMap/ArcGIS license that includes the Spatial Analyst

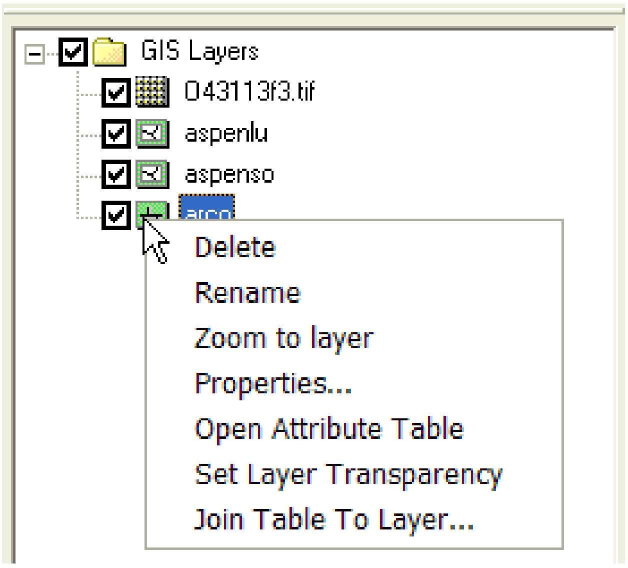


Right clicking on the main GIS Layers folder allows you to Add the various types of GIS data (open new GIS layers) or Clear all data. If ArcObjects is enabled WMS cannot read the GlobalMapper GIS data types. Right-click on the GIS Data folder and clear all your GIS data to disable ArcObjects and read GlobalMapper GIS data using the Add GIS Data button or menu. With ArcObjects enabled, you have access to any type of data that can be read by ArcMap and to some of the functionality of ArcMap from inside of WMS.



Right clicking on a GIS layer allows you to **Delete**, **Rename**, **Zoom to layer**, view/set **Properties**, **Open Attribute Table**, or Set Layer Transparency. If the layer is a vector data layer then you can also **Join Table to Layer**, whereas if the layer is a grid you can **Write DEM ASCII File** or **Convert to DEM**.

The properties are the identical properties available from ArcGIS.



Related Topics

- GIS Module
- GIS Tables
- Coordinate Conversions

References

[1] http://www.bluemarblegeo.com/global-mapper/

Deleting GIS Data Layers

A GIS Data layer can be deleted by right-clicking on the layer folder in the Project Explorer and selecting the Delete command.

Related Topics

- Add Data
- Add Shapefile Data
- Project Explorer Window

Computation Step

The computation step is only used when defining composite curve numbers or runoff coefficients for a drainage coverage. If a TIN is used, then individual basins are composed of several triangles and each triangle can be assigned a land use and/or soil type. However, for a drainage coverage, each basin is typically represented by a single polygon. The computation step is used to divide each basin polygon into a number of square cells (the computation step being the length of a side) that are each assigned a land use and or soil type ID. The smaller the step length the more accurate the composite number will be, but the more time consuming the computation as well.

Related Topics

- Compute GIS Attributes

GIS Tables

Open Attribute Table

The **Open Attribute Table** command, available by right-clicking on a GIS layer, opens up a spreadsheet dialog that shows all of the attribute names (as columns) and the values associated with each feature in the layer (as rows). Tables cannot be edited within WMS; this is only a way to explore and discover the nature of the database associated with the GIS layer. It is important to understand both the names of attribute fields and feature values in order to better understand how the data might be mapped to feature objects in a map coverage.

Joining Tables

The **Join Table to Layer** command, available when right-clicking on a layer in the *Project Explorer*, allows you to join the attributes of one database file (.dbf) to the features of a GIS layer based on a key attribute field. This is particularly important when dealing with NRCS soils data since the features are stored in a shapefile with a minimal set of attributes, and the hydrologic soil group and other soil classifications are stored in a separate .dbf file. The two files are related based on an attribute field named MUID. Other GIS data layers may be similar where the features contain some kind of key indexing field and the attributes are stored in a separate table that can be joined to the features based on the index field values.

After selecting the **Join Table to Layer** command you will be prompted for the database file you wish to join using the standard select file dialog. The *Join Table* dialog will then appear and you will be asked to select the *Join Field* from the GIS data layer attributes and the *Join Field* from the table you are joining to the GIS data layer. Often these field names will be the same as in the example below, but they are not required to be the same. The important thing is that they contain similar information from which a join can be made. Finally, you can select to join all of the attributes from the join table or just add a specific field, as is shown in the example below where we are only interested in adding the field representing hydrologic soil groups.

The join does not permanently alter the GIS data layer on the hard drive of your computer, it only exists within the WMS application.

Related Topics

- GIS Module
- Soil Coverages
- Mapping to Feature Objects

Soil Type

In the *Compute GIS Attributes* dialog, the soil type option determines whether a soil type coverage or a soil type grid will be used. The soil data has a slightly different meaning depending on whether CN or runoff coefficients will be computed. For CN the critical attribute is the hydrologic soil type, whereas for runoff coefficients the critical attribute is a soil ID that can be related to a table of runoff coefficients.

WMS uses integer values to store the hydrologic soil type (0-soil A, 1-soil B, 2-soil C, 3-soil D), however when reading from a database file that is associated with a shapefile WMS will read the letter values and assign the appropriate integer ID. The standard soil shapefiles distributed by the NRCS store the hydrologic soil group parameter in a separate database file than the feature polygons. This file must be joined to the attributes of the feature polygon in order to extract the hydrologic soil group parameter. In previous versions of WMS, this step had to be done outside of WMS using a GIS program like ArcView®. These tables can now be joined directly within the GIS module of WMS.

Related Topics

- Compute GIS Attributes
- Land Use
- Mapping Tables
- Soil Type Coverage
- Obtaining Soil Type Data
- Joining Tables

Creating Watershed Models

With an ever-increasing availability of GIS and other digital data, delineated stream networks and basin boundaries for a given watershed may already exist. In order to take advantage of this type data when available, WMS allows hydrologic models to be built directly from three different features of the map module: polygons representing basin boundaries, arcs representing a stream network, and nodes representing watershed and sub-basin outlet points.

This means that data imported from an ArcView® shapefile can be used directly to set up the hydrologic model. Further, since attributes from the shapefiles can also be read in, much of the hydrologic data developed with the GIS tool can be used to define input parameters of the given hydrologic model. It also means that a tiff image map or other data can be used to establish the boundaries of the watershed at the proper scale so that lengths and areas determined from the feature objects are correct, or simply used as a scaled schematic representation of the watershed (in such cases area and length values would have to be determined by some other means and defined in appropriate dialogs prior to running one of the supported hydrologic models).

Related Topics

- Feature Objects
- Feature Object Guidelines
- Drainage Coverage
- Shapefiles

5.8.a. ArcObjects/ArcView

Introduction to ArcObjects

ArcObjects is a development platform provided by ESRI that allows developers of other applications (such as WMS) to incorporate ArcView/ArcGIS® capability directly within their application. WMS has the ability to use some parts of ArcView® as an integrated part of the WMS application. This allows users to access some of the same functionality in WMS that is available in ArcView®, but this is only providing you are running on a computer that has a current license of ArcView®. Without a license, much of the same functionality is available, the primary differences being that only the shapefile format is supported, and many of the selection and display capabilities are minimal.

Enabling ArcObjects

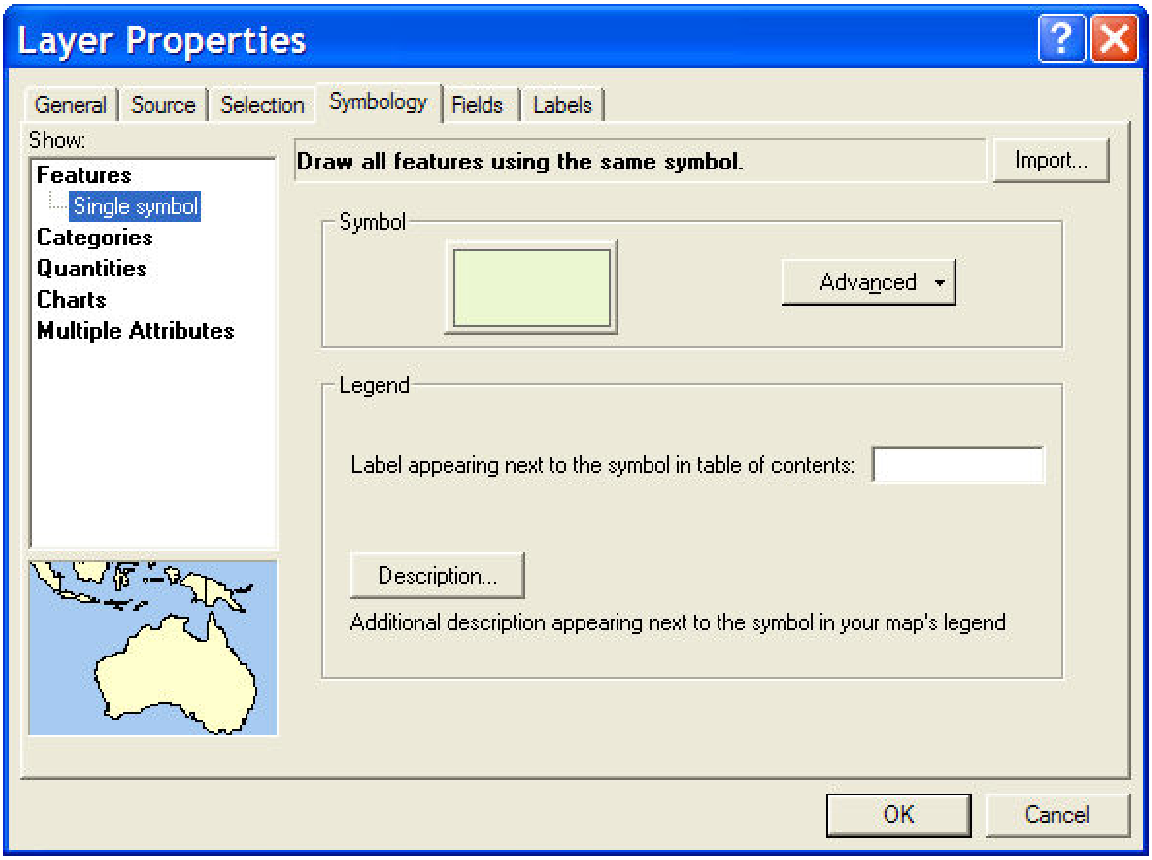
The **Data | Enable ArcObjects** command queries the ESRI license manager for ArcView/ArcGIS® to see if a license for version 8.x exists. If a valid license is found then the ArcView® functionality within WMS is enabled and you will be allowed access. If a license is not found then the ArcView® functions remain dimmed, but you are still able to do many of the same things, the primary limitation being that only shapefile format is supported for reading GIS data.

ArcObjects not Enabled

Without a license to ArcView®, the display capability of shapefile layers is minimal. The primary intent of using the shapefile layers is as a temporary holding place to convert your data to feature objects. As such, the full symbology capability of ArcView® has not been implemented within WMS. All shapes are displayed using black lines.

ArcObjects Display Options

With ArcObjects enabled you have access to the ArcView® Layer Properties dialog. This dialog can be used to specify the symbology of GIS data layers and is accessed by right-clicking on the layer from the Project Explorer and choosing Properties.



When displaying GIS layers using ArcObjects the properties and symbology defined in the Layer Properties dialog are used by ArcObjects to render the GIS layer. ArcObjects renders the features and passes a bitmap to WMS. WMS then renders the rest of the current data (e.g. DEMs, TINs, feature objects, etc.) on top of the data displayed by ArcView®.

Transparency can be defined for a layer, but it will only affect the display that is rendered by ArcObjects, and not the rest of the data rendered by WMS.

Related Topics

- GIS Module

ArcObjects Add Data

The **Add Data** command is available when ArcObjects is enabled and uses the same dialog resource to open GIS data layers that is used by ArcView®.

When ArcObjects is enabled you are able to load any of the ESRI supported formats, including shapefiles, coverages, geodatabases, grids, images, CAD files and others, as GIS data layers in WMS. These data can then be converted to WMS feature objects in map coverages.

Related Topics

- Enabling ArcObjects
- Mapping GIS Features to Feature Objects
- Adding Shapefile Data

ArcView Data Guidelines

Because the data structures used for the three primary methods of hydrologic data development in WMS parallel data types found in GIS software such as ARC/INFO® and ArcView®, there are several possible scenarios for importing and using data developed by GIS applications that support the ArcView® file formats.

Shape Files

There are nearly as many different ways to have stored watershed data in a GIS as there are watersheds stored. Therefore, what you do with basin polygons or stream networks in WMS will likely be somewhat different. This section outlines some of the key issues involved in importing shapefiles (ARC/INFO® or ArcView® data) and provides examples of common problems.

Fundamental to importing any vector data layer is the ability to map attributes associated with the shape file to corresponding parameters used by WMS. The same dialog is used for each of the three basic layer types. A set of key words can be used to define the item names of attributes in the shape file so that mapping to corresponding variables in WMS occurs automatically. In the event that the attribute name is different, the fields can be manually mapped. Regardless of the way your data is stored in the GIS, you should be able to take advantage of as much of the predefined and stored hydrologically related parameters as possible.

Ideally you will have three data layers when importing watershed related shapefiles: 1) a polygon layer representing basin boundaries, 2) a line (arc) layer representing the stream network, and 3) a point layer representing the outlet points (these should be the intersection points of the basin polygons and stream network layers). If you have these three layers properly defined you should be able to import them and automatically create the topologic model used for hydrologic modeling in WMS. If one or more of these layers are not present you will need to either create it in ArcView® or ARC/INFO® or define it in WMS after importing what is available. For example, if you only have a layer defining basin boundaries then you will need to construct a network using feature objects in WMS which properly "connects" the basin polygons together. This stream network may or may not actually represent the conveyance channel in the actual watershed, but must be present at least to the point that connectivity between sub-basins is defined.

Another problem that may occur is the ordering of vertices/nodes in a stream network may not be consistent with what WMS expects. WMS expects that the first or "from" node in an arc be the downstream node, while the second or "to" node is the most upstream node. If your stream vector data is not defined in this fashion it will have to be reordered prior to generating a correct topological watershed representation. The Reorder Streams command can be used to accomplish this.

A special extension for ArcView® has been developed by EMRL to allow some of these editing procedures to be taken care of on the GIS side. This extension also allows you to easily "package" your data in a superfile and then start WMS and pass the necessary data directly. See the WMS home page for more information about downloading the ArcView® extension with accompanying documentation.

Grid (DEM) Files

All ARC/INFO® or ArcView® grid data imported to WMS must be in the ASCII grid format. Grid files can be used as DEMs in WMS. Also, flow direction and flow accumulation grids can be used to define their respective attributes for DEM points. Once you have imported the elevation and flow direction ASCII grids, all of the remaining watershed parameters can be developed directly within WMS. The elevation DEM may also be used as the background elevation map used when creating a TIN.

TIN Files

ARC/INFO® TIN files must be in ASCII-NET format in order to import them and use for watershed characterization. The TIN can be used directly to create streams and basin boundaries, or as a background elevation map used in conjunction with feature objects for TIN creation.

Related Topics

- Importing Shapefiles
- Importing DEMs

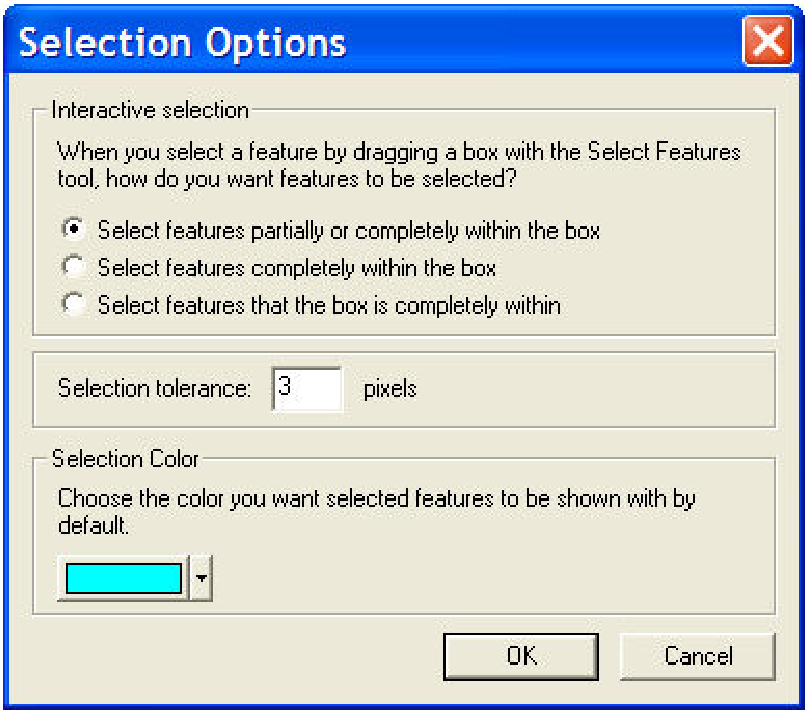
Feature Selection

With ArcObjects enabled three different selection methods are available. However, before attempting to select layers you should make sure that the layer(s) you want are selectable. By default a layer is selectable when you open it, but the status can be changed to unselectable using the **Selectable Layers** command in the **Selection** menu.

Graphical Selection

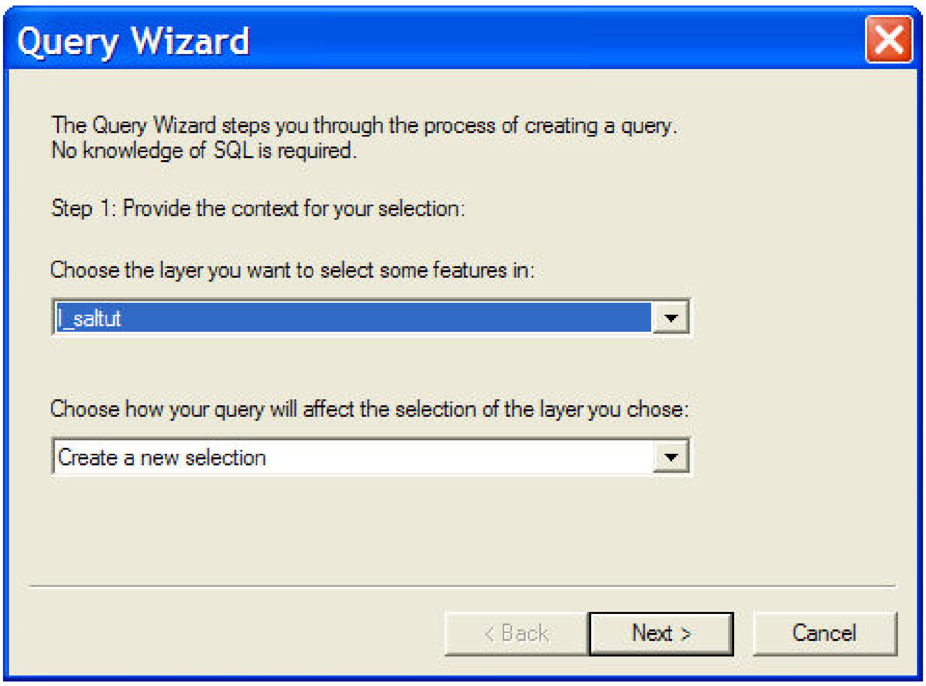
The select features tool can be used to interactively select features from GIS layers. When selecting using graphical selection all features from all selectable layers will be added to the selection list. A rectangular box can be dragged around a region of interest and then all features from all selectable layers that have in part within the rectangle will be added to the selection list.

The **Options** command is used to specify how features are selected and the way they are displayed.



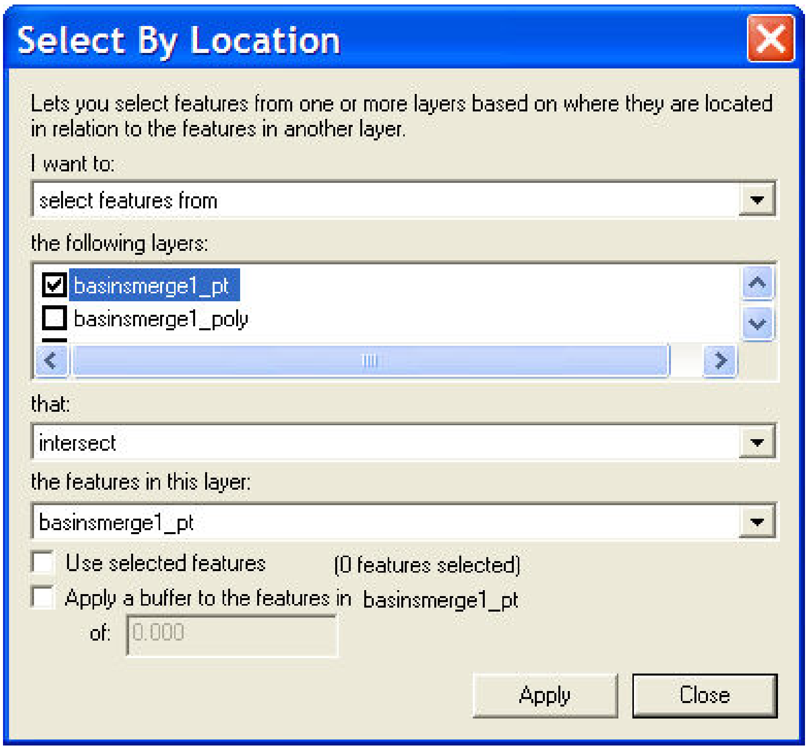
Selection by Attribute

Features of a layer can also be selected based on a query of the attributes and their values. The *Query Wizard* is used to choose the layer you wish to select from and the way the query results will be treated. The query can create a new selection, add to the current selection, remove from the current selection, or select on from within currently selected features.



Selection by Location

You can select features from one or more GIS layers based on their spatial relationship to another layer. The *Select By Location* dialog allows you to specify whether you will select new features, add to currently selected features, select from selected features, or delete from selected features. In addition, it allows you to specify the layers you are selecting from as well as the relationship with the selection layer. Finally, you specify the layer used for selection and whether you will use all features or only selected features. A buffer distance can be added to the features of the selection layer.



Clearing Selected Features

The **Clear Selected Features** command clears all features from the selection list.

Related Topics

- GIS Tool Palette
- Mapping GIS Features to Feature Objects

Importing Shapefiles

ARC/INFO® or ArcView® shape files provide the easiest method to import GIS data into WMS. Unfortunately the shape file format is extremely redundant, meaning that points or lines that are shared by lines or polygons are multiply defined.

Therefore, in order for you to convert a shape file to a WMS coverage it may take up to several minutes (depending on size) to build the correct line or polygon topology. This was very problematic in previous versions because WMS often bogged down when reading moderately large files. This is one of the primary reasons that the GIS module was developed and with or without a license to ArcObjects shapefile data can now be managed better by WMS.

With the addition of the GIS module there are now two different ways to import shapefile data.

Direct Conversion of Shapefile Data to Coverages

The first is the traditional method which allows you to load a shapefile layer (or a point, line, and polygon shapefile layer in the case of the drainage coverage) directly into a coverage, using the File | Open command.

You can then map attribute fields of the shapefiles database (.dbf) file to their pertinent WMS parameters (some key words are automatically recognized as listed in the table below.

Using the GIS Module to Convert Shapefile Data to Coverages

When opening a shapefile in the GIS module using the Add Shapefile Data or Add Data commands WMS first reads the points/lines/polygons into a simple display list and does not try to "build" topology by connecting arcs at nodes, and eliminating shared edges of polygons as required when creating a coverage. This makes the display and selection of the polygons much easier and more efficient. You can then select only the polygons you wish to convert to a coverage and map them. In this way you will only be building topology for the selected polygons.

Cleaning Imported Shapefile Data

If you intend to use the data from the shape file in more than one session you should save it as a WMS map file after importing/mapping the first time. Further, after importing the shape files you may wish to consider the following:

- Clean the feature objects in order to snap nodes within a certain distance, intersect arcs, and eliminate dangling arcs.
- Reorder Streams for arcs which will be used as stream arcs. WMS requires that the direction of an arc (from-node to to-node) be from “downstream” to “upstream.”
- Build Polygon so that WMS can define the topologic tree used for hydrologic modeling. After intersection of arcs, reordering of streams, etc. it is often necessary to rebuild the polygon topology so that the topologic structure is consistent with the tree used for hydrologic modeling.

Key Words for Automatic Mapping of Attributes

The following key words are used to automatically map shapefile (dbase or .dbf) attribute field names to data within WMS.

Keyword names to map Shape file attribute item names to WMS variables:

Parameter	Name	Description/Possible Values
Point Attributes		
Drainage Point type*	draintype	0=generic 1=link break 2=weir 3=bridge 4=culvert 5=outlet point
Temrminus Combine name	comname	HEC-1 terminus name
Terminus Route name	rtename	HEC-1 route name
LA County reach type	la_type	
LA County reach location	la_loc	
LA County reach lateral	la_lateral	
LA County reach slope	la_slope	
LA County reach length	la_length	
LA County reach n	la_n	
LA County reach side n	la_siden	
LA County reach slope	la_sideslp	
LA County reach depth	la_depth	
LA County reach channel def	la_chdef	
LA County reach char width	la_charw	
LA County reach velocity	la_vel	
LA County reach detail	la_detail	
LA County reach hydrograph	la_hydrog	
LA County reach input hydrograph	la_inhydro	
LA County reach input hydrograph area	la_inharea	
LA County reach relief	la_relief	
LA County reach trap	la_trap	
LA County reachsize	la_size	
LA County reach capacity	la_capc	
LA County reach bulked flow	la_bulkedf	
LA County reach reservoir routing	la_resrt	
LA County reach initial pool elev	la_initp	
LA County reach street width	la_streetw	
LA County reach reach curb height	la_curbh	

GSSHA Point type	casctype	GSSHA point type: 0 = generic 1 = link break 2 = hydraulic structure (weir) 3 = bridge 4 = culvert 11 = dynamic well 12 = static well 13 = constant head 14 = river head 15 = rating curve 16 = rule curve 17 = scheduled discharge 24 = GSSHA low point 25 = GSSHA oulet
Link id	linkid	GSSHA
Free flow coefficient	ffcoeff	GSSHA
Crest width	crestwid	GSSHA
Crest elevation	crestelev	GSSHA
Flooded coefficient	floodcoef	GSSHA
Gage depth	gagedepth	Rain gage depth for gage coverage
Z coordinate	elevation	This changes the Z-coordinate of the point
Arc Attributes		
Drainage arc type*	draintype	0 = generic 3 = general stream 4 = highway
GSSHA arc type	casctype	GSSHA arc type: 0 = generic 1 = trapezoidal stream 2 = ridge 3 = general stream 8 = breakpoint stream 10 = Priess. trapezoidal stream arc 11 = Priess. breakpoint stream arc 12 = subsurface trapezoidal stream arc 13 = subsurface breakpoint stream arc 30 = erodable arc 31 = subsurface erodable arc 42 = GSSHA embankment arc 43 = wetland arc
Link ID	linkid	GSSHA Link ID
Manning's	cmannings	GSSHA Manning's "n" value
Bottom width	bwidth	GSSHA bottom width
Channel depth	chdepth	GSSHA channel depth
Side slope	sideslope	GSSHA side slope
M River	mriver	GSSHA M-River value
K River	kriver	GSSHA K-River value
Arc Elevation	elevation	Arc point elevation
Flood barrier elevation	fbelev	Flood barrier elevation

Polygon Attributes		
Drainage polygon type*	draintype	Drainage polygon type: 0 = generic 1 = boundary 2 = lake
Drainage basin id	basinid	Drainage basin id (integer)
sub-basin area	basinarea	Basin area (float)
sub-basin slope	basinslop	Average slope within the sub-basin (float)
sub-basin maximum flow distance	mfdist	Max flow path, including overland and stream flow (float)
sub-basin max flow distance slope	mfdslope	Slope along the max flow path as defined above (float)
sub-basin distance to centroid	centdist	Distance from centroid to closest point on main channel (float)
sub-basin stream centroid to outlet	centout	Distance from point in stream closest to centroid to outlet (float)
sub-basin slope from centroid to outlet	slcentout	Slope along the distance defined above (float)
sub-basin percent southfacing	psouth	Percentage of area facing south, 0.0-1.0 (float)
sub-basin percent northfacing	pnorth	Percent of area facing north, 0.0-1.0 (float)
sub-basin maximum stream length	mstdist	The longest stream distance within the basin (float)
sub-basin maximum stream slope	mstslope	The slope along the longest stream distance (float)
sub-basin length	basinlen	Distance to furthest point along basin perimeter (float)
sub-basin shape factor	shapefact	Basin length divided by basin area (float)
sub-basin sinuosity factor	sinuosity	Maximum stream length divided by basin length (float)
sub-basin perimeter	perimeter	Perimeter of basin (float)
sub-basin average elevation	meanelev	Average elevation (float)
sub-basin centroid	centroidx	Basin centroid, closest point in basin if centroid is outside of the basin (X-coord) (float)
sub-basin centroid	centroidy	Basin centroid, closest point in basin if centroid is outside of the basin (Y-coord) (float)
sub-basin name	basinname	Basin name (string)
sub-basin lagtime	lagtime	Lag time, in Hours (float)
sub-basin time of concentration	tc	Time of Concentration, in hours (float)
sub-basin SCS curve number	cn	SCS Curve number computed from hydrologic soil type and land use (Integer)
sub-basin initial abstraction	ia	Initial abstraction used for the HEC-1 Green and Ampt method
sub-basin volumetric moisture deficit	dtheta	Volumetric moisture deficit used for the HEC-1 Green and Ampt method
sub-basin wetting front suction	psif	Wetting front suction used for the HEC-1 Green and Ampt method
sub-basin hydraulic conductivity	xksat	Hydraulic conductivity used for the HEC-1 Green and Ampt method
sub-basin percent impervious	rtimp	Percent impervious used for the HEC-1 Green and Ampt method
sub-basin Maricopa County adjusted slope	adjslope	Maricopa County adjusted slope for computing Clark Tc and R values
sub-basin Clark R value	clarkr	Clark R value for the HEC-1 Clark unit hydrograph method
sub-basin average precipitation	precip	Basin average precipitation, in inches (float)
sub-basin LA County lateral	la_lateral	LA County lateral
sub-basin LA County location	la_loc	LA County location
sub-basin LA County rainfall depth	la_raind	LA County rainfall depth
sub-basin LA County Tc	la_tc	LA County Tc

sub-basin LA County percent impervious	la_imprv	LA County percent impervious
sub-basin LA County soil number	la_soil	LA County soil number
sub-basin LA County basin hydrograph	la_hydrog	LA County basin hydrograph
sub-basin LA County basin bulked flow	la_bulkedf	LA County basin bulked flow
landuse	lu_code	Land use code from the SCS land use table. Possible values range from 0-127
Land use join ID	mu_code	Land use join ID
Percent impervious	imperv_	LA County percent impervious
soil type	hydgrp	SCS Soil type, A, B, C, or D or 0, 1, 2, or 3
soil type (LA County)	class	LA County soil type
sub-basin rainfall depth	rainfall	Rainfall depth
runoff coefficient, C	runoffc	Rational method runoff coefficient, C (float)
GSSHA polygon type	casctype	GSSHA polygon type: 0 = Generic 1 = Boundary 2 = Lake 3 = No Boundary 4 = Wetland
GSSHA leakage discharge	ldis	GSSHA leakage discharge
GSSHA spillway width	spwidth	GSSHA spillway width
GSSHA discharge coefficient	discoeff	GSSHA discharge coefficient
GSSHA water elevation	welev	GSSHA water elevation
GSSHA spillway crest	spillcres	GSSHA spillway crest
Rainfall zone ID	rnzone_	Rainfall zone ID
Rainfall zone name	name	Rainfall zone name
DPA zone	dpa_zones	DPA zone
Flood area	fexarea	Flood area
Flood depth	fexdepth	Flood depth
Flood class ID	fexcid	Flood class ID
Flood class name	fexcname	Flood class name
NFF state	state	NFF state (2-letter state abbreviation, e.g. UT = Utah)
NFF state's region	nff_region	NFF region in state

* means this is essential to import into WMS and create a watershed model directly. The three essential items are point, arc, and polygon types. The general stream arc should be used to represent a stream in a watershed model. The boundary polygon type should be used to represent a polygon boundary. The outlet point type should be used to represent a watershed outlet or sub-basin outlet point.

In order to import shapefile attributes into WMS and build a tree automatically, the following conditions must be met:

1. A point coverage containing watershed and sub-basin outlets with the appropriate type (outlet point) attribute defined must exist.
2. An arc, or line, coverage containing streams in the watershed with the appropriate type (general stream) attribute defined must exist.
3. A polygon coverage containing watershed boundaries must exist.

4. There cannot be any overlapping arcs.
5. Stream arcs must be created from the downstream to the upstream node for all stream arcs.

If a data value in the shapefile corresponds to a WMS variable but it is not defined with the appropriate keyword it can be mapped manually using the Attribute Mapping dialog. One item from the database fields window is selected and the corresponding coverage attribute field is also identified. Finally, the Map button is selected to define a new mapped attribute. The Unmap button can be used to remove a pair of mapped fields.

Related Topics:

- Supported WMS File Formats
- Feature Object Guidelines
- GIS Module
- Mapping GIS Layer Data to Feature Objects

Add Shapefile Data

The **Add Shapefile Data** command allows you to browse for and open shapefiles as GIS layers in WMS. Without a license of ArcView® on your computer, shapefiles are the only supported format for GIS layers. With a valid license of ArcView®, the **Add Data** command is activated and any of the ESRI supported formats can be opened as GIS Data Layers.

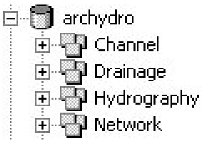
Shapefiles can be imported directly to coverages (by passing the GIS data layer) in the traditional way by opening them from the **File** menu.

Related Topics

- Add Data
- Deleting GIS Layers
- Enabling ArcObjects
- Importing Shapefiles
- Mapping GIS Features to Feature Objects

Feature Objects to ArcHydro Geodatabase

The **Feature Objects → ArcHydro Geodatabase** command is similar to the Feature Objects → Geodatabase command, except that the geodatabase is written out in a particular format. The ArcHydro geodatabase model was developed to create a common GIS framework for storing water resources data. ArcHydro accounts for stream networks and their associated topology, along with the drainage, channel, and hydrography data defining your project. For more information, see Maidment (2002). These data are stored in four different feature datasets within the geodatabase.



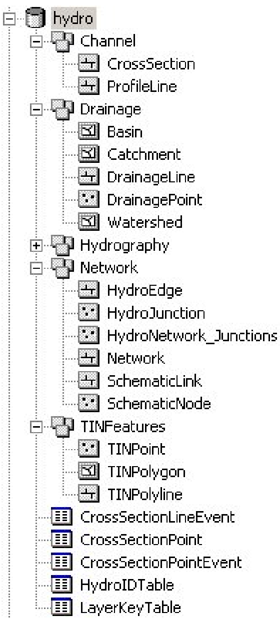
In order to create an ArcHydro geodatabase from WMS, you must first switch to the GIS module and enable ArcObjects. WMS uses the feature objects stored in each coverage to build the geodatabase.

Set up and build your feature objects you wish to add to the geodatabase, and define a coordinate system for your project. Vector data are stored in feature datasets and feature classes in the geodatabase, which require a common spatial reference. The spatial reference in the geodatabase is defined from the current coordinate system in WMS.

A WMS drainage coverage is used to build the Network, Drainage, and part of the Hydrography feature datasets. 1D Hydraulic Centerline and 1D Hydraulic Cross Section coverages are used to define the Channel feature dataset. If you have a DEM, it canl be used to create a raster grid. Each TIN in memory can also be used to create a feature dataset suitable for building a TIN within a GIS.

Once your data and spatial reference are defined, select **Feature Objects → ArcHydro Geodatabase** from the **Mapping** menu in the GIS module. At this point, you will be asked if you wish to create a raster grid from your DEM in WMS, if you have one loaded. You will then be asked whether you wish to create a feature dataset consisting of a TIN boundary, TIN breaklines, and TIN vertices if you have a TIN in memory. WMS will build the ArcHydro geodatabase based on the coverages and data you created. WMS will then create the raster grid and TIN dataset if you have this data in your WMS project and elected to build them.

Once WMS has finished building the geodatabase, you can view the results in ArcCatalog or ArcMap. WMS will have created a new geodatabase with for feature datasets according to the ArcHydro data model, along with an additional feature dataset for your TIN (if applicable). WMS will also create the necessary tables for the model.



Related Topics

- Enabling ArcObjects
- Coordinate Systems
- Feature Objects to Geodatabase

Feature Objects to Geodatabase

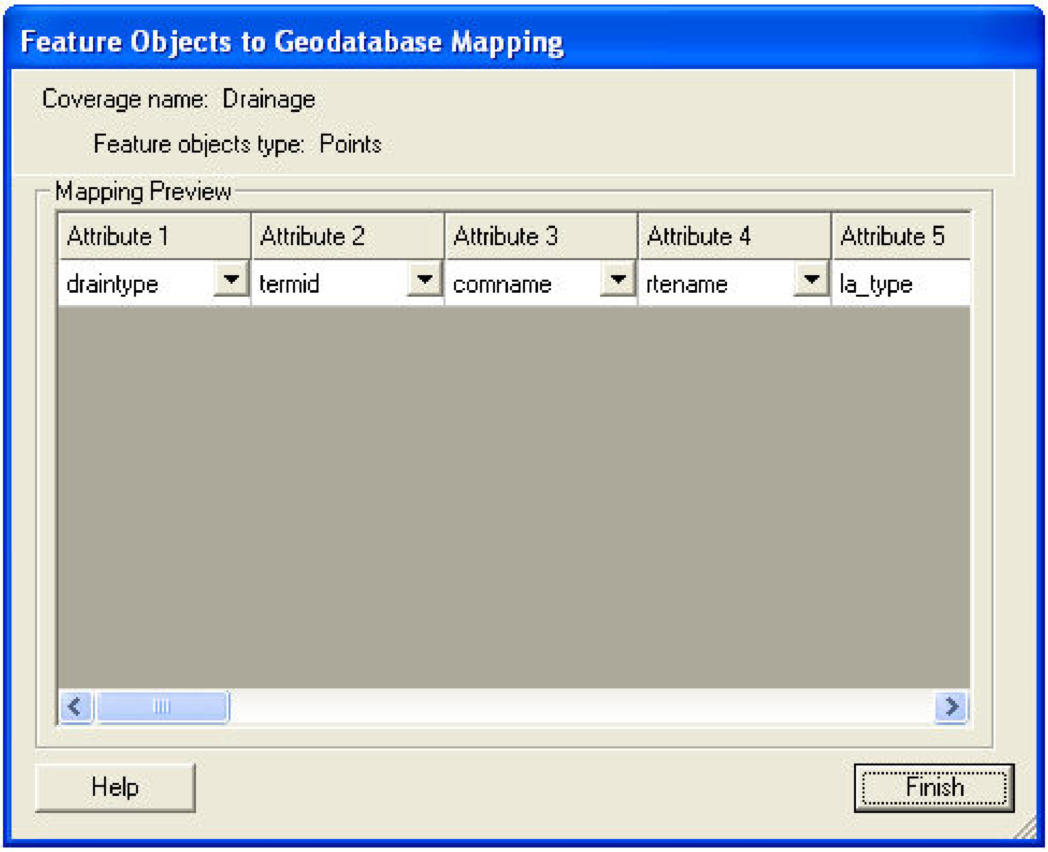
A geodatabase is a geographic database used to store GIS information, such as features and rasters. In addition to storing data, a geodatabase allows you to set up relationships between data and create rules to validate data. Geodatabases are used in ArcGIS products.

In order to create a geodatabase from WMS, you must first switch to the GIS module and enable ArcObjects. WMS uses the feature objects stored in each coverage to build the geodatabase.

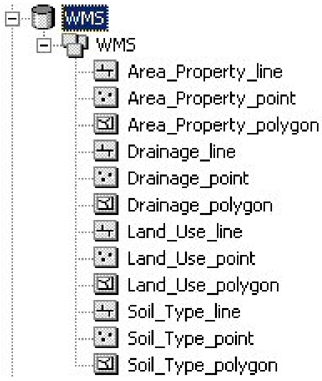
Set up and build your feature objects you wish to add to the geodatabase and define a coordinate system for your project. Vector data are stored in feature datasets and feature classes in the geodatabase which require a common spatial reference. The spatial reference in the geodatabase is defined from the current coordinate system in WMS.

Once your data and spatial reference are defined, select **Feature Objects → Geodatabase** from the **Mapping** menu in the GIS module. At this point, you will be prompted whether you wish to specify which coverage attributes will be mapped to the geodatabase. You have the option to specify which WMS coverage attributes to map over, or to let WMS map all of the appropriate attributes for you.

If you choose to map the WMS coverage attributes yourself, you will be prompted for each feature class created in the new geodatabase, which can take a while. If you choose to let WMS map the attributes, you will not be prompted and WMS will build the geodatabase automatically.



Once WMS has finished building the geodatabase, you can view the results in ArcCatalog or ArcMap. WMS will have created a new geodatabase with a single feature dataset, with feature classes for the points, lines, and polygons of each WMS coverage.



Related Topics

- Enabling ArcObjects
- Coordinate Systems
- Feature Objects to ArcHydro Geodatabase

5.9. 2D Grid Module

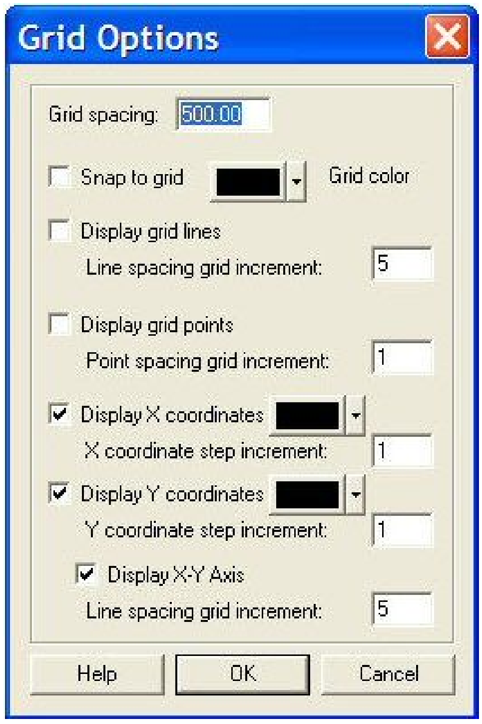
2D Grid Module

The 2D Grid module is used for surface visualization and for the development of a GSSHA rainfall/runoff analytical model. For example, the user can discretize a watershed into a number of grid cells and then define important rainfall, infiltration, and channel properties at grid cells in preparation for running GSSHA. Any parameter such as hydraulic conductivity or rainfall intensity may be interpolated from a set of scattered data points to the grid. Results of the 2D analysis can then be contoured on the grid to display the variation in the computed results.

Related Topics

- Creating Grids
- Grid Display Options

Grid Options



When entering new vertices or entering a polygon or polyline, it is often useful to have the coordinates snap to a uniform grid. This allows accurate placement of the objects when the desired coordinates are even multiples of some number.

A drawing grid can be activated using the **Grid Options** command in the *Display* menu.

Grid Spacing

The *Grid Spacing* edit field specifies the spacing of the grid nodes and grid lines in the drawing grid. The Grid color window specifies the color that is used to display the drawing grid in the *Graphics Window*.

Snap to Grid

If the *Snap to grid* option is selected, all new vertices, nodes, points, etc., snap to the closest grid point as they are being created or when they are dragged interactively.

Display Grid Lines

If the *Display grid lines* option is selected, grid lines are displayed according to the Grid line spacing increment. For example, if the Grid spacing is set to 10 and the Grid line spacing increment is set to 5, a grid line will be drawn every 50 units.

Display Grid Points

If the *Display grid points* option is selected, grid points are displayed according to the Grid *point spacing increment*.

Display X/Y Coordinates

If the Display X Coordinates or Display Y coordinates are selected then text with the coordinate values are displayed along the bottom (X coordinates) and left (Y coordinates) of the screen.

Display X-Y Axis If the Display X-Y Axis is selected then an X-Y coordinate axis is displayed along the left and bottom of the screen. In combination with the Display X/Y Coordinates this is an excellent way to indicate the scale of the model.

Related Topics:

Display Menu

2D Grid Module Project Explorer

The *Project Explorer* serves as a means of organizing all the datasets associated with the several models and modules in WMS. In the 2D Grid module, the *Project Explorer* serves to store and modify the attributes of all gridded datasets.

Datasets

The *Project Explorer* serves as a manager for all of the gridded datasets that have been read into WMS. When other modules are active the Project Explorer references TINs, DEMs, Scatterpoint sets, Coverages, and other data storage types. The main strength of the *Project Explorer* lies in consolidating access to the datasets. Right-clicking on a data set brings up a list of options that are available for that data type. In the 2D Grid module, when a data set is right-clicked, a menu appears that offers the following options: exporting the dataset, renaming the dataset, viewing the properties of the dataset, setting the contour options for the dataset, viewing the values of the dataset, setting the dataset to be the grid elevations, and writing to an ASCII grid file.

Index Maps

Unique to GSSHA are special datasets that serve as index maps that link simulation parameters to their spatial distribution. The principle means of modifying and creating index maps is in the Index Map dialog but the index maps are now also able to be accessed through the Project Explorer. When an index map has been created or read into WMS, a folder appears in the *Project Explorer*, named *Index Maps*, that contains all of the index maps for the simulation. Index maps can then be treated like regular datasets; they can be contoured, renamed, deleted, and edited.

Solutions

While solutions are not new to GSSHA, being able to work with them in WMS is a new feature. Accessible from the *GSSHA* menu in the 2D Grid module is the command Read Solution which looks for a GSSHA project file and then reads in all of the associated datasets and lumps them together into a solution folder in the *Project Explorer*. Solution folders are identified by a lowercase "s" on the folder. All of the datasets in the folder are treated as regular datasets. Organizing the datasets into a solution allows several solutions to be in memory at the same time. Several dialogs look for solutions and the associated datasets to set up and display output graphs. Along with the regular datasets, the summary file for the project is also accessible for each solution by double-clicking the summary file *Project Explorer* item under the solution folder.

Individual solution output time series data for a cell may be viewed from the Solution Results dialog accessed from the *Feature Point/Node Type* dialog. This dialog will only show the output data set time series for the cell that underlies the feature point selected. To compare the solution output at a cell with observed data, see GSSHA Observation Points.

Contour Options

New to WMS 7.0 is the use of OpenGL code for much better graphics display. OpenGL is a 3D graphics library that makes displaying graphics in WMS much faster and more efficient. The grid display routines in WMS have been enhanced with a better interpolation scheme that allows for a much smoother grid display as well as taking advantage of the strengths of OpenGL. The contouring routines have been redone to also use the enhanced grid and OpenGL, which results in faster, smoother surfaces that are continuous color contoured as opposed to step color contoured as in WMS 6.1. Also, OpenGL shades surfaces automatically, so this option is always turned on when the contour is color filled.

Cell filling the grid has been an option since WMS 6.1. However, WMS 7.0 allows much more flexibility in visualizing the output and input to GSSHA through cell filling by making it into part of the contour options. Simply ensure that the cell filled radio button is selected, and each cell will be filled with the color that represents the data point at the center of the cell. This color is taken from a smooth (linearly interpolated) color ramp. Cell filling is independent of block-style cell display, which is set in the display options.

The *Project Explorer* allows access to individual contour options for each dataset. When a dataset is read in it is given the default set of contour options that can then be modified by right clicking on the dataset in the Project Explorer. Whichever dataset is active is the only dataset contoured. The contours for the grid come from the active dataset whereas the elevations of the grid come from the current elevation dataset. This setup allows the results of a GSSHA simulation to be displayed as contours on a grid that is shaped like the underlying terrain. In the *Project Explorer* the active dataset is shown with a yellow icon and bold lettering and the current elevation dataset has "elev" after the name of the dataset.

Related Topics

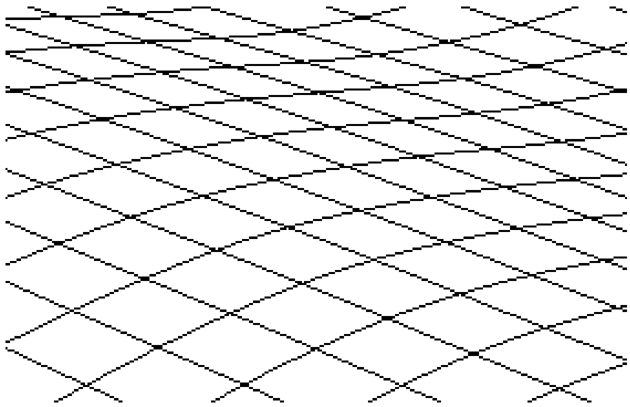
- GSSHA Maps

2D Grid Display Options

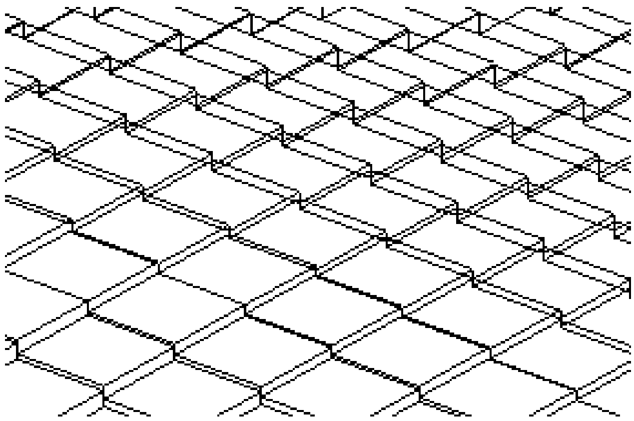
The display options control which components of the grid are displayed. The display options can be set by selecting the **Display Options** command in the *Display* menu. Most of the items in the dialog box are toggle boxes. If the toggle for a component of the grid is set, the component is displayed when the grid is redrawn. The color used to display the component can be set using the pop-up color window to the left of the toggle box.

The grid display options are as follows:

- The **Nodes** item is used to display grid nodes. Since all grids in WMS 7.0 are cell-centered, a dot is displayed at the cell centers.
- The **Cells** item is used to display the edges of active grid cells. The cells are drawn using the specified grid cell color and at the height set by the current elevation dataset.
  - The **cell style** options dictate the form of the cells, whether smooth or block-style. The cells are both drawn and contoured in this style.



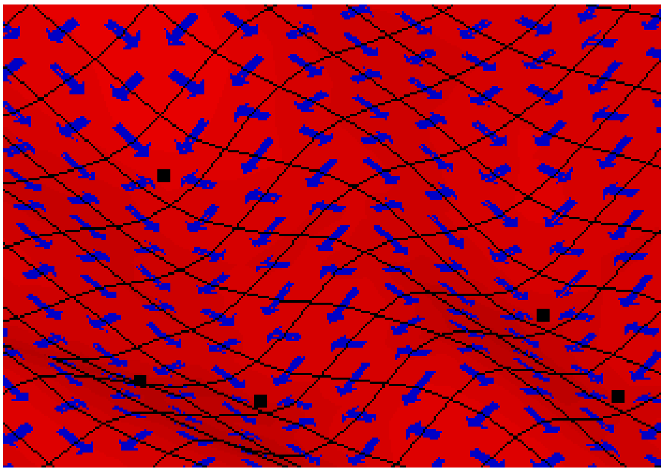
Smoothed cells use interpolated values to match the cell corners.



Blocked cells represent the cells as planar surfaces

- The **Grid Boundary** item is used to display a solid line around the perimeter of the grid. Displaying the boundary is useful when contours are being displayed with the cell edges turned off.
- The **IJ Indices** item is used to display the ij indices of each cell or node.
- If the **Inactive Cells** item is used to display cells which are inactive. If this option is turned off, inactive cells are not displayed. Inactive cells must be displayed before they can be selected.
- The **Elevations** item is used to display the z coordinate of each node or cell; the scalar item displays the active data set value for each cell.
- The **Contours** item is used to display contours computed using the active scalar dataset.

  - The **contours options** button accesses the contour options of the active dataset. Each gridded dataset has its own contours set.
- The **Lake Cells and Wetland Cells** items are used to fill the cells whose centroids are contained within a lake or wetland polygon with the chosen color.
- The **Flow Vectors** item is used to display a set of arrows pointing from cell to cell that show which way water would flow from the cells based solely on elevation. An equals sign is displayed between the cells if the cells have the same elevation.
- The **Digital Dams** item is used to mark those cells that are lower in elevation than the four lateral neighboring cells.



Represented here are flow vectors (blue arrows) that show the direction of flow from a cell to its neighboring cells as well as the cells which have been identified as digital dams (black square markers).

Related Topics

- Active/Inactive Cells
- Digital Dams
- 2D Grid Module Project Explorer

2D Grid Tools

The following tools are contained in the dynamic portion of the Tool Palette when the 2D Grid module is active.

Select Cell

The **Select Cell** tool is used to select individual grid cells or grid nodes. Multi-selection can be performed by holding down the *SHIFT* key while selecting or by dragging a rectangle to enclose the cells to be selected. The ij indices of the selected cell are displayed in the *Edit Window*.

If the grid is cell-centered, selected cells are highlighted by drawing small circles around the cell centers. If the grid is mesh-centered, the grid nodes (corners) are selected, and the circles are drawn around the grid nodes.

Only visible cells can be selected. Cells which have been hidden cannot be selected. Inactive cells can only be selected when they are being displayed by turning on the *Inactive Cells* item in the *Display Options* dialog (see section below on active/inactive cells).

Select i

The **Select i** tool is used to select an entire "row" (set of cells with the same i index) of cells at once. Multi-selection can be performed by holding down the SHIFT key. The i index of the selected row is displayed in the Edit Window.

Select j

The **Select j** tool is used to select an entire "column" (set of cells with the same j index) of cells at once. Multi-selection can be performed by holding down the SHIFT key. The j index of the selected column is displayed in the *Edit Window*.

Contour Labels

The **Contour Label** tool manually places numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contour options are changed, until they are deleted using the Contour Labels dialog, or until the grid is edited in any way. Contour labels can be deleted with this tool by holding down the SHIFT key while clicking on the labels. This tool can only be used in plan view.

The Create Gages Tool

The **Create Gages** tool is used to interactively create gages in the *Graphics Window*. When this tool is active, a new gage is created by clicking in the *Graphics Window* at the desired location of the gage (the Graphics Window must be in plan view when creating gages). The xy coordinates of the gage are defined by the cursor position and you are prompted for the z coordinate. The x, y, and z coordinates of a new gage can be edited using the *dit Window*. In addition, once a gage has been defined with the **Create Gages** tool, the gage can be edited using the Gages dialog.

The Select Gages Tool

The **Select Gages** tool is used to select previously defined gages. A set of selected gages can be deleted by hitting the *DELETE* key or by selecting the **Delete** command from the Edit menu. The coordinates of a selected gage can be edited using the Edit Window. The location of a gage can also be edited by holding down the mouse button when a gage is selected and dragging the gage. This tool is also used to control what is plotted in the *Gage Plot Window*. Only the curves associated with selected gages are plotted.

 The Select Hydrograph Tool

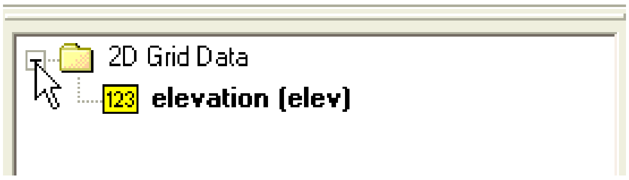
The **Select Hydrograph** Tool allows you to select hydrograph icons that appear at specified locations after completing a GSSHA solution. By double-clicking on the icon a hydrograph plot window will be created (or you can select the hydrograph and choose the New Hydrograph Plot option from the Display menu).

Related Topics

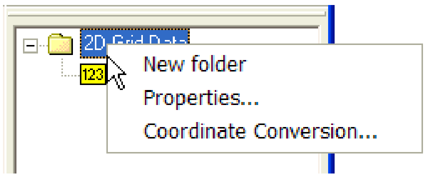
- Terrain Data Tool Palette
- Map Tool Palette
- Hydrologic Modeling Tool Palette
- Scatter Point Tool Palette

Tree Contents for 2D Grid Module

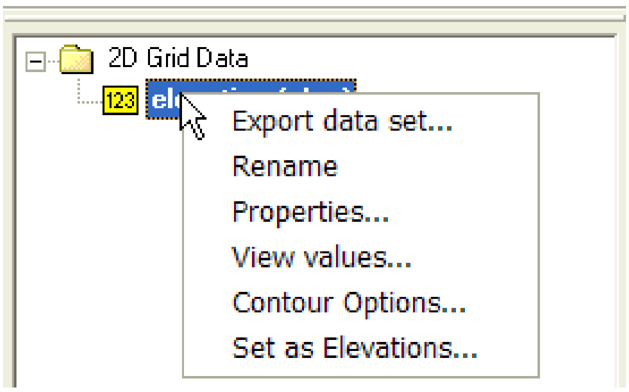
In the 2D Grid module, the grid and associated data sets are listed in the *Project Explorer* (WMS only allows you to work with one grid).



Right clicking on the main 2D Grid Data folder allows you to create a **New folder**, or **Convert Coordinates**.



Right clicking on the dataset for a grid allows you to **Export**, **Rename**, view **Properties**, **View Values**, set **Contour Options**, and **Set as Elevations**.



Related Topics

- Project Explorer Overview
- Coordinate Conversions

Active/Inactive Cells

Each of the cells in a cell-centered grid can be active or inactive. An inactive cell is a cell which is not part of the computational domain. For example when doing a surface runoff analysis using GSSHA cells inside the watershed boundary should be active while cells outside the boundary should be inactive.

An inactive cell is ignored when contours are displayed on the grid. A set of selected cells can be made inactive by selecting the **Inactivate Selected** command in the *Grids* menu. A set of inactive cells can be made active again by turning on the display of inactive cells using the *Display Options* dialog, selecting the cells, and selecting the **Activate Selected** command in the *Grids* menu (inactive cells can only be selected if they are being displayed).

Activate Polygon Region In many cases it is useful to delineate the active/inactive regions in a grid using a polygon. A region can be activated by selecting the **Activate Polygon Region** from the *Grids* menu. A dialog appears prompting you to select either *Read polygon from file* or *Select polygon interactively*. If the *Read polygon from file* option is selected, WMS brings up the *File Browser* dialog, and prompts the user to specify a polygon file. If the *Select polygon interactively* option is selected, WMS prompts the user to define the polygon by selecting at least three points and double-clicking when done.

Once a polygon is entered, each cell is compared to the polygon. If the cell center is on the interior of the polygon, it is made active. Otherwise, the cell is made inactive.

Related Topics

- Creating Grids
- 2D Grid Display Options

Data Type Conversion

It is sometime useful to convert a 2D grid to a set of scattered data points or a TIN data structure. Data for the grid can then be used to perform operations available for scattered points or TINs.

Grid → Scatter Points

The **Grid → Scatter Points** command in the *Grids* menu is used to create a new scatter point set using the nodes or cells of a 2D grid. A copy is made of each of the datasets associated with the grid and the datasets are associated with the new scatter point set.

This command is useful for comparing the solutions from two separate simulations from different grids. For example, if two simulations have been performed with slightly different grids (base vs. plan) it may be useful to generate a contour or fringe plot showing the difference between the solutions. It is possible to generate a dataset representing the difference between two datasets using the data calculator. However, the two datasets must be associated with the same grid before the data calculator can be used. The datasets from one of the grids can be transferred to the other grid as follows:

1. Load the first grid and its dataset into memory.
2. Convert the grid to a scatter point set using the **Grid → Scatter Points** command.
3. Delete the first grid by selecting the **Delete All** command from the *Edit* menu.
4. Load the second grid and its dataset into memory.
5. Switch to the 2D Scatter Point module and select an interpolation scheme using the **Interpolation Options** command in the *Interpolation* menu.

6. Interpolate the dataset to the second grid by selecting the **Interpolate to Grid** command from the *Interpolation* menu.

At this point, both datasets will be associated with the second grid and the data calculator can be used to compute the difference between the two datasets.

Grid → TIN

A new TIN can be created from a 2D grid by selecting the **Grid → TIN** command from the *Grids* menu. Two triangles are created from each cell in the grid. The active scalar dataset becomes the z value of TIN vertices.

Related Topics

- 2D Scatter Point Module

5.10. 2D Scatter Point Module

2D Scatter Point Module

The Scatter Point module is used to interpolate from groups of scattered data points to grids or TINs. The Scatter Point module can be used to interpolate from a set of scattered xy points representing something like rain gages to a finite difference grid or to basin centroids for establishing rainfall curves for HEC-1. A variety of interpolation schemes are supported.

Scattered data can be created using the create point tool. The most common use of scattered data are for creating water surface elevations to compute a floodplain delineation. Generally in such cases you will want to take advantage of a centerline and cross sections to develop a larger dataset for flood plain delineation.

Scattered Data Tools

The following tools are active in the dynamic portion of the Tool Palette whenever the 2D Scatter Point module is active.

 Select Scatter Point

The **Select Scatter Point** tool is used to select individual scatter points for displaying the coordinates and current function value of individual scatter points in the *Edit Window*.

 Scatter Data Point

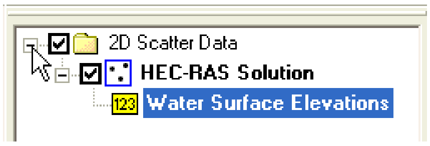
The **Create Scatter Data Point** tool is used to create new scatter points. The primary purpose of creating scatter points is to enter water surface elevations that can be used in a flood plain delineation. It can only be used in plan view.

Related Topics

- Interpolation
- Preparing Stages for Floodplain Delineation
- Interpolating Results Along a Centerline and Cross-sections
- Terrain Data Module

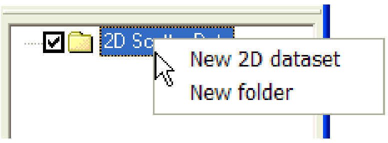
Project Explorer Contents for 2D Scatter Module

In the 2D Scatter module scattered datasets and their associated datasets are displayed in the *Project Explorer*. The toggle box to the left of the scattered dataset controls the visibility.

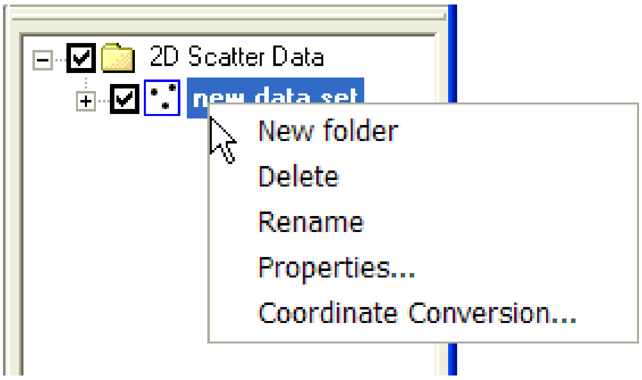


The active data is determined by selecting it from the Project Explorer.

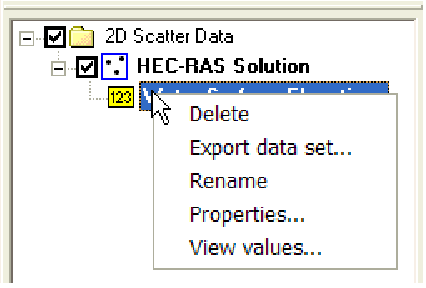
Right clicking on the main 2D Scatter Data folder allows you to create a **New 2D dataset** or a **New** folder to organize scatter datasets in.



Right clicking on a scattered dataset allows you to allows to create a **New folder**, **Delete**, **Rename**, view **Properties**, and **Convert Coordinate Systems** of the scattered dataset.



Right clicking on one of the datasets allows you to you to **Delete**, **Export**, **Rename**, view **Properties**, or **View Values** of the dataset.



Related Topics

- Project Explorer Overview
- Coordinate Conversions

Scatter Point Sets

Each of the points from which values are interpolated are called scatter points. A group of scatter points is called a scatter point set. Each of the scatter points is defined by a set of xy coordinates.

Each scatter point set has a list of scalar datasets. Each dataset represents a set of values that can be interpolated to a grid or TIN. When an interpolation command is selected, the active dataset for the scatter point set is used in the interpolation process.

Multiple scatter point sets can exist at one time in memory. One of the scatter point sets is always designated as the "active" scatter point set. Interpolation is performed from the active scatter point set only. The active scatter point set can be changed using the *Project Explorer*. Whenever a new scatter point set is read from a file or created, it becomes the active set.

The most common uses of scatter point sets in WMS is for rainfall gages and the results of hydraulic modeling.

Related Topics

- Terrain Data Module
- Preparing Stages for Floodplain Delineation

6. Models

Models

Hydraulic Models

The primary purpose of the hydraulic modeling interface within WMS is to process digital terrain and map data (TINs and coverages) to build the basic geometry necessary for a 1D Hydraulic Model. Much of the information for developing models with these tools is described in the information on River Tools in the Map module.

- Hydraulic Modeling

It is possible to establish the hydraulic model with extracting cross section information from a TIN. Cross sections which have already been surveyed can be used by assigning them to an arc. This, along with geo-referencing the data is done using the cross section editor from the **River Tools** menu in the Map module (when River Tools is the active model).

Hydrologic Models

Hydrologic analysis is typically done using lumped parameter models such as HEC-1. The Tree module provides a graphical interface to HEC-1, TR-20, HSPF, TR-55, Rational Method, the National Flood Frequency (NFF), and other programs. In the absence of terrain data, topological or tree representations of a watershed can be created. Then all necessary input data to run one of the supported models can be defined using a series of user-friendly dialogs.

- Hydrologic Modeling

Related Topics

- Hydraulic Modeling
- Hydrologic Modeling

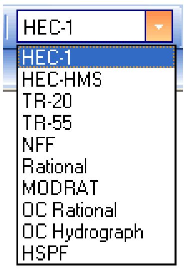
Models Available in WMS

- CE-QUAL-W2
- GSSHA
- HEC-1
- HEC-HMS
- HEC-RAS
- HSPF
- MODRAT
- NFF
- OC Hydrograph
- OC Rational
- Rational
- River Tools
- Storm Drain
- Storm Drain-FHWA
- SMPDBK

- SWMM
- TR-20
- TR-55

Model Selection

When a module supports several models such as the Map module or the Hydrologic modeling module then WMS will only display the menu (and associated commands) for the active model. The active model is set using the drop down combo box at the right side of the edit window.



How to select a model

Do I want to do hydrology or hydraulics?

1. Hydraulics:
   1. Do I want to do a simple dam break simulation?
      1. SMPDBK (Simplified Dam Break)
   2. Do I want to do floodplain delineation, hydraulics, or a more complicated dam break simulation?
      1. HEC-RAS
   3. Do I want to model storm drains?
      1. Do I need additional capability only available in xpswmm such as FHWA hydraulic, etc.?
         1. xpswmm
      2. Do I not need that additional capability?
         1. EPA-SWMM
2. Hydrology:
   1. Am I doing a small urban model?
      1. TR-55
      2. Rational
      3. MODRAT (LA County)
      4. OC Rational (Orange County, CA)
      5. NSS Urban (Statistical model)
      6. HEC-1 (OC Hydrograph)
      7. HEC-HMS
      8. SWMM
      9. TR-20
   2. Am I doing a rural model?
      1. NSS (Statistical model)
      2. HEC-1 (OC Hydrograph)

3. HEC-HMS
4. SWMM
5. TR-20
3. Am I interested in water quality, sediment transport, or the effects of wetlands on a hydrograph
    1. GSSHA
    2. HSPF (Water Quality only)

Louisiana DOTD Modeling Overview

Louisiana Department of Transportation and Development Data Sources for use in WMS

Data Sources

The primary gateway to accessing data for WMS can be found at: http://www.xmswiki.com/ [1] and then go to the GeoSpatial Data Acquisition Page http:/ / www. xmswiki. com/ xms/ GSDA:GSDA [2] Some of the primary locations there are:

DEMS: http://nationalmap.gov/viewer.html [3]

LandUse: http://www.webgis.com/ [4]

Soils: http://soildatamart.nrcs.usda.gov/ [5]

Aquaveo: http://www.aquaveo.com/ [3]

The LSU site can also be helpful, especially for DRG Images of USGS Maps. To download and use these maps you can:

1. Go the site: http://atlas.lsu.edu/ [6]
2. Choose **Download Data**
3. Choose the map scale such as http://atlas.lsu.edu/q24k/1:24,000 [7] **Scanned Topographic Maps**
4. Use the map to navigate to the area you want.
5. Select the map
6. Choose the **Download** button
7. Unzip the map on your computer
8. Open WMS
9. Right Click in the *Project Explorer* of WMS
10. Choose *Preferences*
11. Change the *Image* options to **Always Convert Tiff** and to Save the new JPG in the current directory.
12. Open the image.
13. You now have your image in WMS. Repeat for other Quads as necessary.

Louisiana DOTD Models Using the SCS Method

Creating an SCS Project

1. Using the *WMS Wizard* set the project coordinate system to UTM, NAD83, Zone 15
2. With the *Microsoft Virtual Earth Utility* identify your study area – make sure it is “bigger” than the study area
3. With *Web Services* download the DEM (1/3 arc Second recommended) and Topo map (aerial photograph too if you want)
    1. You can try to get the land use and soils data from the web services as well. They are currently on the Aquaveo server and there is not an intent to remove them but not a guarantee the will stay.
    2. You can download the land use file(s) from http://www.webgis.com/
    3. You can ask for the Ssurgo data and get an email with an ftp from: http://soildatamart.nrcs.usda.gov/
    4. Read these shapefiles into WMS and be sure they are converted to the UTM, NAD83, Zone 15 coordinate system
    5. If there are problems with the topo map you can go to http://msrmaps.com/or to http://atlas.lsu.edu/and navigate to download the DRG Files
    6. For the DEM you can go to http://nationalmap.gov/viewer.html/
4. Delineate the watershed.
    1. Turning on the *Create TC Coverage* is optional
5. Select the HEC-HMS model (you won’t use this but it will line up best with what you want to do for the SCS method).
6. In the *Define Land Use* option
    1. Join the Ssurgo attributes tables by right clicking on each Ssurgo shape file and choosing the *Join Ssurgo Data* option
    2. Back in the wizard Create your Land Use and Soils coverage
7. Under *Hydrologic Computations* in the Wizard.
    1. Compute GIS Attributes to determine the composite CN
    2. In this you will need to load the standard LADOTD SCS table (the current definition for Residential is for ¼ acre lots, you may want to adjust this value or any other as appropriate).
    3. Compute Basin Data and select the **Lag Time – SCS** equation.
8. You can set up the Job Control and Precipitation data for an HMS model if you want, but it is not necessary.
9. Clean up the model.
    1. Turn off the *Model Checker*
10. Exit the wizard and identify the data necessary to run your model.
11. Go into the *Display Options* under *Drainage* and turn on the *Maximum Flow Distance and Maximum Flow Slope*.
12. Save your WMS Project
13. Open the SCSRainfall.jpg [8] file.
14. Run your model with the standard LADOTD program.
    1. http://www.dotd.louisiana.gov/highways/road_design/hydr/

LADOTD SCS table

```
11, "Residential", 61.000000, 75.000000, 83.000000, 87.000000, 0.000000
12, "Commercial", 89.000000, 92.000000, 94.000000, 95.000000, 0.000000
13, "Industrial", 81.000000, 88.000000, 91.000000, 93.000000, 0.000000
14, "Roads", 95.000000, 95.000000, 95.000000, 95.000000, 0.000000
15, "Industrial and Commercial Complexes", 85, 90, 92, 94, 0
16, "Mixed Urban or Built-up Land", 81, 88, 91, 93, 0
17, "Other Built-up Land", 81, 88, 91, 93, 0
21, "Cropland", 54.000000, 70.000000, 80.000000, 84.000000, 0.000000
22, "Orchards, Groves, Vineyards, Nurseries, and Ornamental Horticultural Areas", 67, 76, 83, 86, 0
23, "Confined Feeding Operations", 67, 76, 83, 86, 0
24, "Other Agricultural Lands",  67, 76, 83, 86, 0
31, "Herbaceous Rangeland", 54, 70, 80, 84, 0
32, "Shrub and Brush Rangeland", 54, 70, 80, 84, 0
33, "Mixed Rangeland", 54, 70, 80, 84, 0
41, "Forest", 36.000000, 61.000000, 74.000000, 80.000000, 0.000000
42, "Evergreen Forest Land", 36, 61, 74, 80, 0
43, "Mixed Forest Land", 36, 61, 74, 80, 0
51, "Streams and Canals", 100.000000, 100.000000, 100.000000, 100.000000, 0.000000
52, "Lakes", 100, 100, 100, 100, 0
53, "Reservoirs", 100.000000, 100.000000, 100.000000, 100.000000, 0.000000
54, "Bays and Estuaries", 100, 100, 100, 100, 0
61, "Forested Wetland", 36.000000, 61.000000, 74.000000, 80.000000, 0.000000
62, "Nonforested Wetland", 36.000000, 61.000000, 74.000000, 80.000000, 0.000000
71, "Dry Salt Flats", 50, 60, 70, 80, 0
72, "Beaches", 30, 38, 45, 50, 0
73, "Sandy Areas other than Beaches",, 30, 38, 45, 50, 0
74, "Bare Exposed Rock", 95, 96, 97, 98, 0
75, "Strip Mine Quarries, and Gravel Pits", 40, 50, 60, 70, 0
76, "Transitional Areas", 36.000000, 61.000000, 74.000000, 80.000000, 0.000000
77, "Mixed Barren Land", 36, 61, 74, 80, 0
81, "Shrub and Brush Tundra", 60, 70, 80, 90, 0
82, "Herbaceous Tundra", 60, 70, 80, 90, 0
83, "Bare Ground Tundra", 60, 70, 80, 90, 0
84, "Wet Tundra", 60, 70, 80, 90, 0
84, "Mixed Tundra", 60, 70, 80, 90, 0
91, "Perennial Snowfields", 80, 84, 88, 92, 0
92, "Glaciers", 80, 84, 88, 92, 0
```

Louisiana DOTD Models Using the USGS Method

USGS Project

1. Using the *WMS Wizard* set the project coordinate system to UTM, NAD83, Zone 15
2. With the *Microsoft Virtual Earth Utility* identify your study area – make sure it is "bigger" than the study area
3. With *Web Services* download the DEM (1 arc Second recommended for larger areas) and Topo map (aerial photograph too if you want)
    1. If there are problems with the topo map you can go to http://msrmaps.com/or to http://atlas.lsu.edu/and navigate to download the DRG Files
    2. For the DEM you can go to http://nationalmap.gov/viewer.html
    3. You shouldn't need the Land Use and Soils files for these projects because they do not require you to compute CN, but if you want to run HMS or something else then you can refer to the SCS document to process the land use and soils files
4. Delineate the watershed
    1. Before delineating turn on the *Create TC Coverage* option
    2. Set the *Accumulation Threshold* to 1-3 sq. miles
5. Clean up the model
    1. Turn off the *Model Checker* if you want
6. Exit the wizard and identify the data necessary to run your model.
7. Use the Maximum Flow Slope *5280 to get feet/mile
8. Save your WMS Project.
9. Open the MeanRainfall.jpg [9] or MeanRainfallUpdated.jpg [10] file and determine the Mean Annual Rainfall.
10. Run your model with the standard LADOTD program.
    1. http://www.dotd.louisiana.gov/highways/road_design/hydr/

References

[1] http://www.xmswiki.com/
[2] http://www.xmswiki.com/xms/GSDA:GSDA
[3] http://nationalmap.gov/viewer.html
[4] http://www.webgis.com/
[5] http://soildatamart.nrcs.usda.gov/
[6] http://atlas.lsu.edu/
[7] http://atlas.lsu.edu/q24k/1:24,000
[8] ftp://pubftp.aquaveo.com/download/fhwa/louisiana/SCSRainfall.zip
[9] ftp://pubftp.aquaveo.com/download/fhwa/louisiana/MeanRainfall.zip
[10] ftp://pubftp.aquaveo.com/download/fhwa/louisiana/MeanRainfallUpdated.zip

Maricopa County Modeling

Tips and Locations for Downloading Hydrologic Data

GSDA Site

HEC-1 Modeling with Green and Ampt

1. Read your DEM into WMS (You can download DEMs from the USGS Seamless site [1]).
2. Convert coordinates of the DEM when reading:
   1. **Convert from:** Horizontal system of Geographic NAD 83 and vertical system of NAVD 88, Meters.
   2. **Convert to:** State Plane NAD 83, Arizona Central Zone, horizontal and vertical units of US Survey feet, NAVD 88.
3. Read any image data you want to use into WMS.
4. Run Topaz (Drainage module)
5. Set your flow accumulation threshold (*Display Options* | select **DEM Data** option)
6. Create your outlet(s) (Drainage module tools)
7. Use the delineate basins wizard
8. Read your Land Use shapefile. You can download land use shapefiles from here [2]. The file ending in "giras.exe" will be the file you want. The soils shapefile will be in the file ending in "core31.exe". You will need a HUC number for your watershed, which you can get by search for your watershed on this site [3].
9. If needed, convert the coordinates of your land use data to State plane coordinates as described for DEMs above.
10. Create a new Land Use coverage and make sure it's set as the active coverage.
11. Go to the GIS module, drag a box around your watershed, and select the *Mapping* | *Shapes→Feature Objects* command.
12. Make sure the land use ID is mapped to the correct field in your land use coverage. You can also map the land use name if needed.
13. Read your soils shapefile. The soil shapefile for Maricopa County can be downloaded with a bunch of other data in the FCDMC.zip file here [4]. The "statsgo" (statewide, less accurate than "ssurgo" data) soil shapefile for any watershed in the US can be downloaded with the file ending in "core31.exe" from this location [2]. "Ssurgo" data can be downloaded from the soil data mart here [5].
14. If needed, convert the coordinates of your soil data to State plane coordinates as described for DEMs above.
15. Create a new Soil Type coverage and make sure it's set as the active coverage.
16. Go to the GIS module, drag a box around your watershed, and select the **Mapping | Shapes→Feature Objects** command.
17. Make sure the soil type ID is mapped to the correct field in your soil type coverage. You can also map the soil type name if needed.
18. Define the necessary Green and Ampt parameters for each land use type in the land use coverage. You do this by viewing the attributes of the land use type polygons in the land use coverage and viewing the Green and Ampt parameters. Several sample tables exist that give these Green and Ampt parameters for different land use types. One table is called "landusemagtable.tbl" in the FCDMC.zip file you can download here [4].
19. Define the necessary Green and Ampt parameters for each soil type in the soil type coverage. You do this by viewing the attributes of the soil type polygons in the soil type coverage and viewing the Green and Ampt parameters. Several sample tables exist that give these Green and Ampt parameters for different soil types. One table is called "soilwmsgreenapmt.tbl" in the FCDMC.zip file you can download here [4].
20. Go to the Hydrologic modeling module and select **Calculators | Compute GIS Attributes** from the menu. Change the computation type to "Green-Ampt Parameters" and select OK to compute the Green Ampt

parameters.
21. Make sure your current model is set to HEC-1 in the drop-down box at the top of the *WMS* window.
22. Select the **HEC-1 | Job Control** menu option.
23. Select **Initialize Maricopa County Precipitation Data**.
24. Select a rainfall grid (from the FCDMC.zip file you have downloaded) such as "noaa100y24h", which means 100-Year, 24-Hour precipitation.
25. Select the "Compute Precipitation" button and select **OK**. Select **OK** to close the *HEC-1 Job Control* Dialog.
26. Double-click on your basin and view your basin data and loss method data to make sure they are setup correctly. Select the "Unit Hydrograph Method" button in the **Edit HEC-1 Parameters** dialog.
27. Select the Clark method. Then select **Compute Tc and R – Maricopa County**. Enter the appropriate parameters and select **OK** to compute the Tc and R values.
28. Define routing if desired.
29. Run HEC-1.

Download Maricopa County Tutorials Here [5]

More information about the Maricopa County interface development is located here [6]

References

[1] http://seamless.usgs.gov/
[2] http://www.epa.gov/waterscience/ftp/basins/gis_data/huc/
[3] http://cfpub.epa.gov/surf/locate/index.cfm
[4] http://wms.aquaveo.com/FCDMC.zip
[5] http://wmsdocs.aquaveo.com/maricopatutor80.pdf
[6] http://emrl.byu.edu/FCDMC/

SWMM

WMS 8.1 contains an interface to the Storm Water Management Model (SWMM). WMS is integrated with the xpswmm version of this model and exports XPX files for use in xpswmm using the Export XPX file menu command in the river module. Several scenarios can exist where WMS is useful for developing a SWMM model. Here are three scenarios the WMS developers have considered:

Case 1: Exporting a hydrologic model to xpswmm

Modeling process: 1. Delineate watersheds in a drainage coverage and compute the basin data.

2. In the hydrologic modeling module, make xpswmm the active model; define/compute xpswmm hydrologic data as desired (in a spreadsheet-based dialog).

3. Save an XPX file or launch xpswmm from WMS. Running will implicitly save an XPX file, start xpswmm, and read the XPX file into xpswmm. The polygons from the drainage coverage are saved to the XPX file as polygons, outlet points are saved as nodes, and streams between outlet points are saved as inactive links.

Case 2: Exporting a storm drain coverage and hydrologic results

Modeling process:

1. Delineate watersheds in a drainage coverage and compute the basin data.

2. Create and draw arcs representing pipes or channels in a storm drain coverage.

3. Convert the storm drain coverage arcs to a hydraulic schematic in the river module.

4. Link drainage coverage outlet nodes to storm drain coverage inlet nodes.

5. Define/compute xpswmm hydrologic data as desired as in step 2 of case 1.

6. Save an XPX file or launch xpswmm from WMS. From the drainage coverage, only the polygons are saved (no outlets or streams). Links and nodes are saved from the storm drain coverage and the hydraulic schematic to the XPX file.

Case 3: Starting a model in xpswmm

Modeling process:

1. In xpswmm, define the stream network, pipe linkages, and drainage outlet/storm drain inlet nodes. Save this data as an XPX file.

2. Read the XPX file into WMS. WMS creates a storm drain coverage from the XPX file and reads all the data associated with the links and nodes.

3. Use the storm drain coverage as a base map for creating a drainage coverage and delineating watersheds (drain inlets become outlets for the drainage coverage). Compute basin data.

4. Link drainage coverage outlet nodes to storm drain inlet nodes. (Same as #3 in case 2.)

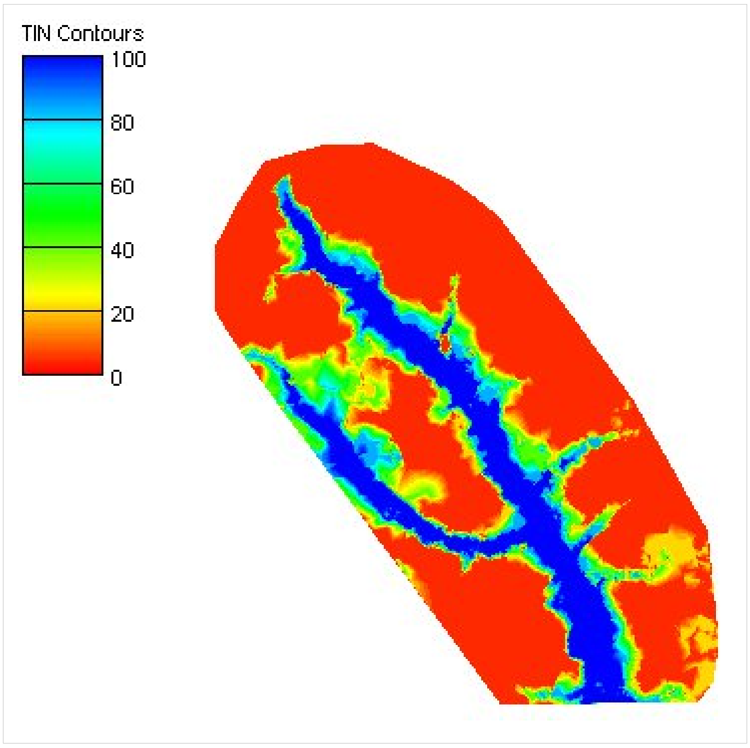
5. Define/compute xpswmm hydrologic data as desired as in step 2 of case 1.

6. Save an XPX file or launch xpswmm (as in step 5 for case 2). All the data defined or modified in WMS is written to the new file and updated in the xpswmm model.

Stochastic Modeling

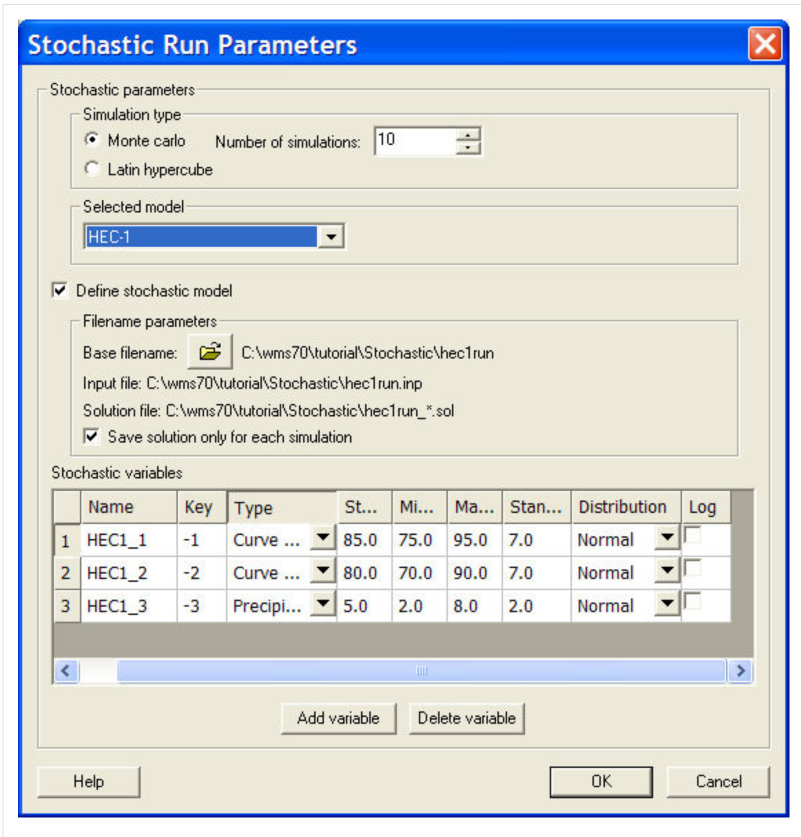
There is always a great deal of uncertainty in hydrologic and hydraulic modeling and the parameters that are used to develop solutions. Despite this, a typical flood plain boundary is black and white in that you are either in or out of the flood plain. A good engineer might be able to dispute a flood plain boundary by performing a hydrologic/hydraulic analysis with a set of equally probable parameters that results in a difference in the flood plain delineation. Until recently, computer programs lacked the ability to consider multiple probable answers and report a probabilistic floodplain boundary, but with the Stochastic Modeling tools in WMS this is possible using a combination of HEC-1 for hydrologic analysis, HEC-RAS for 1D hydraulic river modeling, and the WMS flood plain delineation tools.

You can "connect" the results of HEC-1 to a developed HEC-RAS model and then run them as many times consecutively, with the results of the HEC-1 analysis feeding the boundary conditions for an HEC-RAS model. Certain parameters (at this point only basin CN and precipitation within HEC-1, and Manning's roughness within HEC-RAS) can be varied within a range of reasonable values using Monte Carlo or Latin Hypercube simulations in order to create a number of simulations. The results of each HEC-RAS model can then be used to delineate a series of flood plains. The combination of all floodplains can then be examined in order to derive a "probabilistic" flood plain where a region flooded by 100% of the model simulation combinations can be distinguished from an area that is flooded by only 50% of the models as shown in the figure below:



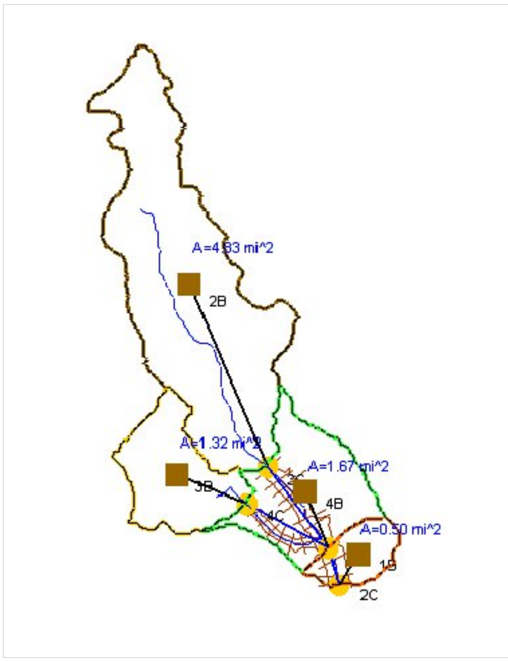
The following steps outline the process for developing a stochastic model of floodplain boundaries using WMS, HEC-1, and HEC-RAS.

Step 1. Develop a hydrologic model with HEC-1



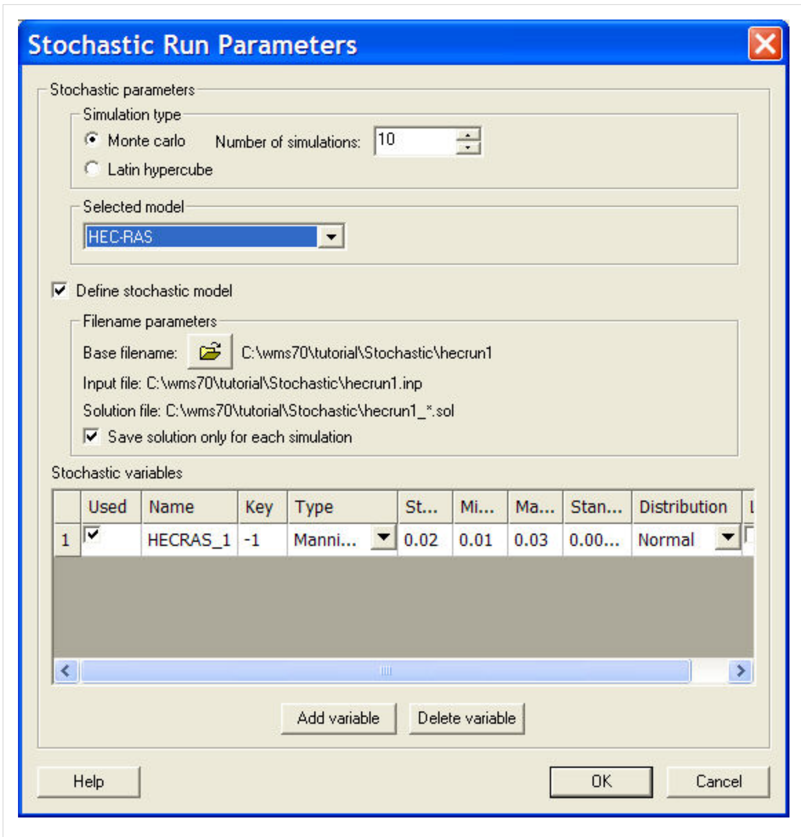
The HEC-1 Interface tools in WMS can be used to develop a working HEC-1 model. It is important that this model be running effectively (i.e. calibrated and/or adjusted to give credible results) prior to using it for the Stochastic Modeling simulation.

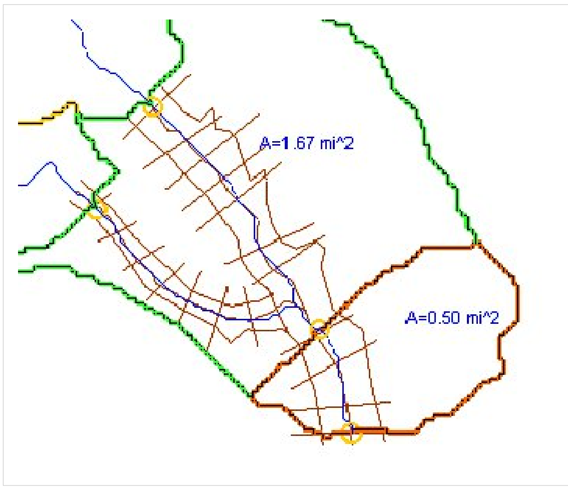
The only parameters at this point that can be varied within a range of probable answers are rainfall and CN (curve number for the different basins. This is done by setting the precipitation or CN to be a negative number in their respective dialogs. This negative number is a key number and should be unique for each stochastic variable you create. When running the stochastic simulation WMS will substitute the simulation specific parameter for the defined key. You can then setup a stochastic variable for HEC-1 in the *Stochastic Run Parameters* dialog. A key value (matching the key you defined in the materials property) starting value, min value, max value, standard deviation and distribution type.



Step 2. Develop a working model in HEC-RAS

The HEC-RAS interface tools in WMS can be used to developing a working HEC-RAS model. It is important that this model be running effectively (i.e. calibrated and/or adjusted to give credible results) prior to using it for the Stochastic Modeling simulation.





The only parameter at this point that can be varied within a range of probable answers are Manning's coefficients for the different material types. This is done by setting the roughness to be a negative number in the HEC-RAS Materials dialog. This negative number is a key number and should be unique for each stochastic variable you create. When running the stochastic simulation WMS will substitute the simulation specific parameter for the defined key. You can then setup a stochastic variable for HEC-RAS in the *Stochastic Run Parameters* dialog. A key value (matching the key you defined in the materials property) starting value, minimum value, maximum value, standard deviation and distribution type.

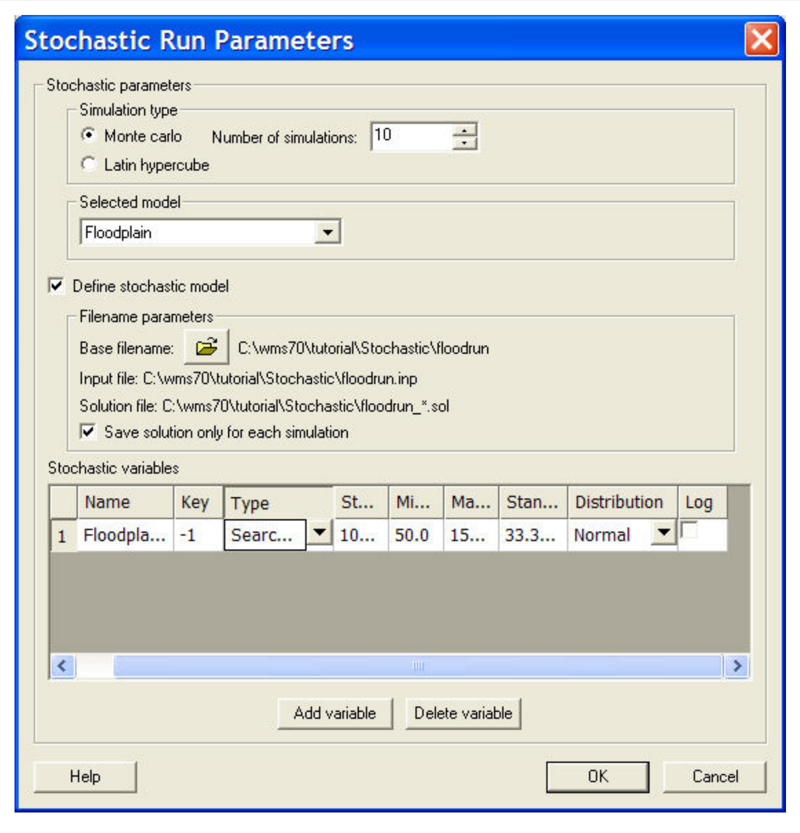
Step 3. Establish appropriate flood plain delineation parameters for your area

The floodplain delineation portion of the stochastic modeling uses the results from each HEC-RAS model to develop a floodplain for each run. The floodplain delineation is the same model used by WMS to perform individual floodplain delineations.

The only parameters at this point that can be varied within a range of probable answers the search radius of the flood plain delineation. This is done by setting the search radius in the *Floodplain Delineation Options* be a negative number. This negative number is a key number and should be unique for each stochastic variable you create. When running the stochastic simulation WMS will substitute the simulation specific parameter for the defined key. You can then setup a stochastic variable for Floodplain in the *Stochastic Run Parameters* dialog. A key value (matching the key you defined in the materials property) starting value, min value, max value, standard deviation and distribution type.

Step 4. Set up stochastic simulation parameters

There are several simulation parameters that control a stochastic simulation. They are defined in the *Stochastic Run Parameters* dialog shown below. Each section of this dialog is discussed below.



Simulation Type

The simulation type and number of simulations can be set. In a Monte Carlo simulation, each specified input variable is randomly varied within a specified minimum and maximum value a given number of times. If only a few simulations are run it may not be guaranteed to fully explore the parameter space. A Latin Hypercube simulation, on the other hand, divides the range into intervals and insures that parameters are chosen from each interval. With this kind of simulation you are more likely to explore the parameter space with fewer simulations.

Stochastic Models

HEC-1 or TR-20 for hydrologic modeling, HEC-RAS for hydraulic modeling, and Floodplain Delineation are the only currently available models for stochastic modeling. For each model you include a basin input file and solution files directory needs to be defined. For HEC-1, TR-20, or HEC-RAS you should select the input file of the already created model. These models will have key values (negative numbers) for the input parameters that will be defined as stochastic variables. The current floodplain delineation options will be saved in the flood run file.

You can add stochastic variables for any of the models. Each stochastic variable requires a key value (a negative number that has been entered in place of a parameter such as precipitation), a type, a starting value, a minimum value, a maximum value, a standard deviation, and a distribution. The distribution can be either normal or uniform and optionally defined as log.

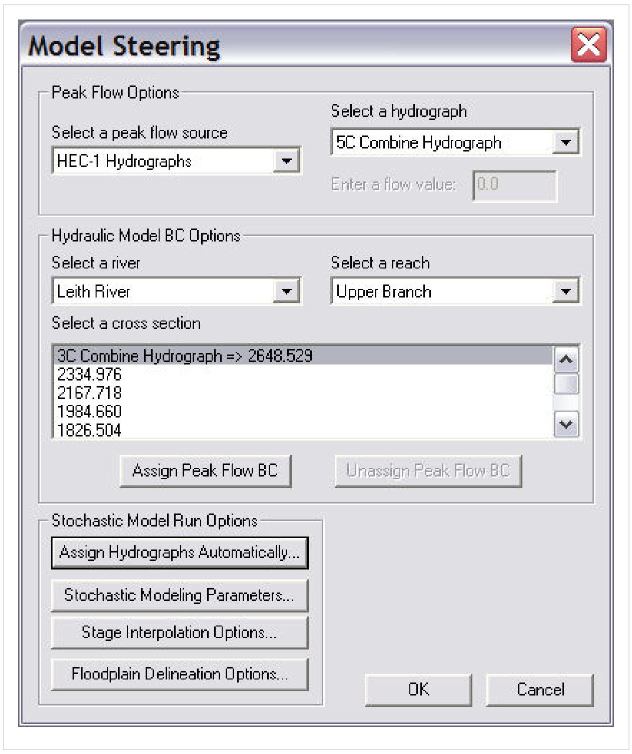
Step 5. Assign boundary conditions between models and run the stochastic simulation

The final step before running a stochastic simulation is to link any required models. In particular you have to link how the hydrologic modeling results are used as boundary conditions for the HEC-RAS hydraulic model. When you choose the *Run Stochastic Model* option the *Model Steering Dialog* appears. For the hydrologic model you are using (HEC-1) you must assign the appropriate hydrograph (basin or outlet) to the river reach and section in the hydraulic model.

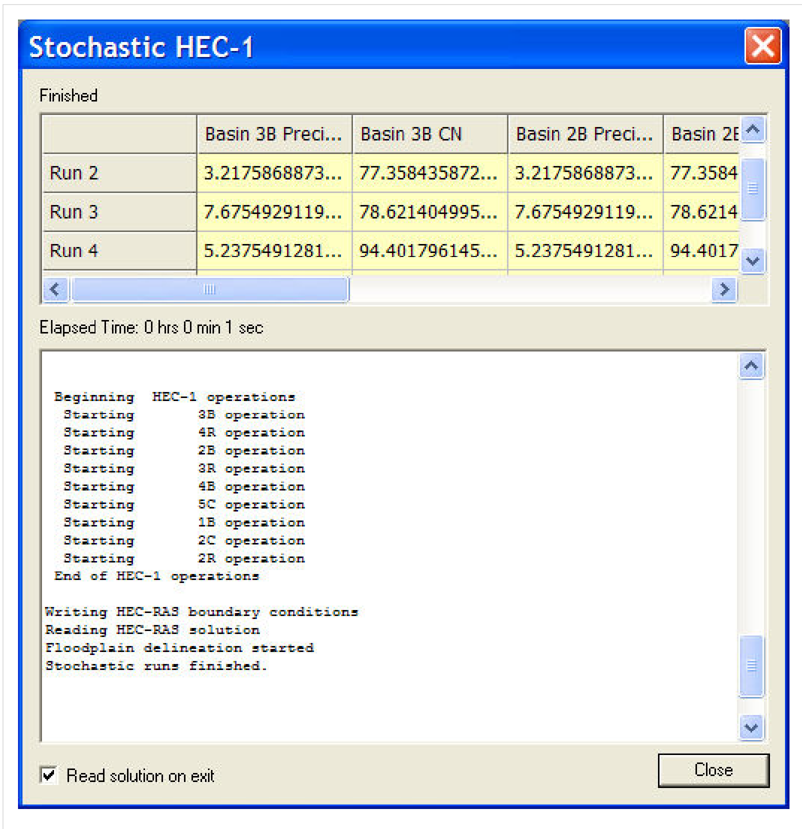
The option to *Assign Hydrographs Automatically* can be used providing a drainage coverage and centerline coverage are being used to set up the hydrologic and hydraulic models.

After identifying the hydrograph and the river, reach, and cross section station select the **Assign Hydrograph BC** button to link the models for this point. Continue until all of the appropriate model locations are linked.

When selecting **OK** your model simulation will run.



Step 6. Run the simulation



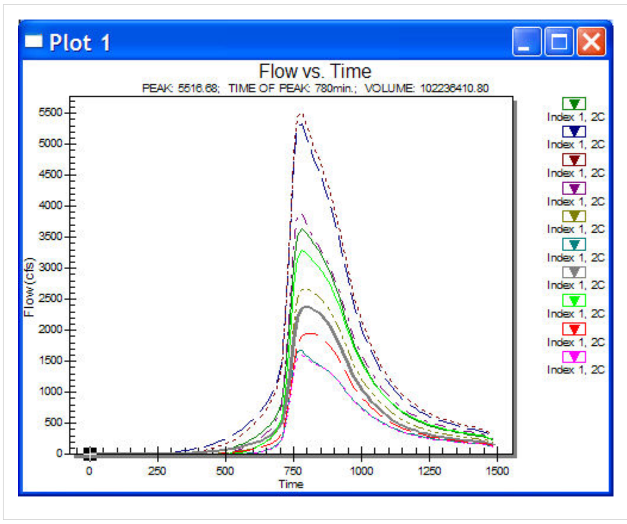
Once the simulation begins running you will see the parameters selected for each model run, as well as the status of each run. The *Read solution on exit* toggle is on by default and will cause that all model solutions (hydrographs, water surface elevations, and floodplain delineations) are read when the simulations are completed.

Step 7. Post Process the results

After finishing a stochastic simulation there are two primary results read back into WMS for each simulation: hydrographs from the HEC-1 model, and the floodplain depths and water surface elevations for each run.

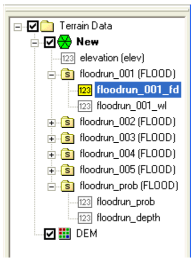
Hydrographs

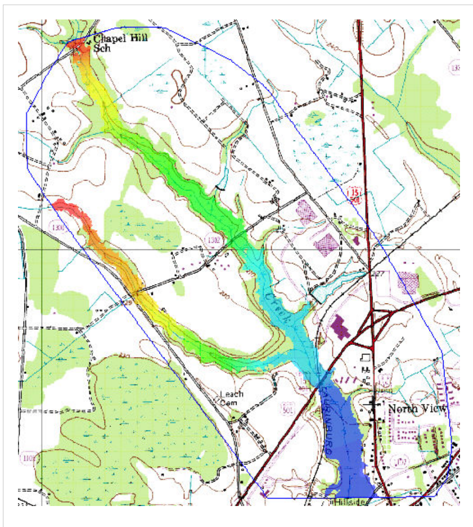
A series of hydrographs are loaded for each hydrograph station and can be viewed in the normal way hydrographs are viewed.



Floodplains

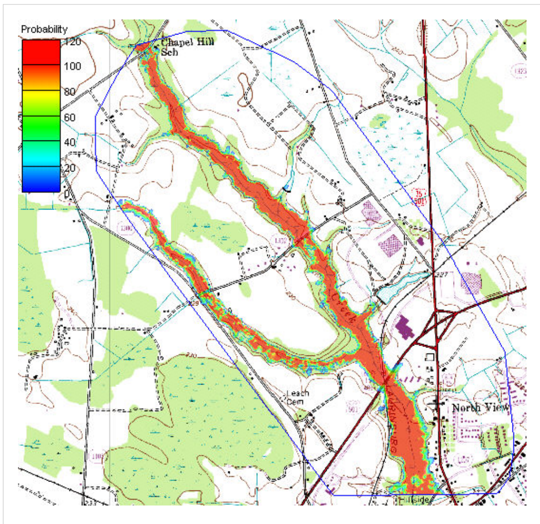
Each floodplain delineation results in a water surface elevation and a flood depth dataset. Each pair of datasets are organized in a folder underneath the TIN in the *Project Explorer*. You can set the contour options for a TIN and select the dataset you wish to be active and displayed from the *Project Explorer*.

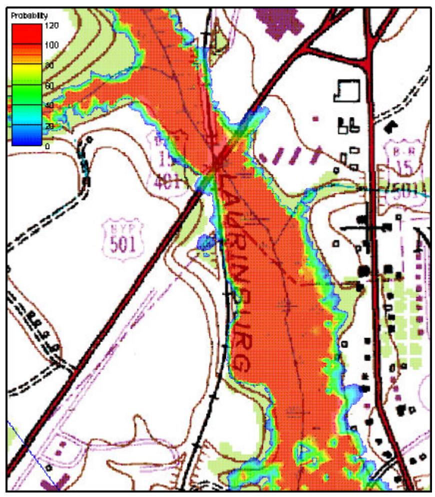




Probabilistic Floodplain Boundary

A final dataset showing the probability of flooding is also created from all of the individual floodplain datasets. The probabilistic flood plain indicates for each vertex on the TIN the percentage of model runs that resulting in inundation at the point.

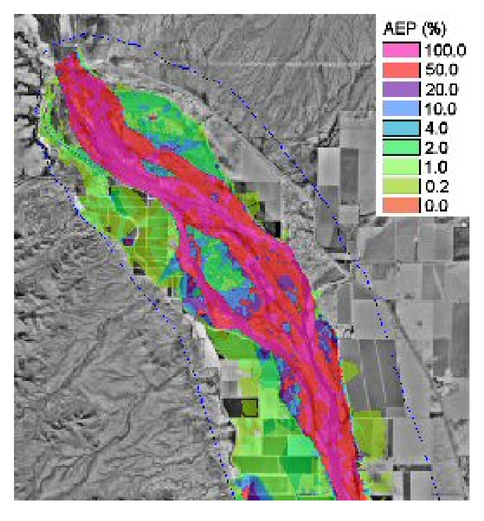


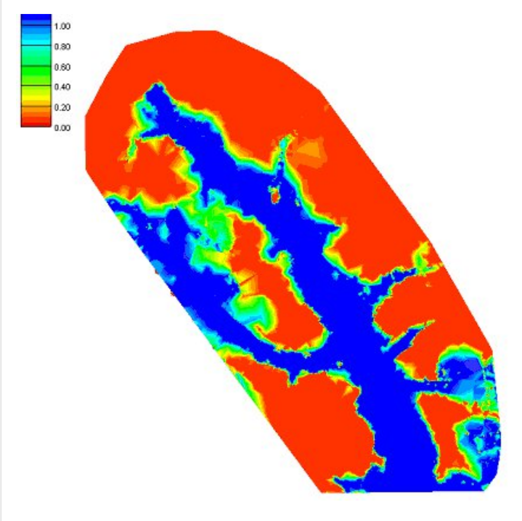


Close up view

Annual Exceedance Probability (AEP) Map

When using the stochastic hydrologic/hydraulic/floodplain delineation tools it is possible to generate an annual exceedance probability map. This is done by generating inundation maps that consider the range of all possible floods for all return periods. The result is a map that identifies the annual exceedance probability of flooding for every TIN vertex. For example if a point is flooded 10 times in 1000 simulations then it would represent the .01 probability. The **Return Period → Feature Objects...** command allows you to generate contours from the AEP map for specified return periods. You should remember that such a map does not represent a solution from a single set of input parameters, but is rather the composite of several hundred or thousand simulations.





Related Topics

HEC-1 Modeling

HEC-RAS Modeling

Flood Plain Delineation

6.1. CE-QUAL-W2

CE-QUAL-W2

Creating and Running a CE-QUAL-W2 Model using WMS

WMS is a very effective aid for setting up the CE-QUAL-W2 bathymetry and control files.

In order to create the bathymetry file, the user will need to create a TIN representing the bottom elevations of the reservoir. This can be created from contour maps, depth soundings, or digital contours in CAD format. The accuracy of the TIN is very crucial to calibration of the model.

The created TIN is opened in WMS under the map module mode. A new CE-QUAL-W2 simulation should be started before editing any of the TIN. To edit the TIN the user must create a branch coverage and then trace an outline of the water body using the create feature arc tool. The user then can divide the water body into separate branches according to his/her discretion.

Next, the branch coverage is duplicated and renamed as segment coverage. Segments are created using the *create feature arc* tool according to the user's discretion.

From here the user appropriately numbers the segments and branches. A tool is available that automatically numbers segments. Segment length and orientation is done using a tool designed for measuring distances. Then the user can create a storage-capacity curve. Options exist for plotting this curve as well as exporting the data that create this curve so that the user may compare it to an existing storage-capacity curve. In many instances the generated storage capacity curve will not match well with a given storage capacity curve. In this case the TIN will need to be calibrated to match the given storage capacity curve. Refer to Calibrating the TIN to match a given storage capacity curve and Verification of Bathymetric Data for more instructions. Finally, layers are created and widths calculated to complete the bathymetry creation process. For a more detailed description of setting up a bathymetry file the user should refer to Creating a CE-QUAL-W2 bathymetry file from a TIN.

If the user requires the WMS generated storage capacity curve to be very accurate when compared to a given storage capacity curve then it is recommended that the user create only one branch and segment to begin with in WMS. This makes generating a storage capacity curve simpler and faster. Once a reasonable storage capacity curve is created the user should then proceed to make a detailed bathymetry representation of the model.

With a completed bathymetry file the user can proceed to begin editing the control file. In the *Job Control* dialog the user can edit all of the fields that are needed for the W2 control file (*job control* dialog help file). Most of these values should be left at their defaults (if there are default values) unless the user has knowledge or experience that would dictate he/she do otherwise. Saving the simulation will produce the control file and bathymetry file.

Other input files will be needed to run a CE-QUAL model. WMS supports the creating and editing of some input files; however, it is recommended that the user employ a spreadsheet in the creation of these files because of the simplicity and speed of a spreadsheet for editing and formatting of text. Following is a list of the required files for a basic simulation (adding constituents, tributaries, precipitation, etc will require more input files to be created):

- Meteorological file
- Inflow file
- Outflow file
- Inflow temperature file
- Wind Sheltering file
- Shading file

- Bathymetry file
- Control file

It should be remembered that there is variation between every simulation and every user. All input files need to be spaced correctly or errors will prevent the program from running. Also, because of the vast amount of information it is possible that some information may be forgotten or misplaced. The best check for accurateness of the simulation is to run the preprocessor before running the actual model. This can help identify problems in the control file and other files. These errors and/or warnings are output to an error and/or warning file. After there are no further errors identified by the preprocessor the actual program should be run. Other errors may still exist that will prevent the program from running. These may be output to an error file and identified. If not then it is recommended that the user compare his/her simulation to any examples or known working simulations. This is a tedious way of isolating the problem, but it provides an opportunity to "get familiar" with the system, especially for new users. If none of this works there is a forum that can be used to contact the developers or ask questions of other users (http://www.cee.pdx.edu/w2/).

Processing the results can be done using many tools. A spreadsheet is a common tool and there are post processors available that simplify the process. A future update of WMS will contain post-processing features.

In most cases the user will be required to do some amount of calibration for the model to resemble observed data. The first calibration that should be performed is a water balance. The user should then look at calibrating the temperature. After a thermal calibration has been reached the user should then proceed to calibrate any constituents that are modeled. For help on calibration the user should refer to the CE-QUAL-W2 user's manual[1] as well as looking at other published reports on models done using CE-QUAL-W2.

Grid Z-Magnification

Note that it may be necessary to change the Z-magnification in the WMS Display Options dialog to better visualize the CE-QUAL-W2 grids. The Z-magnification controls the height of the grid display in the WMS graphics window.

Related Topics

- CE-QUAL-W2 Branches
- CE-QUAL-W2 Layer Editor
- CE-QUAL-W2 Menu

References

[1] http://www.ce.pdx.edu/w2/

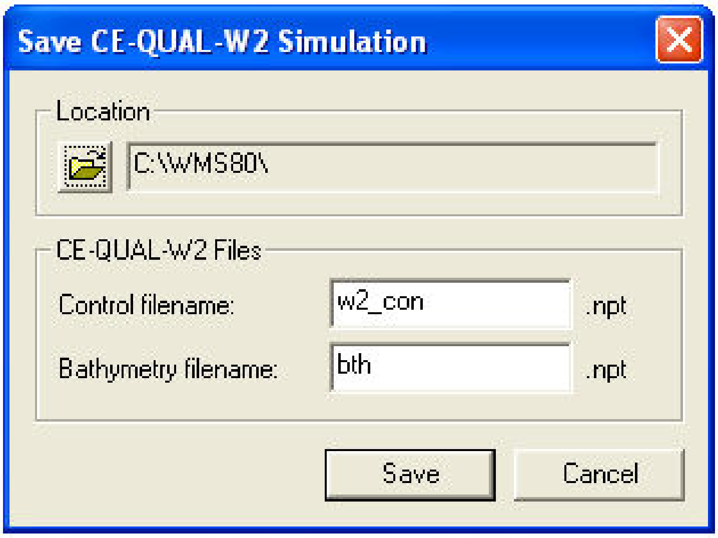
CE-QUAL-W2 Menu

New Simulation

The **New Simulation** command from the *CE-QUAL-W2* menu initializes the CE-QUAL data and calculations. In order to begin a bathymetry file or CE-QUAL-W2 model this must first be selected from the CE-QUAL-W2 option of the *Models* menu. If **New Simulation** is selected during a model it will clear all data and begin a new model.

Save Simulation

This function allows the user to create the input files that are used in the CE-QUAL-W2 model. The user should select the the correct folder to save the file to, highlight the filename that is desired and click **Save**. This function does not allow for multiple files to be saved at once. Refer to the *Job Control* help file for instructions on editing the input files.



Delete Simulation

This option of the *CE-QUAL-W2* menu clears all data for the current simulation.

Read Simulation

The CE-QUAL-W2 **Read Simulation** command reads a CE-QUAL-W2 bathymetry and control file. To read a CE-QUAL-W2 simulation, you should have defined the simulation using WMS. If you have not defined your simulation using WMS, you should have branch and segment coverages representing the branches and segments defined in the bathymetry file associated with the CE-QUAL-W2 control file.

To read a CE-QUAL-W2 simulation, select and open the control filename from the file browser after selecting this menu item.

Read Solution

A CE-QUAL-W2 solution can be read into WMS by selecting the **Read Solution** command. When you read the solution, 2D grids representing the layers and segments for each branch polygon are created. These grids are generated in the same way that grids are created when selecting the **Map→CE-QUAL-W2 Grids** menu command. WMS reads the solution values from the CE-QUAL-W2 snapshot files and the solution contours can be plotted for each time step that was output from the model.

Run CE-QUAL-W2

While the option to run a CE-QUAL-W2 simulation exists in the WMS interface, the WMS developers recommend running CE-QUAL-W2 outside of WMS to better view the progress of the run.

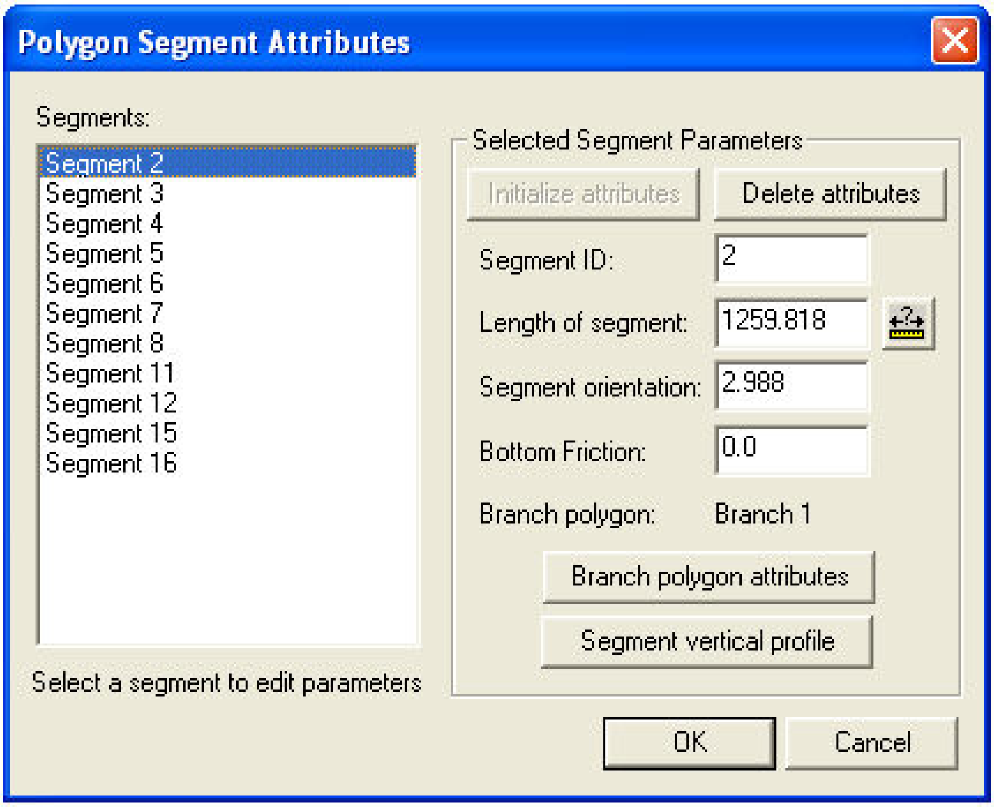
This menu item can be used to run a CE-QUAL-W2 simulation from the WMS model wrapper given the control filename. If the model runs to completion, the results can be read and analyzed in WMS using the **Read Solution** menu item.

Branches

This feature allows the user to edit the features of a branch. See CE-QUAL-W2 Branches for more information.

Segments

The **Segments** command in the *CE-QUAL-W2* menu allows the user to edit the parameters of a segment. The dialog below appears with a list of all created segments for the current model.



To edit a segment, the desired segment should be highlighted and then the segment parameters can simply be edited.

All attributes of the segment are shown. If the lengths of the branch or segment have not been obtained, the *Length of segment* and **Segment orientation** text boxes will be blank. The *measure* tool to the right of the *Length of segment* text box can be used to obtain the length of a particular segment. This tool also calculates the orientation. The

Branch polygon text box shows which branch the segment is assigned to. This can be changed using the dropdown box option.

The **Branch polygon attributes** button brings up the Polygon Branch Attributes dialog. Refer to the Branches help file for more information on the features of this dialog.

The **Segment vertical profile** button allows the user to view and edit the layers of the segment. (Refer to the Layers help file for information on creating layers)

Tributaries can be added to a segment by selecting the Segment tributaries button. A dialog is opened and the user can enter or import the data necessary to add a tributary to the simulation.

Map Segments and Branches

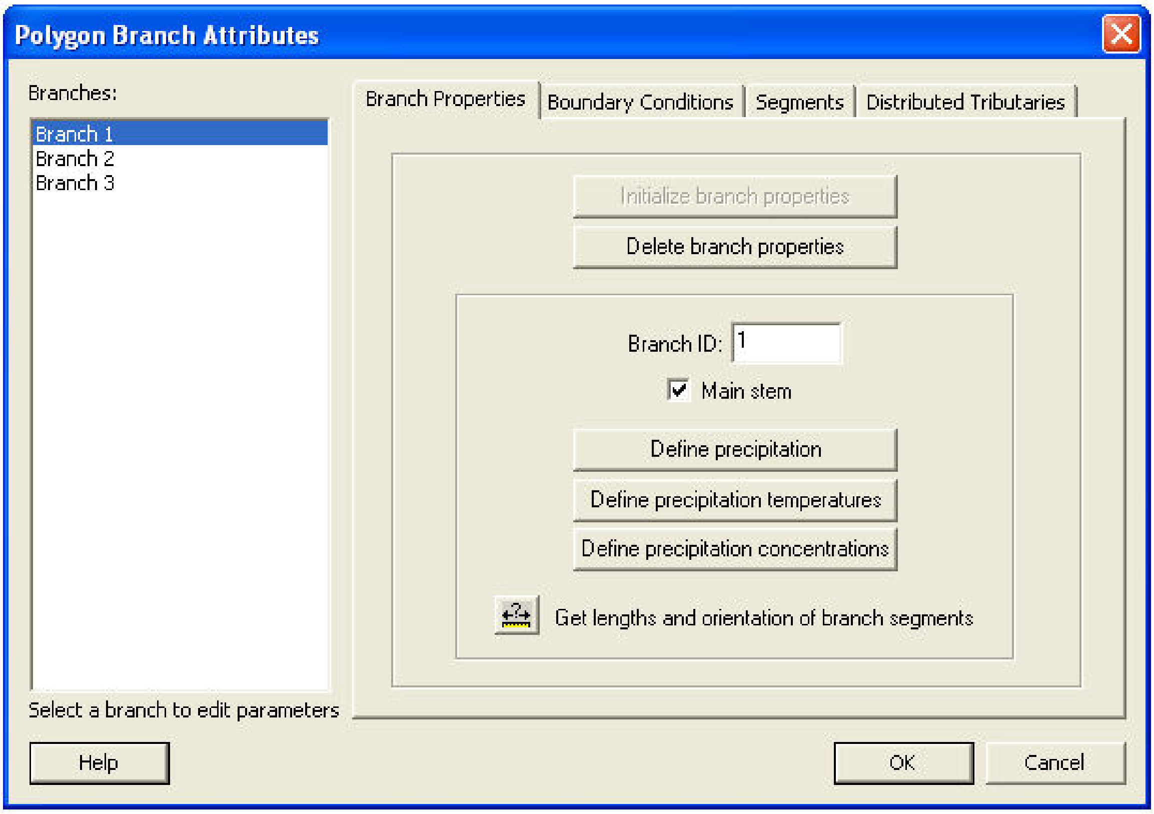
The **Map Segments→Branches** command in the *CE-QUAL-W2* menu assigns all created segments to the branch in which the segment polygon lies. The user should refer to the segment tab of a branch polygon editor to see the segments that are assigned to a branch.

Related Topics

- CE-QUAL-W2
- CE-QUAL-W2 Branches

CE-QUAL-W2 Branches

This feature allows the user to edit the features of a branch. This is done by highlighting the desired branch in the *Polygon Branch Attributes* dialog. The *Branch ID field* is used to number branches. The *Main stem* checkbox should be checked if the branch is the main stem of the waterbody. Checking the *Selective withdrawal* checkbox activates the **Structures** button. Selecting this button opens a dialog for defining any selective withdrawal structures the user may wish to add. The **'Define precipitation**, **Define precipitation temperatures**, and **Define precipitation concentrations** buttons open dialogs in which the user can add precipitation and its temperature and constituent concentrations to the CE-QUAL simulation. Each dialog also features an import function which allows the user to import either a W2 input file or a comma or space delimited text file. The *Get lengths and orientations of branch segments* tool allows the user to trace the flow path of the branch and returns the lengths and orientations of each segment in the branch.



The *boundary conditions* tab of the *polygon branch attributes* dialog is used to define the upstream and downstream boundary conditions. For the upstream boundary condition there are three options to select from: external head, external flow, and internal head.

When *external head* is selected the **Define head elevations**, **Define head temperatures**, and **Define head concentrations** buttons become active. Selecting these buttons opens dialogs that allow the user to specify the elevations, temperatures, and concentrations of the external head.

When *external flow* is selected the **Define inflow flowrates**, **Define inflow temperatures**, and **Define inflow concentrations** buttons become active. These buttons open dialogs that allow the user to specify any inflows and the temperature and constituent concentration of the inflow.

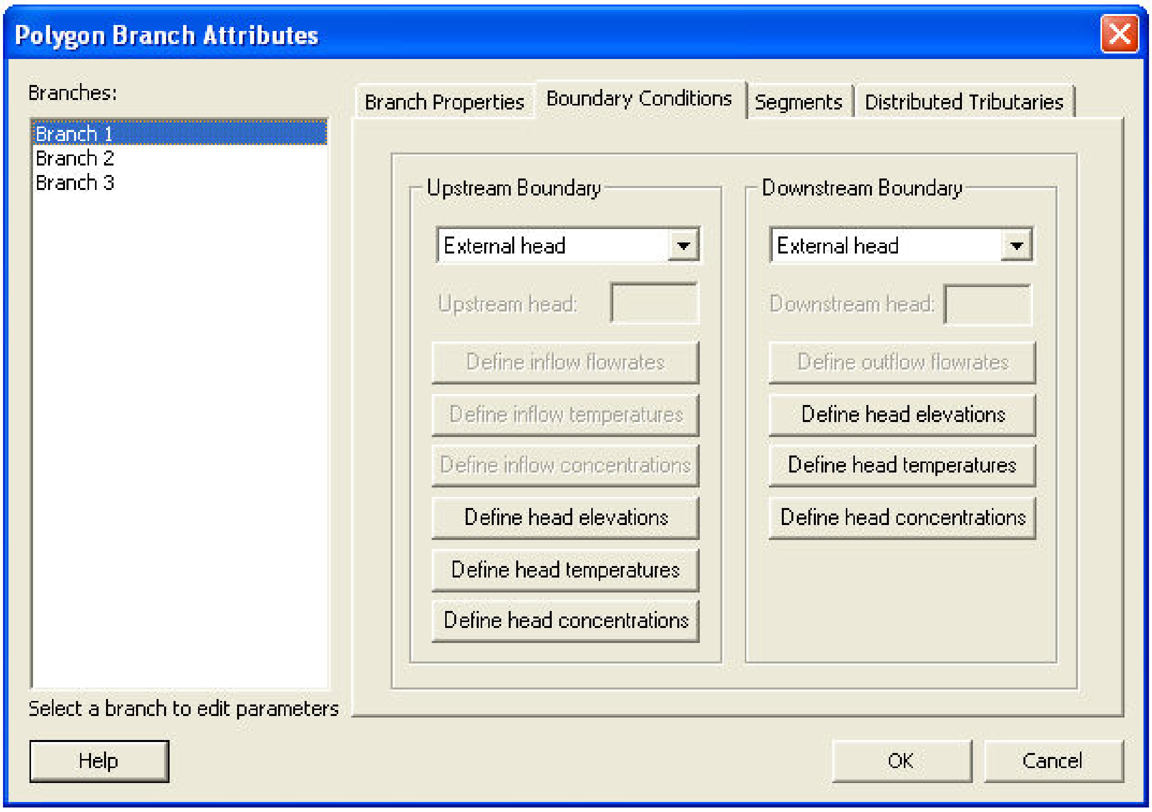
Selecting *internal head* specifies that the branch is attached to another branch. Nothing else is required when this option is selected.

The downstream boundary condition has three options to choose from: external head, external flow and internal head.

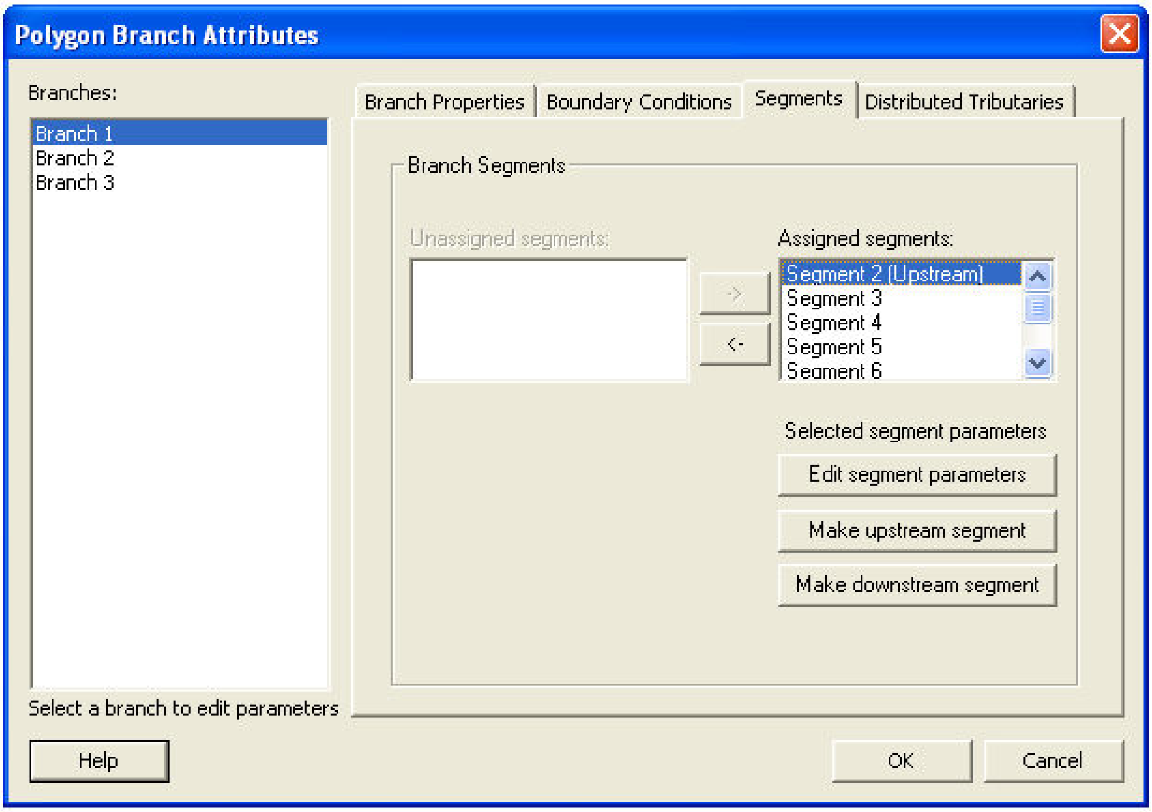
When *external head* is selected the **Define head elevations**, **Define head temperatures**, and **Define head concentrations** buttons become active. Selecting these buttons opens dialogs that allow the user to specify the elevations, temperatures, and concentrations of the external head.

When *external flow* is selected the **Define outflow flowrates** button becomes active. This button leads to a dialog that allows the user to specify any outflows from the branch.

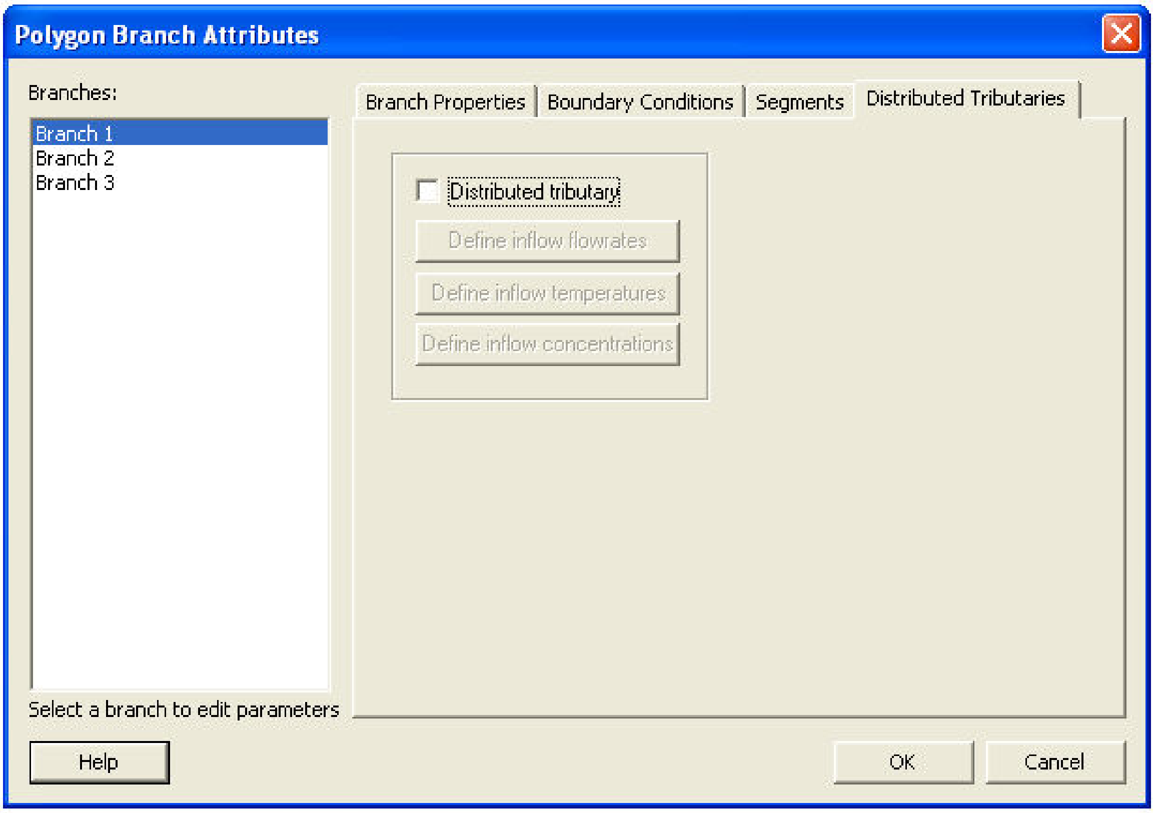
Selecting *internal head* specifies that the branch is attached to another branch. Nothing else is required when this option is selected.



In the *segment* tab of the *branch attributes* dialog (shown below) the user can edit segment parameters of the branch and assign upstream and downstream segments. This is done by highlighting the desired segment and then selecting the appropriate segment parameter button.



The *distributed tributary* tab is used to add a distributed tributary to the branch. To do so the user must check the box which then activates the **Define inflow flowrates**, **Define inflow temperatures**, and **Define inflow concentrations** buttons. These buttons open dialogs where the user can create input files for the distributed tributary.



Related Topics

- CE-QUAL-W2
- CE-QUAL-W2 Segments
- CE-QUAL-W2 Map Segments and Branches

CE-QUAL-W2 Layer Editor

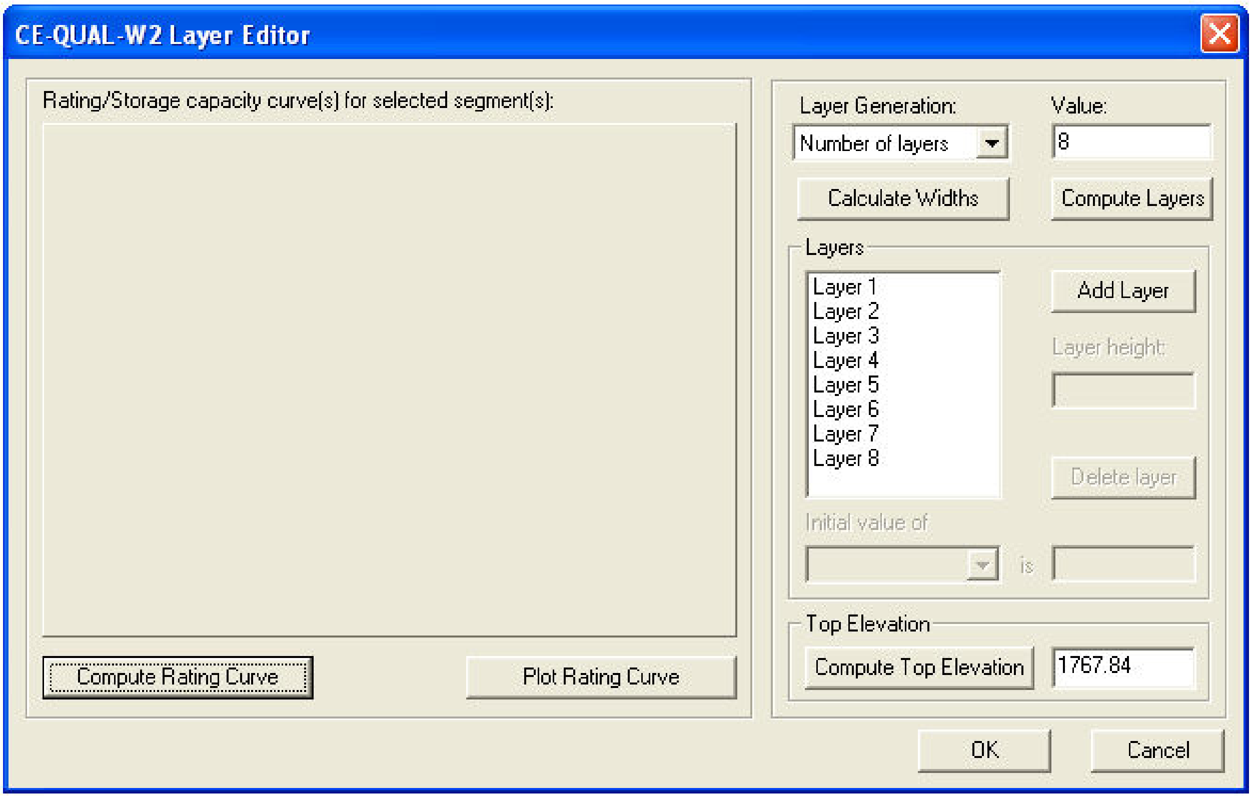
The layer editor is used for generating a storage capacity curve for a CE-QUAL-W2 application. The curve is computed from selected segment(s) of a representative waterbody and is generated by selecting the **Compute storage-capacity curve** button. Generation of the curve may take some time, especially where several segments have been selected.

The **Plot storage-capacity curve** button produces a separate image of the storage capacity curve. This image has several options which can be displayed by right clicking the mouse over the image.

The **Compute Top Elevation** button will give the top elevation of the TIN in the *Top elevation* text box. It should be noted that this is not necessarily the top elevation of the waterbody. If the top elevation of the waterbody is desired then it should be manually input into the *Top elevation* text box.

The *Layer Generation* pulldown box gives the user two options for the number of layers in the bathymetry file. If the user desires to prescribe the number of layers then the *Number of layers* option should appear in the box and the desired number of layers entered in the text box titled value. If the user desires a certain layer height then the *Layer heights* option should be selected with the desired value for layer heights entered in the Value text box. Selecting the **Compute** button will compute the number of layers and/or layer heights for the waterbody. The layers will then appear in the large text box for viewing. These can be edited according the user's needs by using the **Add Layer** and **Delete Layer** button. Individual layer heights are shown in the *Layer height* text box.

Selecting the **Calculate Widths** button will calculate the widths of all selected segments.



Related Topics

- CE-QUAL-W2

CE-QUAL-W2 Bathymetry

Introduction

The text and pictures on this page were written by E. James Nelson (BYU), Douglas J. Gallup (Aquaveo), and Christopher M. Smemoe (Aquaveo). The information contained in this text is copyrighted by Aquaveo (2013).

This page discusses the process WMS uses to compute bathymetric layer widths for CE-QUAL-W2 simulations. WMS is extremely useful for setting up CE-QUAL-W2 bathymetry files. WMS can also be used to setup CE-QUAL-W2 modeling parameters, but this pages focuses only on the bathymetry file setup.

Laterally Averaged Finite Difference Grids

Most hydrodynamic models are depth-averaged, meaning that the numerical mesh (finite difference or finite element) is oriented in the x-y plane with each element representing a vertical column, or an average depth, of the domain being modeled. Tools for generating depth-averaged numerical models have been well developed because of the number of applications (Fugal, 2000; Thibodeaux, 1992; Gaspar et. al., 1994). Average depths for each element are determined by estimating an initial water surface level and then calculating the depth at each element centroid using bathymetric elevation data. The typical process requires the user to define a spatial domain of the model and then appropriately fill the regions with finite elements (quadrilaterals and/or triangles). Finally, a depth for each column is determined by calculating the difference between the assumed or starting water surface elevation of the element and the elevation determined from data gathered that describes the underlying bathymetry (see Figure 1).

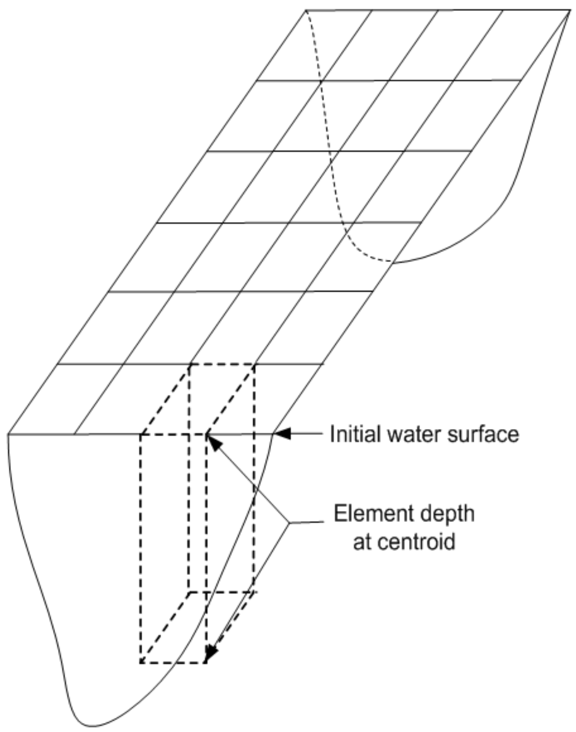


Figure 1: Depth calculation for a depth-averaged finite difference grid.

A major limitation of depth-averaged models is that they do not represent vertical variations well. For example, velocities computed using such a model do not account for variations that may occur between the top and bottom of the model since a single, average, value for each vertical column is computed. On the other hand, a laterally averaged numerical model better represents vertical variations, an important aspect of deeper water bodies such as lakes and reservoirs. However, in order to develop a laterally averaged numerical model an average width for each element must be determined. Unlike the depth-averaged models it is not possible to assume some initial “bank location” and then estimate the width to each element. Instead the bathymetry describing the shoreline elevations on either side of the model must be known so that a width at each element can be estimated as illustrated in Figure 2.

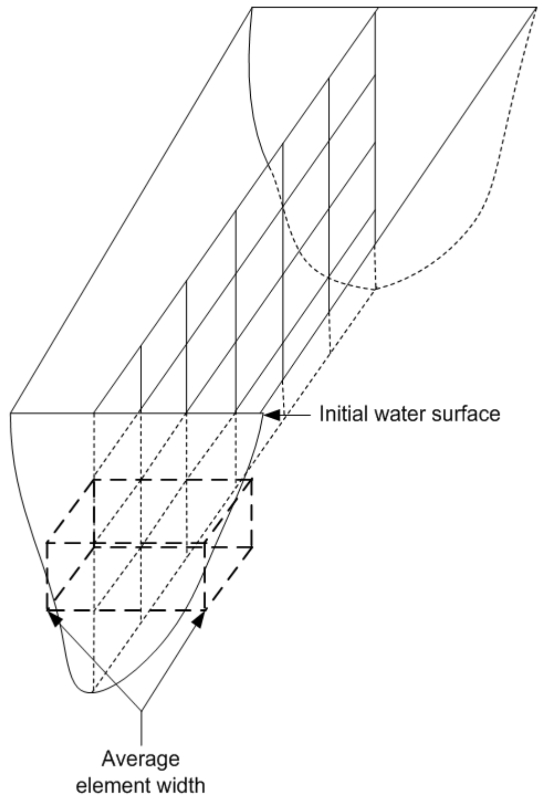


Figure 2: Width calculation for a laterally-averaged finite difference grid.

To develop the geometric representation for such models, discretization along the length and depth of the model must first be determined. In CE-QUAL-W2, segment lengths are determined from the spatial orientation of the water body. Tributary inflows, widening/narrowing of the water body and sampling or computational points are all indicators of how to divide the model up along its length into segments. Layer depths are generally a function of the sensitivity, or gradient of the vertical variations of the variables being analyzed (i.e. temperature, phosphorous, dissolved oxygen, etc.). The higher the gradient the smaller the element heights should be. Once lengths and heights are determined a finite difference grid representing the profile of the water body can be constructed as shown in Figure 3. Cells are inactive if their centroid elevation drops below the bottom elevation of the water body. The final dimension required for the grid geometry of the model is an average width as illustrated in Figure 3.

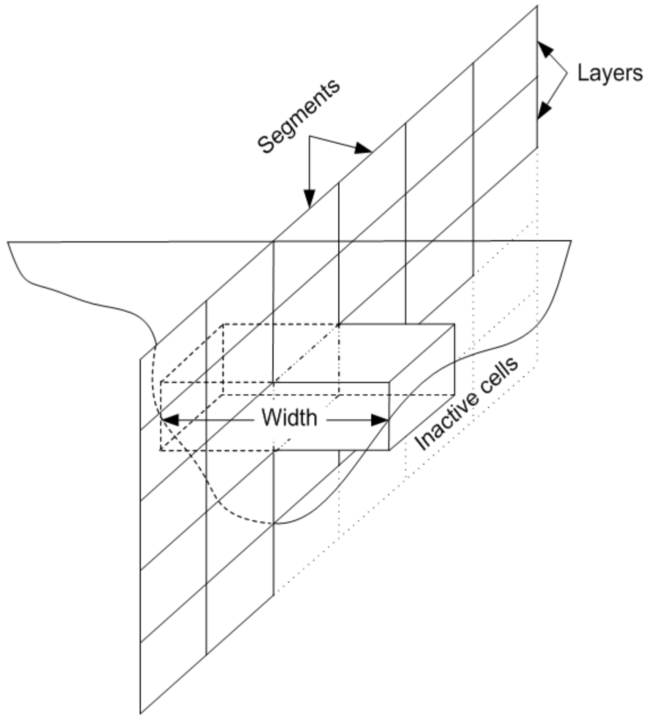


Figure 3: Finite difference grid representation of a CE-QUAL W2 model.

Average widths are generally determined from the water body's bathymetric data. This is the most difficult task in preparing the model geometry because each element width is unique (i.e. separate measurements required).

Estimating Average Widths

Widths are generally calculated from contours or sediment survey transects (cross sections) of the model bathymetry. In the case of contours a width is calculated by first determining the depth at the centroid of the segment at the layer and then measuring the distance at either side of the segment centerline to the contour with that elevation as shown in Figure 4.

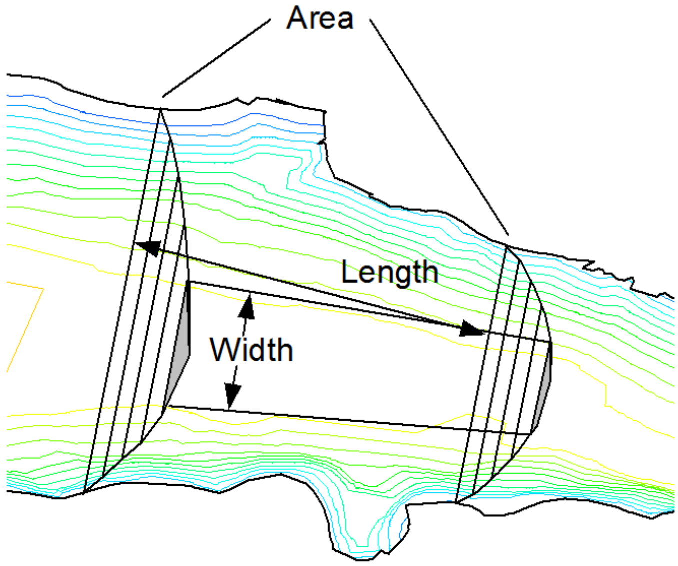


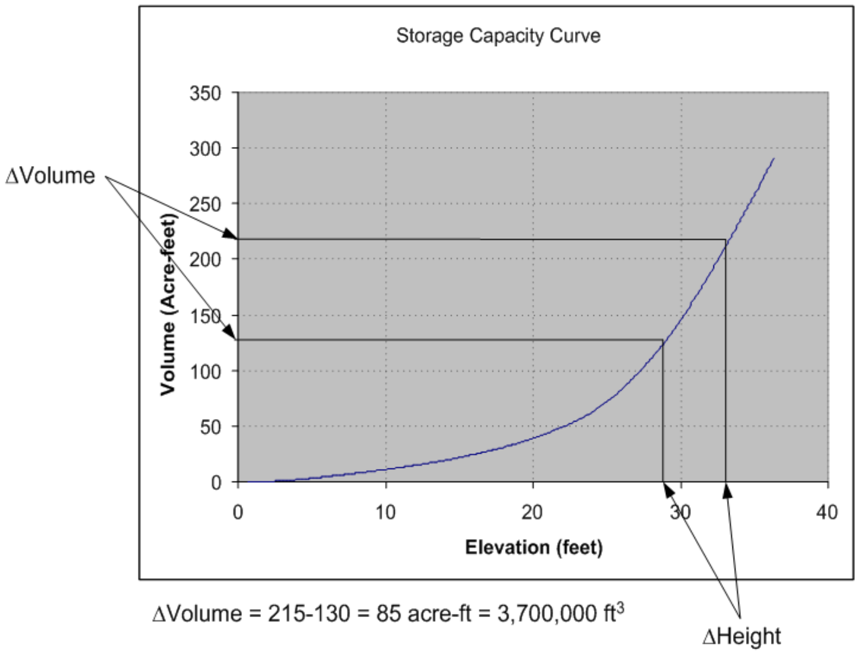
Figure 4: Measuring element widths from contours.

Alternatively, transects at both ends of the element segment can be drawn until they intersect the contour at the centroid elevation. The area bounded by these contours and the segment ends determines a width calculated by dividing the area by the segment length. Even with excellent computer aided drawing (CAD) tools this is a tedious process and has prevented widespread use of such models, even though they are superior for problems in which vertical variations are important.

Most existing reservoirs have volume-elevation curves and/or area-elevation curves (referred to throughout the rest of this page as storage capacity curves) as part of their design. Such curves allow managers to quickly determine storage volumes or surface areas for a given water surface elevation. If a storage capacity curve for each of the model segments could be generated, computing element widths at the different layers within the segment would be straightforward and easy to automate in a computer algorithm. Once a volume is determined for an element centroidal depth the average width is easily computed from the three known quantities:

$$Width = \frac{Volume}{(Length)(Height)}$$

Figure 5 illustrates how volumes and a resulting segment width are determined for any given depth.



ΔVolume = 215-130 = 85 acre-ft = 3,700,000 ft^3

ΔHeight = 33-29 = 4 ft

Length = 1480 ft (measured)

$$\text{Width} = \frac{\Delta\text{Volume}}{\text{Length} \times \Delta\text{Height}} = \frac{3{,}700{,}000}{1480 \times 4} = 625\text{ft}$$

Figure 5: Calculation of width from incremental volumes between two reservoir heights.

The key then to automating a process for developing geometric properties from bathymetric data is to develop a separate storage capacity curve for each of the model segments.

Storage Capacity Curve Generation From TIN

Triangulated Irregular Networks (TINs) are used by many programs to develop contours and compute other surface properties from scattered xyz elevation data (Lee and Schacter 1980, Watson 1981). A TIN is created by triangulating the xyz data points into a series of non-overlapping triangles which collectively form a piece-wise linear approximation of the surface. For CE-QUAL-W2 models, a TIN is used to compute a storage capacity curve for each segment.

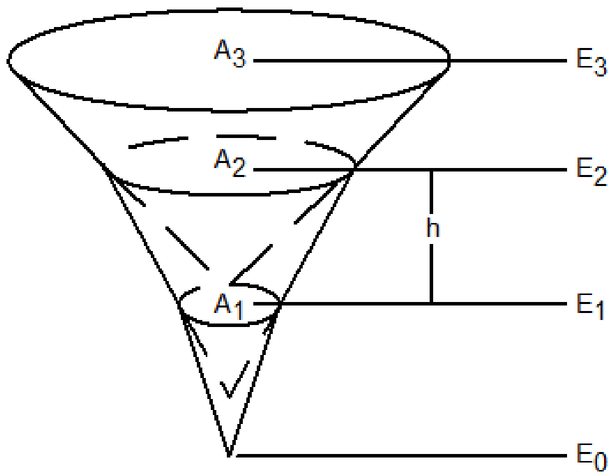


Figure 6: Conic method of computing volumes from surface areas.

Because a TIN is a piecewise linear representation of a surface, isolines can easily be computed for any given elevation. By bounding the TIN with the segment beginning and ending transects the area for any given contour elevation can be determined automatically. Any elevation interval can be chosen, but since the computations are automated with a computer algorithm a large number of areas (i.e. a small interval) for given elevations can be computed in a relatively short period of time generating a smooth elevation vs. area curve for the segment.

In order to compare with existing data typically developed during construction of a reservoir, a storage capacity curve (elevation vs. volume) is also determined using the conic method as outlined by the U.S. Army Corps of Engineers (HEC-1 1991). In the conic method incremental volumes between areas at elevations E1 and E2 (see Figure 6) are computed using the following equation:

$$\Delta V_{12} = \frac{h}{3}(A_1 + A_2 + \sqrt{A_1 A_2})$$

Once a storage capacity curve is calculated (either elevation vs. area or elevation vs. volume), the widths are determined as described earlier (see Figure 5).

Implementation Using a Conceptual Modeling Approach

The algorithms to subdivide a TIN by segments and automate the calculation of a storage capacity curve for each segment have been implemented in the Watershed Modeling System (WMS) developed by the Environmental Modeling Research Laboratory (EMRL) of Brigham Young University with the cooperation of the Army Corps of Engineers (COE) Waterways Experiment Station (WES). The WMS is a modeling tool that supports geometric pre-processing of digital terrain data for several different watershed models including the COE's CE-QUAL-W2 (W2). When developing a W2 model using the WMS, the lake, reservoir, or water body being modeled is first conceptually defined as a series of branches and segments (Smemoe et. al., 2000). An underlying digital terrain model representing the bathymetric elevations of the reservoir must be obtained. Wagner (2000) discusses several possible data sources for bathymetry elevations including: a gridded elevation matrix derived using a GPS and depth-finding device, digitization of contours from a topographic map that pre-dates the construction of the reservoir, and sediment survey transects. For newer reservoirs the USGS may have digital elevations already compiled. The bathymetric elevations must be defined for the extents of all segments in the W2 model.

The TIN representing the bathymetry of the entire water body is then subdivided into a separate TIN for each segment. A storage capacity curve for each segment is computed using the method described in the previous section.

Layer depths are user-defined based on expected gradients and available computing resources. Finally, using the segment lengths, an estimate of the width at the mid point of each layer can be computed from the storage capacity curve of the given segment.

WMS further allows the definition of all of the CE-QUAL-W2 model components (Smemoe 2000), but the focus of this page is to discuss how the bathymetry data for a W2 (or a similar laterally averaged hydrodynamic model) model can be generated from a digital terrain model. The following case study illustrates this overall process.

Case Study

The purpose of the case study is to demonstrate how the algorithm for computing average widths in a laterally averaged finite difference model was implemented in the WMS software for creating CE-QUAL-W2 input files. The reservoir modeled is the East Canyon Reservoir located in northeastern Utah (See Figure 7).

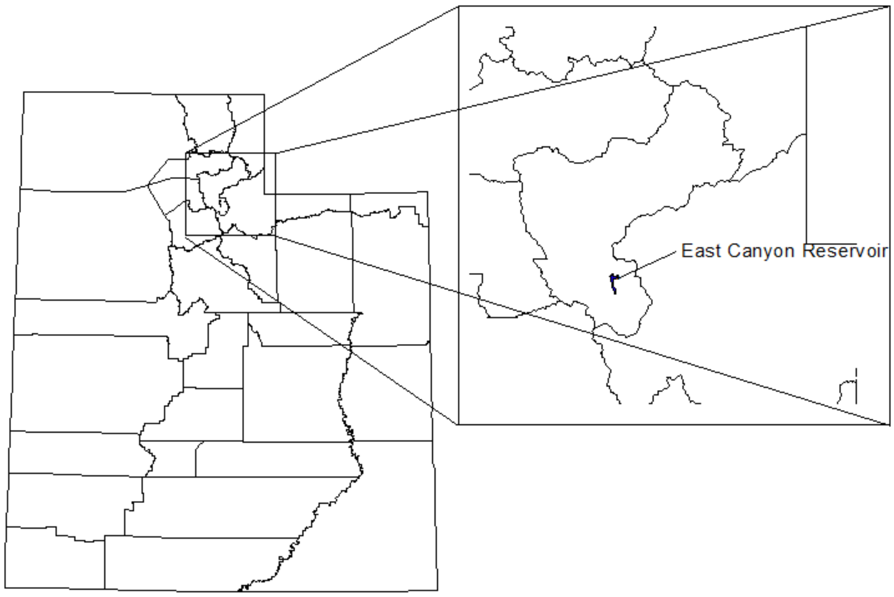


Figure 7: Location of the East Canyon Reservoir.

East Canyon is a medium sized reservoir, containing a maximum storage of 6,200 acre*ft at the maximum water surface elevation (spillway invert). It is a shallow reservoir, averaging only 8 feet. The shallow depth combined with sources of reservoir inflows has caused chronic algal bloom problems. The Bureau of Reclamation, Salt Lake City office is currently undergoing an investigation into the existing TMDL using the CE-QUAL-W2 model. The WMS software was used to generate the grid parameters (known as the bathymetry input file in CE-QUAL-W2) for this model. The following generalized steps were used in WMS for creating a bathymetry file for the East Canyon Reservoir:

- Obtain underlying digital elevation data
- Determine the model boundary
- Verify the accuracy of the digital elevation data – adjust if necessary
- Determine the reservoir branches and segments
- Compute storage capacity curves for each model segment
- Calculate average widths for segment layers

Digital Elevation Data

The underlying digital elevation data used to create the bathymetry data (Figure 8) were derived from a high-resolution x-y-z data file of the reservoir bottom topography. For the East Canyon reservoir these data were created by making several passes with a depth-finding device that automatically recorded position and elevation.

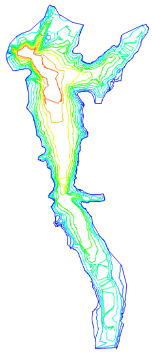


Figure 8: Bathymetric data for the East Canyon Reservoir.

Wagner (2000) discusses other possible methods for creating an underlying digital elevation file representing the topography of the bottom surface that includes digitizing a contour map and using transects from sediment surveys.

Determining the Model Boundary

For the East Canyon model the underlying digital elevation data included areas that were above the design or modeled water surface elevation. The maximum water surface elevation for the model was used to establish the model domain. In WMS this elevation can be contoured and the boundary traced to create a single polygon defining the model domain as shown in Figure 9.

Figure 9: Determining the modeling domain.

Verifying the Underlying Digital Elevation Data

Before discretizing the model into a number of segments the accuracy of the elevations used to define the reservoir bottom are compared against the known storage capacity curve of the reservoir. Using WMS, a single storage capacity curve is derived from the bounding polygon created in the previous step and the underlying elevation data. This curve is then compared against the storage capacity curve for East Canyon developed from original surveys performed during the design and construction of the reservoir. The comparison is shown in the plot of Figure 10.

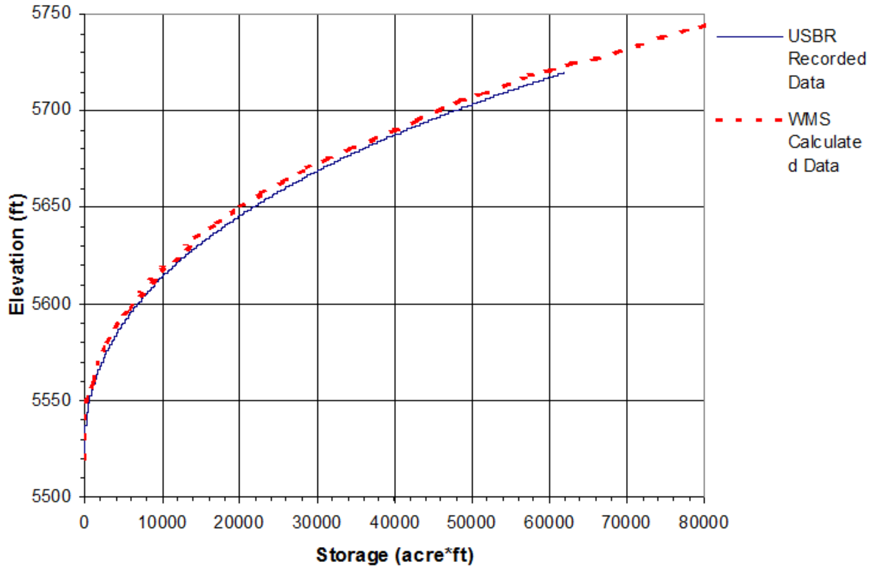


Figure 10: Comparison of original storage capacity curve with one computed by WMS.

Even at the maximum water surface elevation the curves vary by less than 5% and so no further modifications were required. When using data sources such as sediment survey transects or less detailed contour maps it may be necessary to adjust the bathymetry by scaling as described in Wagner (2000).

Determining Reservoir Branches and Segments

A CE-QUAL-W2 model is defined from a number of branches that are further subdivided into segments. The East Canyon reservoir was modeled as two branches as shown in Figure 11.

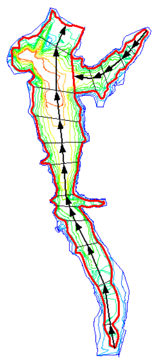


Figure 11: East Canyon conceptual model of branches and segments.

The primary branch was further subdivided into 12 segments and the tributary branch into 5 segments. In a CE-QUAL-W2 model the segments define the longitudinal dimension of the resulting finite difference grid.

Compute Storage Capacity Curves for Model Segments

In order to compute average widths for layers within a segment, a storage capacity curve for each segment must be derived. WMS does this as described earlier where the distance between elevations used to calculate the storage capacity curve is .1 unit (feet or meters depending on the units of the underlying digital elevation data). This establishes a smooth curve from the lowest elevation within the segment to the maximum water surface elevation and can be used to establish volumes between any two elevations as illustrated in Figure 5.

Calculate Average Widths for Segment Layers

The depth or layer dimension of the grid is established by the user and generally depends on the overall depth of the reservoir, gradients within the reservoir, and other model objective criteria. A layer depth of approximately 6 meters was used to construct the grid for this case study. The final grid dimension parameter required is the average width of each grid cell. The storage capacity curve for each segment was used to compute average widths using the storage capacity curve to determine the volumes between consecutive layer depth elevations and dividing by the product of the segment length and layer depth. A complete bathymetric description (lengths, depths and widths of all segments and layers) was developed for the full CE-QUAL-W2 model that is now being used by the Bureau of Reclamation in their TMDL analysis of the entire watershed.

Conclusion

CE-QUAL-W2 is a useful model for determining variations in temperature, dissolved oxygen, and other water quality parameters in reservoirs and other water bodies. Normally, most of the time spent in creating a CE-QUAL-W2 model is spent in building the laterally averaged finite difference grid and preparing the input files. The approach outlined in this page describes a method for estimating cell widths that makes the development of the grid manageable. The process is general enough to be used on any similar class of laterally averaged numerical models.

Acknowledgements

Support from the Bureau of Reclamation and Army Corps of Engineers for this research is gratefully acknowledged. Insights gained through personal consultations with Tom Cole of the Waterways Experiment Station and Jerry Miller of the Salt Lake City Bureau of Reclamation office were instrumental in the development of this procedure.

References

- Nelson, E.J., N.L. Jones, and A.W. Miller, March 1994 "An algorithm for precise drainage basin delineation," ASCE Journal of Hydraulic Engineering, pp. 298-312.
- Cole, T., “CE-QUAL-W2 Reference Manual,” Waterways Experiment Station, Vicksburg, Mississippi, 1995.
- Aquaveo, “WMS 9.1 User Manual,” 2013.
- Fugal, A.L., (2000). Two-Dimensional Finite Element Density Meshing, Masters Thesis, Brigham Young University, Provo, Utah.
- Gaspar, C., J. Jozsa, and J. Sarkkula, “Shallow Lake Modeling Using Quadtree-Based Grids,” International Conference on Computational Methods in Water Resources, vol 10, pp. 1053-1060, Heidelberg, 1994.
- Lee, D. T., and Schacter, B. J., 1980, "Two algorithms for constructing a Delauney triangulation," International Journal of Computer and Information Sciences, Vol. 9, No. 3 pp. 219-242.
- Smemoe, Christopher M., E. James Nelson, and Tom Cole, “A Conceptual Modeling Approach to CE-QUAL-W2 Using the Watershed Modeling System,” Proceedings of the Hydroinformatics Conference, Iowa City, Iowa, July 2000.
- Thibodeaux, K.G., “Mesh-Generating Computer Program for the FESWMS-2DH Surface-Water Flow Model,” Hydraulic Engineering Water Forum, ASCE, 1992.
- Wagner, Jason G., (2001). “Bathymetry Generation and Documentation for Water Quality Modeling”, Masters Project, Brigham Young University, Provo, Utah, 2001.
- Watson, D.F., 1981, "Computing the n-dimensional Delauney tessellation with application to Voronoi polytopes," The Computer Journal, Vol. 8, No. 2, pp. 167-172.

6.2. Gridded Surface Subsurface Hydrologic Analysis (GSSHA)

GSSHA

GSSHA is a two-dimensional finite difference rainfall/runoff model. A finite difference grid is used to establish the computational domain and parameters for surface runoff. The GSSHA model is fully coupled with hydraulic stream flow/routing models. Parameters for stream channels are defined using arcs and then mapped to the appropriate underlying grid cells.

A detailed online reference manual for GSSHA is found here:

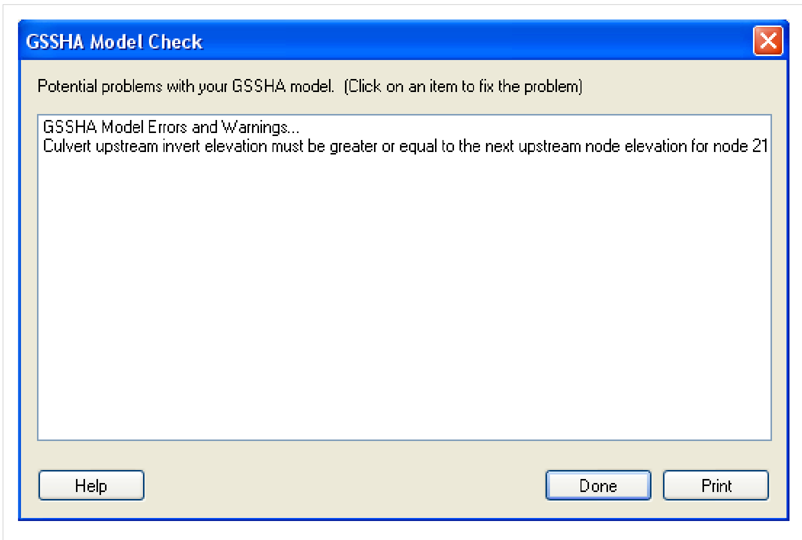
http://www.gsshawiki.com [1]

Open Project File

The **Open Project File** command in the *GSSHA* menu looks for a GSSHA project file. A GSSHA project file begins with the first line either being GSSHAPROJECT or CASC2DPROJECT. Once a valid project file has been identified, WMS will read in all of the files needed for input to GSSHA. The channel input file is not read in but instead the channel input information is re-created from the WMS map file that stores the GSSHA coverage. GSSHA output files are not read in by the **Open Project File** command but instead are read by the **Read Solution** command.

Model Check

The GSSHA **Model Check** command runs through the input data for a GSSHA simulation and looks for obvious inconsistencies and problems with the model. The model checker will not identify values that are outside of plausible ranges, etc., only logical problems. For example, an index map has one or more cells whose ID is 0 (which is not allowed in GSSHA), or a specified process that has an ID with parameters whose values are 0.00 (which indicates an incomplete entry). An example of a problem that the model checker would not find is if a stream arc has a pit that pools water.

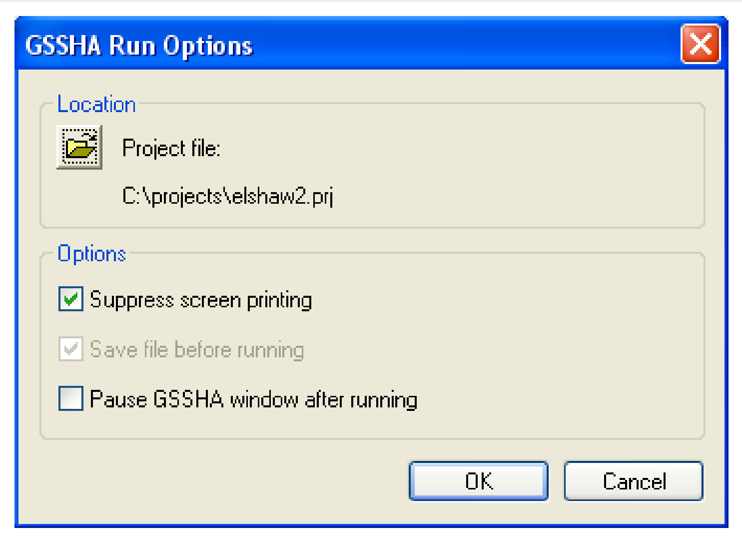


Run Simulation

The **Run GSSHA** command launches GSSHA in the WMS Model Wrapper. Select an existing GSSHA project file or specify the name and location of the GSSHA project file to be saved before running the simulation. The *GSSHA Run Options* include:

- Suppress screen printing
- Save files before running
- Pause GSSHA window after running

The *Suppress screen printing* option suppresses the output of runtime data from GSSHA at each time step, but significantly reduces the overall run time required for a GSSHA simulation. This option also requires that the GSSHA input files be saved before running the simulation.



WMS will automatically read the GSSHA solution produced by the simulation when the **Close** button is clicked in the WMS Model Wrapper if the *Read Solution on Exit* option is toggled on.

Note: GSSHA currently supports file names (including path and file name) with lengths less than or equal to 128 characters. If GSSHA tutorial files are placed in a directory with a path that causes the combined path and file name to exceed 128 characters, then GSSHA will not run.

Save Project File

Specify the GSSHA project file name and location in the *Save GSSHA Project File* dialog. Select *Save* to save the GSSHA project file and all other GSSHA input files. WMS automatically uses the GSSHA project file name as a prefix for all GSSHA input files (file extensions are used to differentiate between GSSHA input files), except for the index map files, which are assigned default names using the convention id_map_##.idx, where ## is generated by WMS. All files are written to the same directory according to the specified location.

For long term simulations WMS copies the HMET file from its original location to the new GSSHA project directory.

It is also possible to manage all of the GSSHA input and output files names using the *Manage files* option in the *GSSHA Job Control Parameters* dialog.

Cell Properties

The 2D grid cell properties include an I, J location and a scalar or index map value. The scalar or index map value is associated with the currently selected dataset in the *Project Explorer Window* and can be edited in the *Properties Window*. The I, J location can be viewed but cannot be edited in the *Properties Window*. The 2D grid contours are a way of visualizing the spatial variation of the scalar or index map values for the selected dataset.

Related Topics

- GSSHA Calibration
- GSSHA Channel Routing
- GSSHA Contaminants
- GSSHA Digital Dams
- GSSHA Embankment Arcs
- GSSHA Groundwater
- GSSHA Maps
- GSSHA Job Control
- GSSHA Join SSURGO Data
- GSSHA Mapping Tables
- GSSHA Maps
- GSSHA Multiple Scenarios
- GSSHA Nutrients
- GSSHA Overland Soil Erosion
- GSSHA Output Control
- GSSHA Precipitation
- GSSHA Read Solution
- GSSHA Solution Analysis

GSSHAWiki [2]

Primer [3]

- Overview [4]

User's Manual [5]

- Introduction [6]

Tutorials [7]

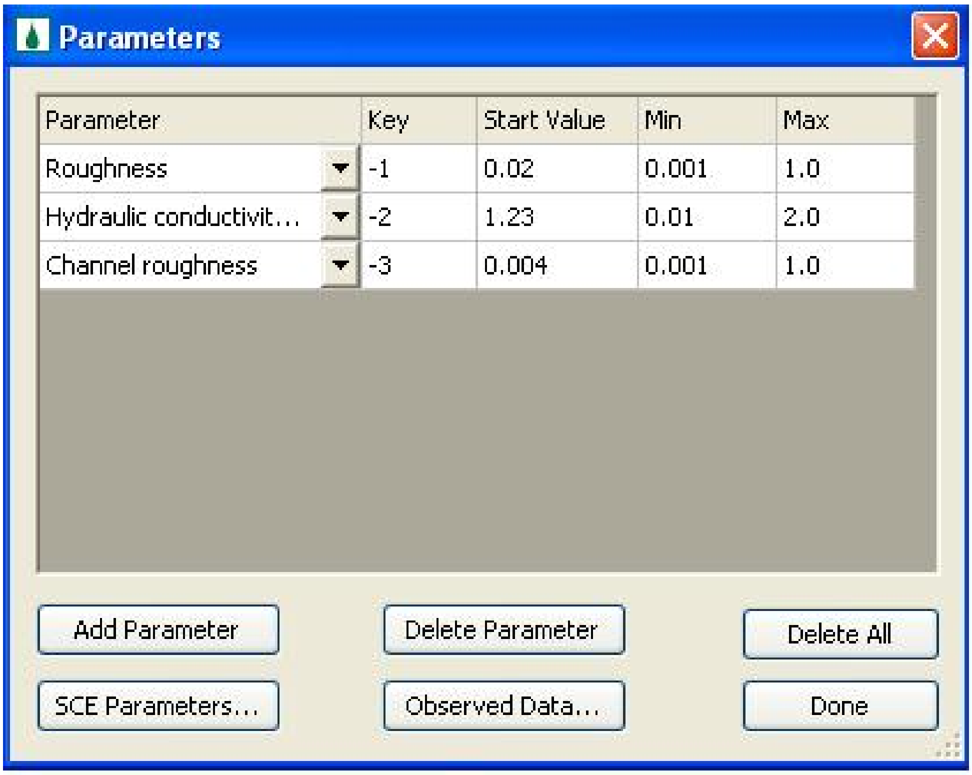
References

[1] http://www.gsshawiki.com
[2] http://gsshawiki.com/index.php?title=Main_Page
[3] http://gsshawiki.com/index.php?title=GSSHA_Software_Primer
[4] http://gsshawiki.com/index.php?title=Overview:Overview
[5] http://gsshawiki.com/gssha/GSSHA_User%27s_Manual
[6] http://gsshawiki.com/index.php?title=Introduction:Introduction
[7] http://gsshawiki.com/gssha/Tutorials:Tutorials

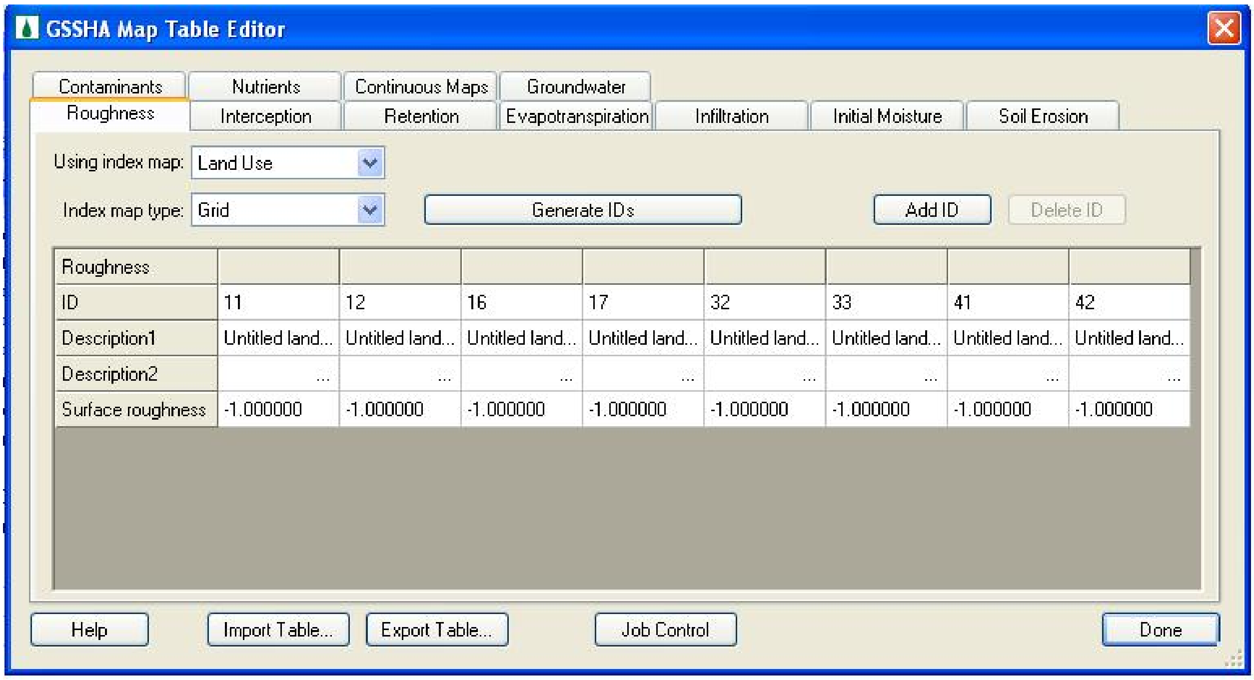
GSSHA Calibration

Auto Calibration

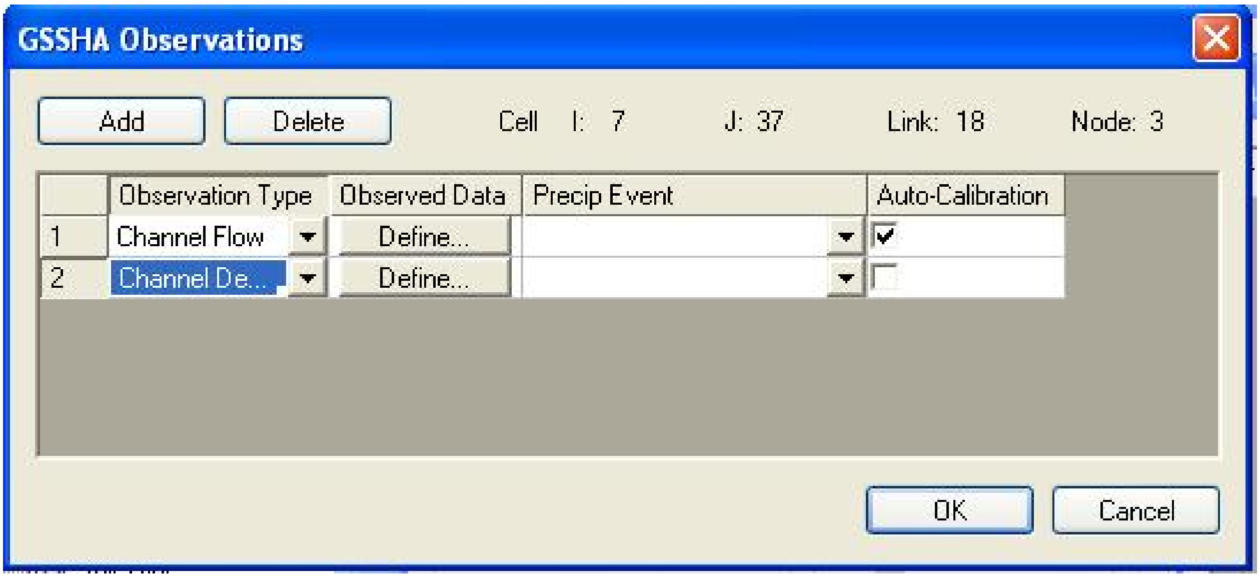
WMS writes the parameters, observed data, and calibration files necessary to run GSSHA in shuffled complex evolution (SCE) calibration mode. You can enter in the needed data by toggling **Calibration** in the *job control* dialog. Clicking on the **Edit Parameters** button allows you to select which parameters are to be calibrated. The dialog shown below appears, in which you can enter the maximum, minimum, and initial values for each parameter.



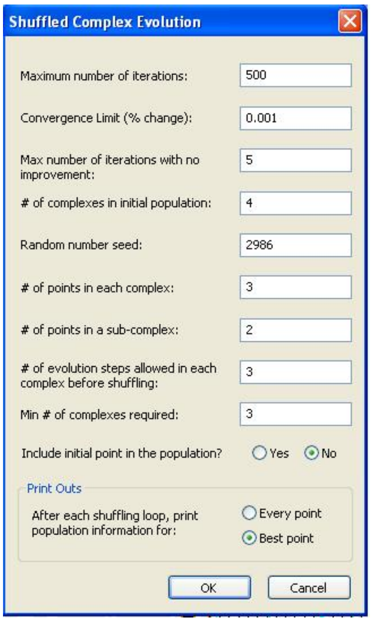
The key for each parameter should be entered in the mapping tables or properties where you desire to calibrate. An example of calibrating the roughness values using a key of -1 is shown below.



Clicking the **Observed Data...** button in the *calibration parameters* dialog opens another dialog, which is shown below. In this dialog you can enter in various types of observed data for each event of the simulation.



Lastly, you need to set the SCE parameters, which can be accessed by clicking the **SCE Parameters...** button. The following dialog will appear:



The parameters in this dialog are used for the calibration input file. This file, along with the parameters and observed data files are written out by WMS when you save the project.

Manual Calibration

Manual calibration is the process of changing simulation input so that the simulation output matches observed values. Manual calibration takes a lot of experience and a lot of patience, but it is possible to achieve a good fit between the simulation and the observed data. Being able to successfully manually calibrate a simulation is a necessary skill to successfully set up and run an automatic calibration program.

Steps in the Manual Calibration Process

1. Set up and run a successful simulation.
2. Identify the calibration variables.
3. Decide on a valid range for each variable.
4. Set initial values of variables and run the model.
5. Compare model results to observed values.
6. Change variable values and re-run.
7. Repeat steps 5 and 6 until the simulation results closely approximate the observed values.

1. Set up and run a successful simulation

The first step to calibrating a simulation is to set up and successfully run a reasonable simulation. All known parameters, such as precipitation, should be defined; all unknown parameters should be set to physically realistic values. For example, if actual roughness values are unknown then set all of the roughness values to 0.035 or some other reasonable number. Setting the roughness values to 0.00 or some default value will not allow the simulation to proceed. One important consideration is that if the spatial or temporal resolution is too coarse than the simulation will be unduly influenced by numerical issues related to the implementation of the partial differential equations. The result of too coarse of a temporal or spatial resolution will be delayed flows. For more information, see the Primer: Using Watershed Modeling System (WMS) for Gridded Surface Subsurface Hydrologic Analysis (GSSHA) Data Development – WMS 6.1 and GSSHA 1.43c (Downer et. al 2003).

2. Identify the calibration variables

The calibration variables are the simulation parameters whose exact quantities are unknown. This may range from a small handful to several dozen. At this stage it is also often necessary to identify which parameters the simulation is sensitive to and which can be left at good approximations without unduly affecting the model. The number of calibration variables must be pared down to a manageable number as well. Attempting to manually calibrate a simulation with dozens of unknown parameters will lead to one major headache and not to a good, robust simulation. Calibration cannot overcome a general lack of data.

Occasionally lab tests for such parameters as hydraulic conductivity will be available. Such data is very valuable but it still may be necessary to calibrate on that specific parameter because a simulation parameter represents a uniform parameter over an area while lab results generate the parameter for a specific point. The lab data is a very good starting value but may need some modification before it is applicable to a general area.

3. Decide on a valid range for each variable

Knowing a range for each variable is very important. To accurately simulate what is actually present in the watershed requires knowledge of the physical meaning of all of the numerical parameters for the watershed. Without this understanding a simulation that does not accurately reflect reality will be created and the simulation will be worthless in a predictive capacity. Consulting published works that describe the formulas used in GSSHA and detail the values and physical meaning of the formula parameters is highly recommended.

4. Set initial values of variables and run the model

Once the calibration variables have been decided upon and the valid range for each has been identified, the next step is to set an initial value for each variable. The usual process is to begin with the middle value. Later on these values will be modified little by little, either up or down. Beginning with the middle value of the range gives a good reference point for later simulations where what happens with a higher or lower value can be judged against the middle value to determine simulation trends.

5. Compare model results to observed values

This is the key step to calibrating a simulation. Click on the button in the Solution Results column of the Feature Point/Node Properties dialog to display the Solution Analysis dialog, which allows both visual inspection of the solution result as well as numerical evaluation of the "fitness" of a solution. Using these criteria judge how well the simulation output fits the observed.

6. Change variables and re-run

If the simulation output is not sufficiently close to the observed data then the next step is to adjust one or more of the model parameters to try to get a better fit. This step takes practice, experience, and patience. If by adjusting the variables outside of the predefined range a better fit is obtained then either the simulation is poorly set up or the data on which the model is based may be in question. It may also be that the interdependence of variables is such that the other variables in the model should be adjusted before the one that seems to call into question the parameter bounds. After adjusting the variables and running the simulation, check the new output and judge the results of the new variable setting.

Simulation non-uniqueness

One important facet of calibrating a simulation is that often changing more than one variable can have very similar results on the simulation output. Calibrating a simulation attempts to extract spatial simulation parameters from observed data through a process called inverse modeling. Problems arise in calibrating when modifying more than one variable produces only one type of result in the simulation. The question then arises as to which variable values should be the actual variable values. This problem is not solvable and the simulation is said to be non-unique or over-specified. The only way to overcome this problem is by utilizing more data that is of a different type than that already being used. For example, using a stream-flow hydrograph as well as a set of observed groundwater elevations would help eliminate simulation non-uniqueness.

Related Topic

- GSSHA Observation Points
- GSSHA Overview

GSSHAWiki [2]

Primer [3]

- Watershed Delinieation and Grid Construction [1]

User's Manual [5]

- Building a GSSHA Model [2]

Tutorials [7]

- 11 Manual Calibration [3]

References

[1] http://gsshawiki.com/index.php?title=Pre-Processing:Watershed_Delineation_and_Grid_Construction
[2] http://gsshawiki.com/index.php?title=Building_a_Model:Building_a_GSSHA_Model
[3] http://gsshawiki.com/index.php?title=Tutorials:11_Manual_Calibration

GSSHA Channel Routing

GSSHA is a two-dimensional finite difference rainfall/runoff model. A finite difference grid is used to establish the computational domain and parameters for surface runoff. The GSSHA model is fully coupled with hydraulic stream flow/routing models. Parameters for stream channels are defined using arcs and then mapped to the appropriate underlying grid cells.

In order to define GSSHA channel parameters using arcs, the current coverage type must be set to GSSHA.

Smoothing Stream Cells

Because elevation data used to define the surface runoff component of GSSHA does not contain the detailed resolution required to capture the actual stream bed elevation of the channels, the bed elevation profile can be highly irregular. These irregularities, or abrupt changes in elevation can cause instabilities in the channel routing computations, and therefore must be smoothed out.

The **Smooth Stream Arcs** command is used to adjust the elevation of the stream bed for GSSHA. Smoothing is done by first selecting a continuous set of arcs which represent the stream and then choosing the **Smooth Stream Arcs** command from the *GSSHA* menu.

Initially the bed elevation is assigned the same value as the grid. Whenever a profile is shown, the bed elevation profile is displayed in blue while the grid elevation of the cell is shown in red. Care should be taken to see that no bed elevation is higher than the grid cell elevation.

Renumber Links And Nodes

In order to properly execute the channel routing routines of GSSHA, the stream channel must have the proper order and connectivity. This ordering or numbering can be done automatically using the **Renumber Links and Nodes** command from the *GSSHA* menu.

Links define whole channel segments and must be numbered such that any segment has no "upstream" segments with a link number that is greater than itself. In other words all channel segments must "flow" into downstream segments with a higher link number.

Related Topics

- GSSHA Overview
- Grids

GSSHAWiki [2]

Primer [3]

- Overview [4]

User's Manual [5]

- Preface [1]
- Introduction [6]

Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Introduction:Preface

GSSHA Contaminants

To model contaminants in GSSHA the contaminant transport option in the *Job Control* dialog must be turned on.

Contaminants are set up in three steps:

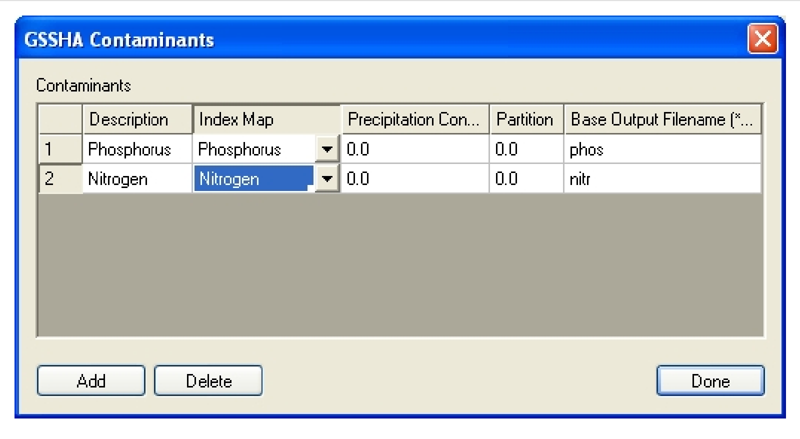
1. Set up the contaminant types.
2. Create the index maps for the contaminants.
3. Set up the contaminant parameters in the mapping table.

Set up the contaminant types

Contaminants are defined in the *GSSHA Contaminants* dialog, which is accessed either in the *GSSHA* menu or from the *Job Control* dialog. Click on **Add** to add new contaminants. Five parameters must be defined for each contaminant:

1. A description of the contaminant.
2. An index map showing the spatial distribution of the contaminant.
3. The GSSHA output base filename for that contaminant.
4. Concentration of the contaminant in rainfall.
5. Soil water partition coefficient.

Note that the index map for each contaminant may be specified in the *GSSHA Contaminants* dialog or the *GSSHA Index Map Table Editor* dialog.



Create the index maps for the contaminants

The next step is to create index maps for the contaminants. These index maps are created in the same manner as other index maps and will usually be a cross between a soil type and a contaminant distribution coverage. However, there is no "Contaminant Distribution" coverage type in WMS. Simply create either a land use or soil type coverage and create polygons representing the different contaminant plumes. Different polygons for different initial concentrations of the contaminants will be useful later on as the specified contaminant concentration level for each index map ID (polygon) is a uniformly distributed concentration over the polygon. Also, in the coverage representing contaminants the entire grid must be covered by a polygon. Outside the contaminant plumes, the concentration will be zero.

When creating the index map from the coverages, a cross between a soil type and the contaminant distribution coverage will prove to meet the needs of the contaminant transport routines. The parameters that must be specified for each index map ID are a dispersion rate of the contaminant in standing water, a decay rate for the contaminant (first order kinetics), a cell uptake rate, and an initial mass loading. The contaminant distribution coverage should be used to define the initial mass loading and the soil type coverage should be used to define the cell uptake rate. The dispersion and decay rates will typically be uniform throughout the grid.

Set up the contaminant parameters in the mapping table

Finally, go to the GSSHA Index Map Table Editor. Choose one of the contaminants that have been set up and associate the correct index map with it. Follow the usual steps for setting up an index map and initialize each of the parameters for each ID.

Contaminant Parameters

- Dispersion rate ($m^2 s^{-1}$)
- Decay rate (s^{-1})
- Cell uptake coefficient
- Initial mass loading (kg)
- Groundwater concentration (mg/l)
- Initial concentration (mg/l)
- Soil water distribution coefficient
- Max concentration/solubility (mg/l)

Related Topics

- Overland Soil Erosion
- GSSHA Job Control
- GSSHA Overview
- GSSHA Maps

GSSHAWiki [2]

Primer [3]

- Mapping [1]

User's Manual [5]

- Mapping Table File [2]
  - Index Maps [3]
  - Mapping Tables [4]

Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Mapping:Assigning_Parameter_Values_to_Individual_Grid_Cells
[2] http://gsshawiki.com/index.php?title=Mapping_Table:Mapping_Table_File
[3] http://gsshawiki.com/index.php?title=Mapping_Table:Index_Maps
[4] http://gsshawiki.com/index.php?title=Mapping_Table:Mapping_Tables

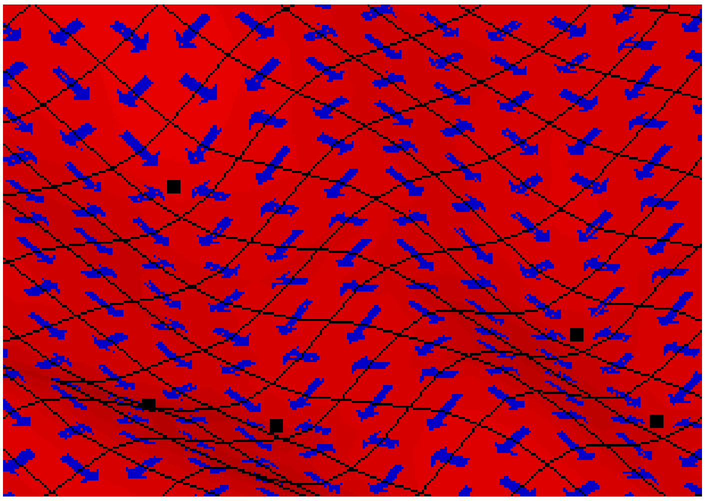
GSSHA Digital Dams

Digital dams occur where a stream leaves a cell at or near a corner. Since flow occurs between adjacent cells and not between diagonal cells, the stream leaving at the cell corner leaves behind two stream banks adjacent to the flowing cell. These two stream banks block the overland flow, which can create numerical instabilities and will pool up the flowing water behind the banks. WMS now has several ways to visualize and adjust digital dams in the overland flow plane.

Visualization

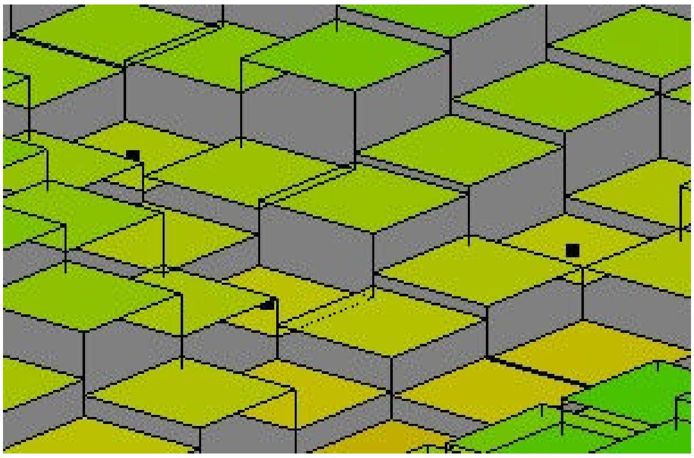
WMS now has two ways to visualize digital dams. These options are accessible on the *2D Grid* tab of the *Display Options* dialog. Toggle on the *Digital Dams* option to have WMS identify digital dams is by looking for cells that are lower than their four neighbors. The cells that meet this criterion will be identified with a small square marker in the middle of the cell.

The second method of visualization is employed by toggling on the *Flow Vectors* option in the *2D Grid* tab of the *Display Options* dialog. WMS displays flow direction arrows between cells based only on the differences in cell elevation. GSSHA computes overland flow based on the hydraulic grade surface, but this is often approximately equal to the elevation surface. A digital dam occurs when all four flow direction arrows indicate flow only coming into cell and not leaving the cell.



A grid is shown with flow vectors (blue arrow) and digital dam markers (black squares). The flow vectors show the direction of flow between adjacent cells. The digital dam markers indicate cells where all four neighboring cells are higher in elevation.

A method to help visualize the computational flow surface where the digital dams are is to turn on the *Blocked Cells* option in the *2D Grid display options*. Blocked cells treat each grid as a planar surface and draw them as such. A vertical line is drawn connecting the cell corners. Blocked cells are often used in combination with cell-filled contours.



The cells can be drawn in a block style that affords greater visualization of the computational surface. The black squares indicate cells where digital dams exist. The cells in this diagram are also cell filled, meaning that the cell is colored in one color based on the elevation of the center of the cell.

Modification

There are two ways in WMS of fixing digital dams.

The first method is to run the CleanDam algorithm found in the **GSSHA | Clean Digital Dams...** menu command. CleanDam slightly alters the cell elevations in order to fix the digital dams. A summary of the CleanDam results can be viewed before closing the *WMS Model Wrapper* dialog. WMS automatically reads in the new elevations and replaces the existing elevation dataset.

You can define a *Depression mask* polygon in a GSSHA coverage that masks cells contained within the polygon. The elevations of the cells located inside a depression mask polygon are not altered when the CleanDam program is run from WMS. There is a display option to display the 2D grid cells located within depression mask polygons.

The other method of correcting any remaining digital dams is to manually adjust the cell elevations. Sometimes changing the grid resolution can help initially eliminate digital dams.

Related Topics

- GSSHA Overview
- 2D Grid Display Options

GSSHAWiki [2]

Primer [3]

- Watershed Delinieation and Grid Construction [1]
  - Editing the Grid to Correct Elevation Errors [1]

User's Manual [5]

- General Considerations [2]
  - Elevation Map [3]]

Tutorials [7]

- Fixing Digital Dams [4]

References

[1] http://gsshawiki.com/index.php?title=Pre-Processing:Editing_the_grid_to_correct_elevation_errors
[2] http://gsshawiki.com/index.php?title=General_Considerations:General_Considerations
[3] http://gsshawiki.com/index.php?title=General_Considerations:Elevation_Map
[4] http://gsshawiki.com/index.php?title=Tutorials:3_Fixing_Digital_Dams

GSSHA Embankment Arcs

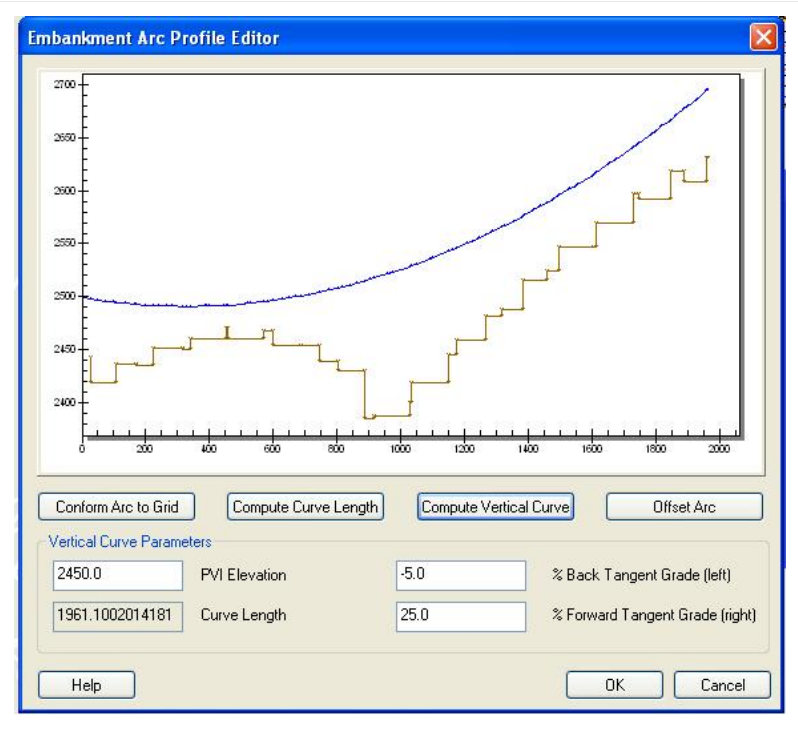
Embankment Arcs in WMS

In WMS, man-made embankments are represented as embankment arcs. These embankment arcs are placed along the centerline of the actual embankment. Often these embankment arcs represent roads that cross a drainage. The embankment arc can intersect a stream arc and an in-stream hydraulic structure (a culvert) placed at the intersection to allow water to pass through. For the embankment arc to function correctly in GSSHA, the elevations along the top of the arc must be defined. This is done through the *Embankment Arc Profile Editor*.

In the *Embankment Arc Profile Editor* the arc elevations can be generated by entering the parameters for a vertical curve. WMS will then take the given parameters and compute the correct elevation values for each vertex of the arc. The parameters needed for the vertical curve are the elevation of the point of vertical intercept, the left (back) tangent slope, the right (forward) tangent slope, and the curve length. Note that the arc need not be a straight line in plan view for the vertical curve to be computed.

The arc elevations can also be set by using the *Conform Arc to Grid* option, which sets the elevations of the arc to the corresponding grid elevations. The *Offset Arc* option will change all arc elevations by a constant specified by the user.

The *Embankment Arc Profile Editor*, shown in the image below, is accessible from the *Feature Arc Attributes* dialog in the GSSHA coverage. The feature arc type in order to access the dialog.



The *Embankment Arc Profile Editor* allows the embankment profile to be manually adjusted or set as a computed vertical curve.

Embankment Arcs in GSSHA

In WMS, the embankments are represented as feature arcs and are given profile elevations that define the top of the embankment. These feature arcs are then mapped to the nearest cell edge. The embankment acts as a wall all along the edge, interrupting flow between the two adjacent cells.

The wall elevation at each cell is determined by interpolation along the embankment arc. Once the water elevation has risen to this elevation, water will commence flowing over the embankment in a horizontal broad-crested weir fashion.

Related Topics

- GSSHA Feature Nodes

GSSHAWiki [2]

Primer [3]

- Watershed Delinieation and Grid Construction [1]
- Lake and Channel Routing [1]

User's Manual [5]

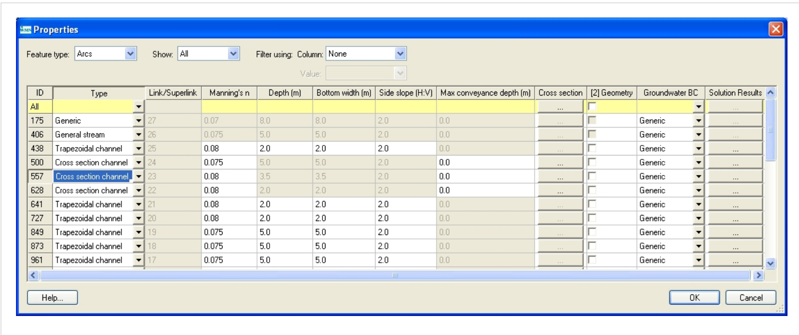
- Surface Water Routing [2]

Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Routing:Lake_and_channel_routing
[2] http://gsshawiki.com/index.php?title=Surface_Water_Routing:Surface_Water_Routing

GSSHA Feature Arcs



The *Coverage Properties* dialog is where the attributes associated with feature arcs on the GSSHA coverage are defined. This dialog is accessed either by double-clicking on a feature arc or by selecting the **Feature Objects | Attributes...** menu command in the Map Module. It is possible to change the feature type attributes that are displayed in the table, select whether to show attributes for all features or just the selected features, and to filter the data shown in the table based on specific attribute values.

GSSHA Arc Types

Generic

Generic arcs have no attributes and are typically used when constructing polygons.

Parameters: None

General Stream

General stream arcs are identical to streams defined for drainage coverages and are used when going back and forth between coverage types. General stream arcs are not used to generate input for GSSHA simulations.

Parameters: None

Trapezoidal Channel

Trapezoidal cross-sections are used to define channel routing in GSSHA models. WMS automatically assigns a link number to trapezoidal channel arcs. By default the geometric parameters of the trapezoidal channel are applied to the entire channel (link), although it is possible to assign both upstream and downstream geometric parameters for the channel (link) by toggling on the [2] Geometry option. GSSHA will interpolate a cross section at each node in the link using the upstream and downstream channel geometries.

Parameters: Manning's n value, channel depth, bottom width, and side slope Other: Enter the maximum depth of channel erosion if the soil erosion option is turned on in the Job Control

Cross Section Channel

The profile of irregular cross sections are defined using X, Y coordinate pairs. GSSHA computes conveyance parameters including area, top width, and conveyance at incremental depths of flow up to the maximum depth specified in the Max conveyance depth column of the table.

Parameters: Manning's n value, cross section profile, and maximum conveyance depth Other: Enter the maximum depth of channel erosion if the soil erosion option is turned on in the Job Control

Embankment

Embankments represent overland flow hydraulic structures such as levees or roads. They modify flow by either preventing flow between adjacent cells or acting as a weir if the flow reaches above the crest elevation of the embankment.

Parameters: embankment profile

Pipe

Pipe arcs are used to conceptualize sub-surface storm/tile drain systems. Looped configurations are allowed.

Parameters: type (circular or rectangular), geometric properties, slope, Manning's n value, length, conductance (tile drains only), and superlink number

Sub-surface Losses/Gains

If the *Groundwater (sub-surface)* option in the *GSSHA Job Control* is turned on then it is possible to toggle on the option to compute sub-surface losses/gains by entering sediment thickness and hydraulic conductvity.

Groundwater BC

Assign one of the following groundwater boundary conditions to the 2D grid cells intersecting the feature arc: Generic, No Flow, Constant Head, Flux River, or Head River.

Solution Results

Click on the button to view results at all node locations for the link in the stream/channel network.

Related Topics

- GSSHA Embankment Arcs
- GSSHA Feature Nodes
- GSSHA Feature Polygons
- GSSHA Groundwater
- GSSHA Job Control
- GSSHA Solution Results
- GSSHA Stream Arcs

GSSHAWiki [2]

Primer [3]

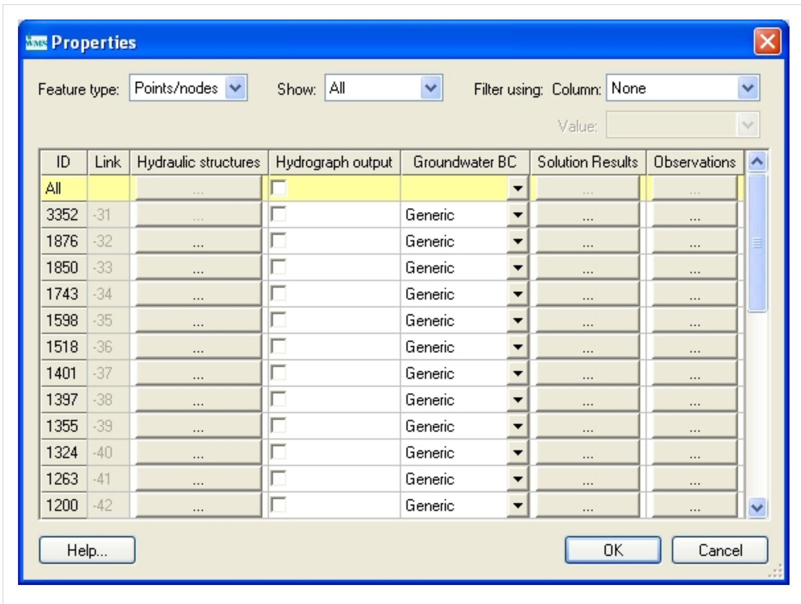
- Watershed Delinieation and Grid Construction [1]
- Lake and Channel Routing [1]

User's Manual [5]

- Surface Water Routing [2]

Tutorials [7]

GSSHA Feature Nodes



The *Coverage Properties* dialog is where the attributes associated with feature points/nodes on the GSSHA coverage are defined. WMS automatically assigns the correct type (generic, link break, or junction) to each point/node based on the type of feature arcs connected to the node, although the point/node type is not displayed in this dialog. It is possible to change the feature type attributes that are displayed in the table, select whether to show attributes for all features or just the selected features, and to filter the data shown in the table based on specific attribute values.

GSSHA Feature Node Properties

Link

The link number is assigned by WMS to nodes that have hydraulic structures defined.

Hydraulic Structures

Click on the **Hydraulic Structures** button to define hydraulic structures which exist in the GSSHA stream network. Link-break type nodes on GSSHA stream arcs (trapezoidal or cross-section) are used for defining known points of control and modification of the stream flow by in-stream hydraulic structures such as weirs and culverts. Hydraulic structures are represented as a separate link in the stream network in GSSHA. As such, if a feature node does not exist where a hydraulic structure exists, a feature node (with the link-break type) must be placed there.

There are three main types of hydraulic structures: weirs, culverts, and implicit defined structures. Any number of weirs, culverts, and/or curves may be specified together, but with only one instance of each curve type allowed.

Weirs

- Horizontal Broad-Crested
  - Parameters for a Horizontal Broad-crested Weir: crest length, discharge coefficient (forward flow), discharge coefficient (reverse flow), crest low point elevation
- Infinite Sag Vertical Curve
  - Parameters for a Infinite Sag Vertical Curve: left slope, right slope, discharge coefficient (forward flow), discharge coefficient (reverse flow), crest low point elevation

Culverts

- Round
  - Parameters for Round Culverts: diameter, upstream invert, downstream invert, inlet loss coefficient, reverse flow inlet loss coefficient, slope, length, Manning's n value
- Oval
  - Parameters for Oval Culverts: axis width, axis height, upstream invert, downstream invert, inlet loss coefficient, reverse flow inlet loss coefficient, slope, length, Manning's n value
- Rectangular
  - Parameters for Rectangular Culverts: box width, box height, upstream invert, downstream invert, inlet loss coefficient, reverse flow inlet loss coefficient, slope, length, Manning's n value

Hydraulic Structure Curves

A hydraulic structure can be implicitly represented by either a rating curve, rule curve, or scheduled release curve. These curves will usually be used in conjunction with a lake polygon to accurately reflect a flow control device. There may be more than one curve type present, but only one of each curve type is allowed at a node. The button for adding a curve type is dimmed out when a curve of that type has been added.

- Rating curve – piece-wise set of linear stage versus discharge values
- Rule curve – step-wise set of stage versus discharge values
- Scheduled release – step-wise set of time versus discharge values

Parameters: table of values representing the curve entered in the XY series editor

Pipe Junction

Data required for modeling storm/tile drain systems is entered at pipe junction nodes.

Super Junction Number

The super junction number is computed by WMS by using the **GSSHA | Number Storm Drain** menu command in the 2D Grid Module, but can also be manually edited at any time.

Invert Elevation

Enter the invert elevation (m) of the pipe at this location.

Manhole Area

Enter the manhole area (m^2).

Inlet Type

The inlet type is used to account for flow into or out of the pipe network. Select *0-9 grate inlets* to specify the amount of flow into the pipe network. Choose *Empty to grid cell* or *Empty to channel* to indicate that flow from the pipe network will be routed either back to the grid cell or into the 1-D stream network.

Hydrograph Output

In addition to writing the hydrograph for the outlet cell location to the *.otl file, GSSHA can output hydrographs at specific nodes to the *.ohl file. Toggle on this option to tell GSSHA to output a hydrograph at this location.

Sediment Output

If the *Soil Erosion* option is selected in the *GSSHA Job Control*, then this option can be used to tell GSSHA to output the sediment load versus time at this location to the *.osl file.

Groundwater BC

Choose one of the following groundwater boundary conditions for modeling groundwater interaction: Generic, Constant Head, Static Well, or Dynamic Well.

Pump Rate

The static well option requires a daily pump rate (m^3/day) and the dynamic well option uses a time varying pump rate entered in the XY series editor by clicking on the **Pump Rate** button.

Solution Results

Click on this button to visualize grid or link/node datasets for this node.

Observations

Click on this button to access the *GSSHA Observations* dialog and compare observed data to results simulated by GSSHA. This can be helpful for calibrating a GSSHA model.

Related Topics

- GSSHA Feature Arcs
- GSSHA Groundwater
- GSSHA Calibration

- GSSHA Observation Points
- GSSHA Solution Results

GSSHAWiki [2]

Primer [3]

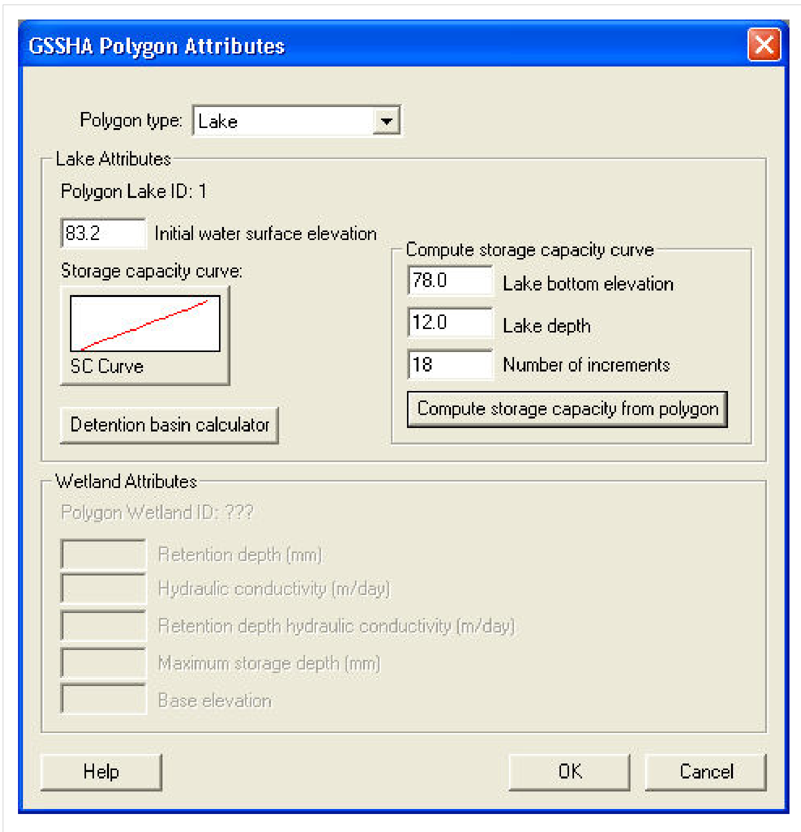
- Watershed Delinieation and Grid Construction [1]
- Lake and Channel Routing [1]

User's Manual [5]

- Surface Water Routing [2]

Tutorials [7]

GSSHA Feature Polygons



A new feature of GSSHA 2.0 is the ability to simulate lakes and wetland areas. By attenuating and storing the incoming flow, lakes and wetlands prove to be important hydrologic features of a basin and dramatically alter the response of a watershed to a precipitation event. Both in-stream and out-of-stream lakes and wetland areas can be simulated. Lakes and wetlands are conceptually modeled in WMS as polygons in the GSSHA coverage.

GSSHA Polygon Type

Generic

Generic polygons have no attributes.

Boundary

Boundary polygons are used to define the perimeter of the watershed and are used for creating grids from an already known watershed boundary.

Lake

Lake polygons are used to define cells within a grid used to simulate lakes during a GSSHA simulation. Initial leakage discharge, spillway crest width, discharge coefficient, initial water elevation and crest elevation must be defined for each lake.

Wetland

Wetland polygons define the cells that form part of a wetland. Five parameters must be specified for each wetlands area, retention depth, hydraulic conductivity, retention depth hydraulic conductivity, maximum storage depth, and the base elevation

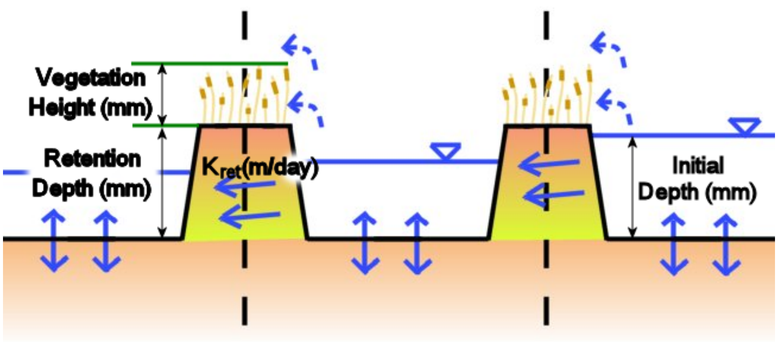


Figure 27. Lateral seepage flow conceptually is the flow through the subsurface peat layer. It is calculated as a darcian flow across the entire cell face. The flow through the vegetation is computed as a mixed mode flow, darcian and manning's. The flow is computed for both darcian and manning's flow and the actual flow rate varies between the two depending on the water height in relation to the plant height.

Depression Mask

Depression mask polygons mask cells contained within the polygon. The elevations of the cells located inside a depression mask polygon are not altered when the CleanDam program is run from WMS.

Related Topics:

- GSSHA Lakes and Wetlands
- GSSHA Digital Dams

GSSHAWiki [2]

Primer [3]

- Watershed Delinieation and Grid Construction [1]
- Lake and Channel Routing [1]

User's Manual [5]

- Surface Water Routing [2]

Tutorials [7]

GSSHA Groundwater

Groundwater interaction in GSSHA models can only be performed in conjunction with one of the following infiltration methods:

- Green & Ampt with soil moisture redistribution
- Richards' equation

To model groundwater interaction in a GSSHA model:

1. Assign groundwater boundary conditions conceptually to feature points/nodes and arcs on the GSSHA coverage.
2. Toggle on the Groundwater option in the Job Control and set the global parameters.
3. Toggle on the Sub-surface losses/gains option for GSSHA stream arcs in order to model groundwater interaction with the GSSHA channel network. (optional)

Groundwater Boundary Conditions

WMS can assign these boundary conditions to feature points/nodes:

- Generic
- Constant head
- Static well
- Dynamic well

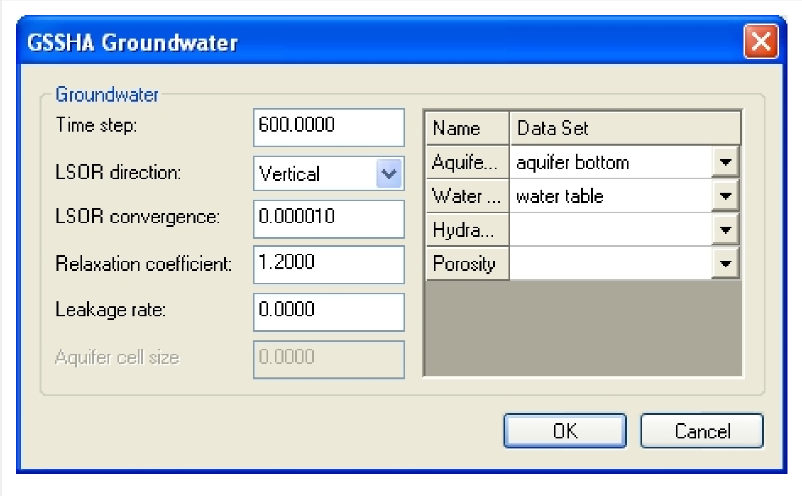
The following groundwater boundary conditions are allowed for feature arcs:

- Generic
- No flow
- Constant head
- Flux river
- Head river

WMS automatically generates Gw Boundary and Wells index maps, according to the groundwater boundary conditions conceptualized using feature points/nodes and arcs on the GSSHA coverage, when the *Groundwater* option is toggled on in the *Job Control*. These index maps will appear in the Index Maps folder of the Project Explorer and are useful for visualizing boundary conditions and well locations/pumping rates. If wells are defined then the Wells map table, which is not visible in the Map Tables dialog, is also populated when the Wells index map is created. Any changes that are made to either the boundary conditions or wells require that you regenerate the index maps (and wells map table) by right-clicking on either of the index maps in the Project Explorer and selecting the **Regenerate** command.

The groundwater boundary condition map and, if necessary, the well index map and map table are also written when the GSSHA *.prj file is saved.

Global Parameters



Enter parameters that control the groundwater computations in the *GSSHA Groundwater* dialog. The Aquifer cell size parameter is the vertical cell size used with Richard's infiltration. Specify continuous datasets used to define the aquifer bottom and water table. Hydraulic conductivity and porosity can be defined using continuous datasets or by assigning parameters in the Groundwater map table based on an index map generated using soil type data.

WMS writes all global groundwater parameters to the GSSHA *.prj file. The groundwater map table (hydraulic conductivity and porosity) will be written to the *.cmt file if Ids exist. Otherwise, hydraulic conductivity and porosity must be specified as continuous maps.

Channels

Interaction between the groundwater and the channel network is controlled for each link by toggling on the *Sub-surface losses/gains* option and specifying a sediment thickness and hydraulic conductivity. These values are written to the *.cif file.

Related Topics

- GSSHA Feature Arcs
- GSSHA Feature Nodes
- GSSHA Maps
- GSSHA Job Control
- GSSHA Map Tables

GSSHAWiki [2]

Primer [3]

- Groundwater [1]
- Richards' Equation [2]

User's Manual [5]

- Groundwater [3]
- Infiltration [4]

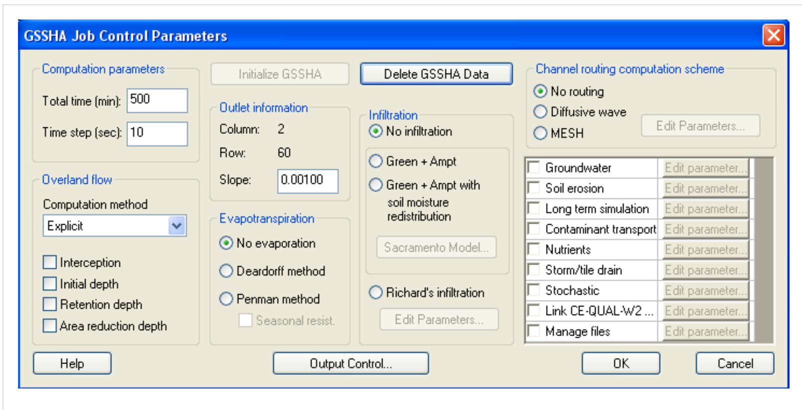
Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Saturated_Flow:Modeling_Two-dimensional,_Saturated,_Lateral_Groundwater_Flow
[2] http://gsshawiki.com/index.php?title=Richards_Equation:Modeling_the_Unsaturated_Zone_with_Richards_Equation
[3] http://gsshawiki.com/index.php?title=Project_File:Saturated_Groundwater_Flow_–_Optional
[4] http://gsshawiki.com/index.php?title=Infiltration:Infiltration

GSSHA Job Control

The *GSSHA Job Control Parameters* dialog is where options and data for computational processes in a GSSHA simulation are specified. The only required process, which is always included in every GSSHA simulation, is overland flow.



Computation Parameters (required)

- Total Time (min)
  - Total time (simulation duration) / time step = total number of time steps. (e.g. 1440 min (total time) / 20 sec (time step) = 4320 computational time steps)
- Time Step (seconds)
  - Choosing an appropriate time step is critical to the success of the simulation. If the time step used is too large then excessive numerical averaging will take place, delaying simulation flows and possibly causing oscillating results. Too small of a time step will take an inordinate amount of time for the simulation to run to completion.
  - A time step that is evenly divisible into 60 sec must be chosen, i.e. 1, 2, 3, 4, 5, 6, 10, 12, 15, 20, 30, or 60 seconds.

Overland Flow

- Overland Flow Computation Method **(required)**
  - The Explicit method is the fastest but least-robust method; the ADE-PC is the slowest, most-robust method
- Overland Flow Modifiers **(optional)**
  - Rainfall Interception – specify the interception parameters using the Interception map table
  - Initial Surface Water Depth – requires a continuous map of initial depths, which WMS does not currently write
  - Surface Water Retention Depth – specify the retention depth using the Retention map table
  - Areal Reduction of Retention Depth – WMS does not currently write the data that GSSHA requires for this option

Outlet Information (required)

- Outlet Cell Location (row, column) – WMS automatically computes these values
- Outlet Slope

Evapotranspiration (optional)

- Deardorff Formulation
- Penman-Montieth Formulation

Infiltration (optional)

- Green & Ampt
- Green & Ampt with Soil Moisture Redistribution
- Green & Ampt with a Long-Term Sacramento Soil Moisture Redistribution Function
- Richard's Formulation for Soil Moisture Redistribution and Groundwater Processes

Channel/Stream Flow Routing (optional)

- Diffusive Wave
- MESH

Other Processes/Options (optional)

- Groundwater (Subsurface)
- Soil Erosion
- Long-Term Simulation of ET, Soil Moisture Redistribution (using hydrometeorological data)
- Contaminant Transport
- Nutrients
- Storm/Tile Drains
- Stochastic – use this option to write stochastic parameter files
- Link CE-QUAL-W2 Output – use this option to write files linking CE-QUAL-W2 output to GSSHA
- Manage Files – use to manage the paths and filenames of all GSSHA input/output files

Related Topics

- GSSHA Overview
- GSSHA Output Control
- GSSHA Precipitation

GSSHAWiki [2]

Primer [3]

- Overview [4]
  - Modeling Process [1]

User's Manual [5]

- Building a Basic GSSHA Simulation [2]

Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Overview:Modeling_process
[2] http://gsshawiki.com/index.php?title=Model_Construction:Building_a_Basic_GSSHA_Simulation

GSSHA Join SSURGO Data

The *.dbf file associated with SSURGO shapefiles contains the following attributes: AREASYMBOL, SPATIALVER, MUSYM, and MUKEY. In order to join more attributes, which can then be mapped to the soil coverage, you should use the **Join SSURGO Data** command.

The **Join SSURGO Data** command opens three *.txt files located in the tabular folder the downloaded SSURGO data: comp.txt, chtextur.txt, and chorizon.txt.

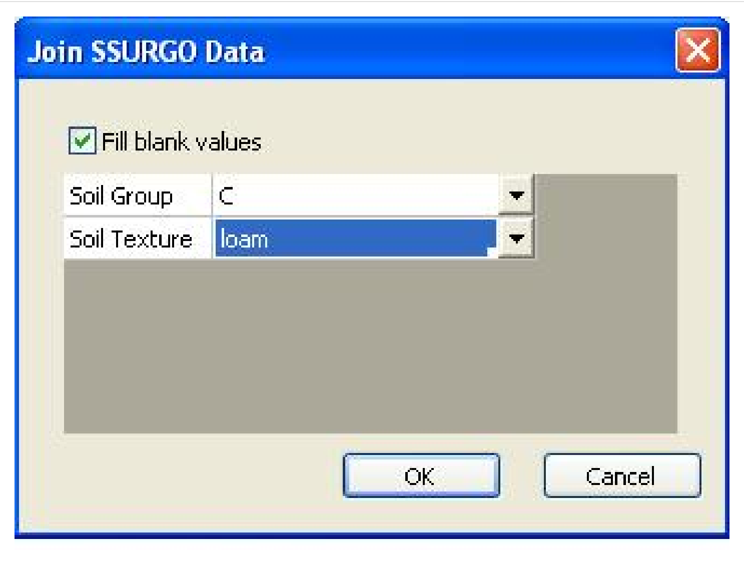
The following data is extracted from the *.txt files:

- **comp.txt**: HYDGRP
- **chtextur.txt**: TEXTURE
- **chorizon.txt**: KSAT, MOISTURE, FIELDCAP, and WILTINGPT.

Clean and Fill Values

Some SSURGO data may contain hybrid HYDGRP classes, such as "A/B." Because WMS does not support hybrid HYDGRP fields, the entry will be changed to first group listed. For example, if the entry is "B/D," WMS will change the entry to "B."

Often there are missing fields in the data. WMS provides the option to set a default value to be assigned to missing fields to avoid blanks. You can set these values in the dialog shown below.



The combo boxes list the soil groups and textures that exist in the SSURGO data and are defaulted to the group and texture that occur most often.

After cleaning and filling the data, the attributes are then matched up with their corresponding MUKEY and joined to the GIS layer. If more than one entry exists for an MUKEY, the corresponding parameters that occur most frequently are joined to the layer.

It should be noted that the shapefile's *.dbf file is not changed in this process. If the shapefile is opened in another WMS project, you will need to select the **Join SSURGO Data** command again in order to join the data.

Related Topics

- GSSHA Overview

GSSHAWiki [2]

Primer [3]

- Overview [4]

User's Manual [5]

- Preface [1]
- Introduction [6]

Tutorials [7]

GSSHA Lakes and Wetlands

Lakes and wetlands are created as feature polygons in a GSSHA coverage.

After creating arcs representing your lake boundary, follow these steps to finish setting up your lake:

1. Create an embankment arc downstream from your lake that represents the dam elevation for the lake. Make sure the type of arc is set to be an embankment arc and that the elevations of the arc are set in the embankment arc editor. When you create your embankment arc, the grid edges defined by your embankment arc must be downstream from the outlet point. If the edges are not downstream from the outlet (you can display the edges to make sure they are outside), you will get an error after you define your lake water surface elevations.
2. Select the arcs representing your lake boundary and build polygons using these selected arcs.
3. Go to the polygon attributes and set your minimum water surface elevation, initial water surface elevation, and maximum water surface elevation in the GSSHA Polygon Attributes dialog.
4. Define any outlet structures, such as culverts, weirs, rating curves, scheduled discharges or rule curves, for the feature node immediately downstream from the lake.

After outlining the area with feature arcs and converting those feature arcs to a polygon, double-click the polygon to bring up the feature attributes dialog. The polygon type then needs to be assigned to either lake or wetland.

The important hydrologic feature of a lake is its ability to store water and route water flows. To accomplish this, an initial water surface elevation and a storage capacity curve must be generated for a lake. The storage capacity curve can be created either manually, through the XY Series Editor accessible by clicking on the **XY series** button, or by using the *Detention Basin Calculator*. In the Detention Basin Calculator the storage capacity curve can be defined as a set of Volume-Elevation curves or Area-Elevation curves. The *Detention Basin Calculator* can also approximate the storage capacity curve by assuming the lake has a pyramidal shape (with constant side slope). In the *feature attributes* dialog, the storage capacity curve can be generated by assuming the lake has a polygonal footprint with vertical sides and a flat bottom. One important note here is that WMS can compute a storage capacity curve from the bathymetry data for a lake or reservoir. To learn more about this option review the TIN-related tutorials in the WMS 8.1 tutorials.

For infiltration from the lake to the groundwater the same infiltration options as are set for the rest of the simulation will be applied. The water surface elevation of the lake will be used when calculating the hydraulic gradient for the infiltration routines.

Wetland areas store small amounts of water and act as important aquifer recharge areas. The areal parameters are assumed to be homogenous over the polygon. Currently, the wetlands areas will act as small lakes with slightly different infiltration parameters. A different hydraulic conductivity value can be specified for the sediments responsible for the wetland condition if a high water table is not the principle cause of the wetlands. Currently the wetlands and lakes do not have any special sediment trapping or contaminant removal parameters. They are simple treated as modifying the stream and overland flow processes. Lakes and wetlands may be in-stream or out-of-stream features.

Lakes and wetlands must be used in conjunction with a rating curve (or other curve or hydraulic structure, if applicable) at the outlet to describe how they release the stored water. The rating curve is set up as a hydraulic structure at the outlet node of the lake or wetland.

The wetland areas are treated as simple flow storage areas with special sediment conditions. The wetland sediments are treated as a lens that must be saturated before it will release water into the groundwater domain. Thus five parameters must be specified for the wetlands area. These are the maximum storage depth, the retention depth (sediment lens thickness), the base elevation, the hydraulic conductivity of the sediment lens, and the hydraulic conductivity of the soil just below the sediment lens (retention depth).

Related Topics:

- GSSHA Feature Polygons

GSSHAWiki [2]

Primer [3]

- Wetlands [1]
  - Wetlands Conceptual Model [2]
  - Wetlands Setup [3]

User's Manual [5]

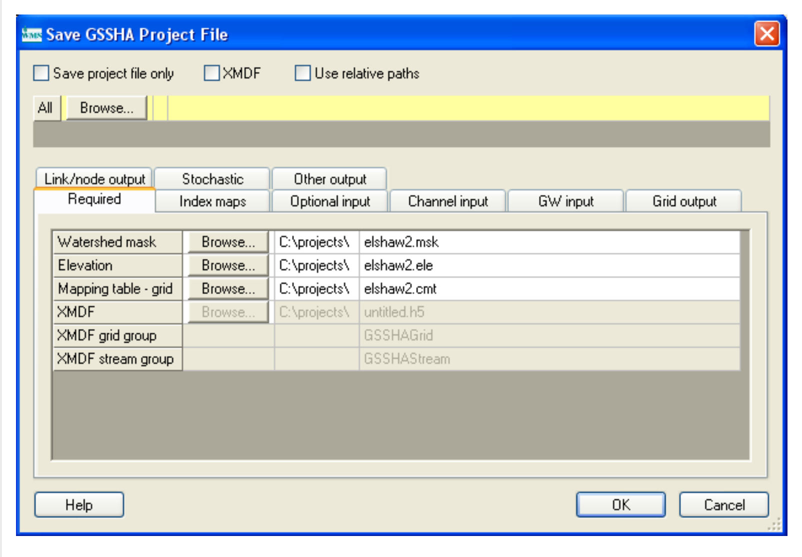
- Surface Water Routing [2]

Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Wetlands:Wetlands
[2] http://gsshawiki.com/index.php?title=Wetlands:Wetlands_conceptual_model
[3] http://gsshawiki.com/index.php?title=Wetlands:Wetlands_setup

GSSHA Manage Files



The *Save GSSHA Project File* editor can be accessed from the GSSHA Job Control options. In the *GSSHA Job Control Parameters* dialog, turn on the spreadsheet option to *Manage files* and select *Edit parameters* to bring up the *GSSHA Project File* editor. You would not normally use this option if you are setting up a basic GSSHA model or if you were new to the GSSHA model.

There are several files that are included as a part of the GSSHA project. If you were building a basic GSSHA model using the WMS interface, you would not normally edit the individual filenames associated with the GSSHA project. However, you might have a specific index map or other project file you want to use or you might want to rename one of the files saved from WMS. Use the *GSSHA Project File* editor to rename one of the files in the GSSHA project to a different name.

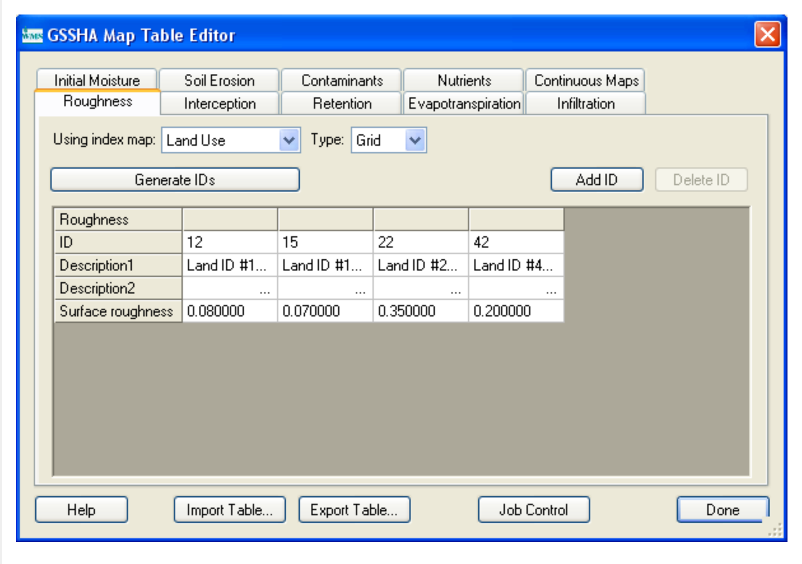
Fields in the *GSSHA Project File* editor are turned off or on depending on whether certain features are turned off or on in the GSSHA interface. For example, if the option to output infiltration depth is turned off in the output control, you cannot edit the name of the infiltration depth filename. Turn on the option to export the infiltration depth from GSSHA to define an infiltration depth filename in the *GSSHA Project File* editor.

If you wish, there is a *Save project file only* option at the top of the dialog to only save a *GSSHA project file* with all of the names specified in the *GSSHA Project File* dialog.

Related Topics

- GSSHA Overview
- Run GSSHA
- GSSHA Job Control
- GSSHA Save Project File

GSSHA Mapping Tables



The mapping tables are where all of the spatially distributed parameters for all of the options specified in the *Job Control* are inputted. The mapping tables relate parameter sets for each process to an index map that shows their spatial distribution.

The several processes available for a GSSHA simulation are shown in the *Process* window. Once a process is chosen, an index map may be assigned to it in the drop-down box below the *Process* window. The **Generate IDs From Map** button takes the assigned index map and creates a list of IDs, shown in the ID window, that are used in the index map. Once the IDs have been generated one may be selected and its properties edited by selecting the property in the *Property Table* and editing the value or string in the edit field below the *Property Table*. The individual ID parameters may be edited as well as the descriptive text that was automatically generated by WMS.

If a process is selected in the *Process* window, and the *Job Control* option for that process is not turned on, WMS will ask if you would like to turn on that process in the *Job Control* and will bring up the *Job Control* dialog for you to do so. If an index map has been assigned to a process and the IDs used in the index map increases then you will need to generate IDs from the index map again and re-input the parameters for each ID for that map.

Related Topics:

- GSSHA Overview
- GSSHA Maps

GSSHAWiki [2]

Primer [3]

- Mapping Tables [1]

User's Manual [5]

- Mapping Tables [4]

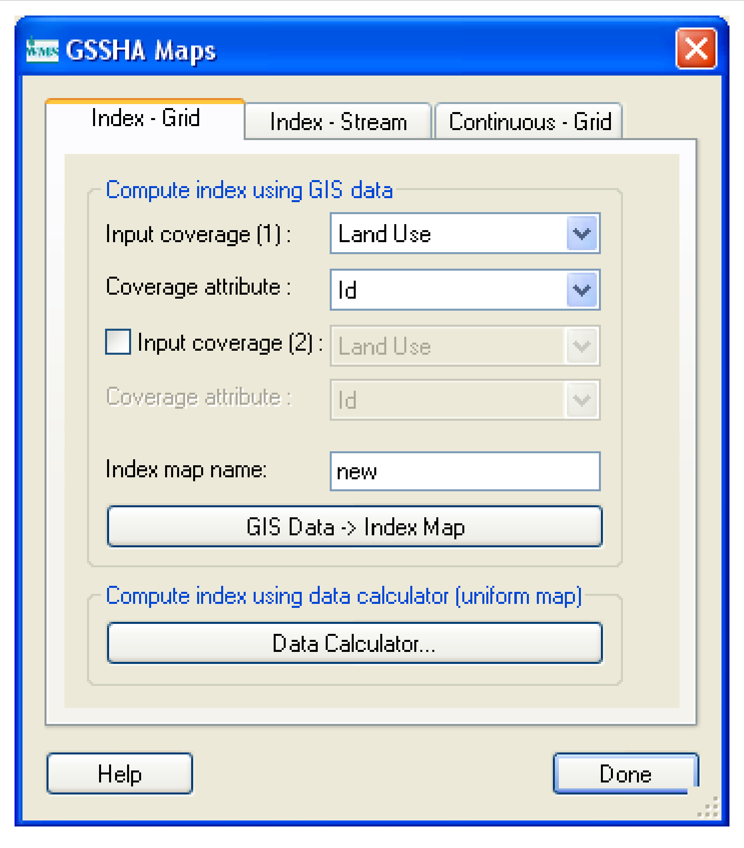
Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Mapping:Mapping_tables

GSSHA Maps

Index Maps



One of the greatest assets of distributed hydrologic models like GSSHA is the ability to spatially distribute the parameters for processes, such as overland flow and infiltration, over the watershed. Assigning values, grid cell by grid cell, is tedious and makes all but the simplest and smallest models impossible. Using WMS, GIS coverages (layers) representing land use and soil texture can be used to assign model parameter values to groups of grid cells sharing the same characteristics.

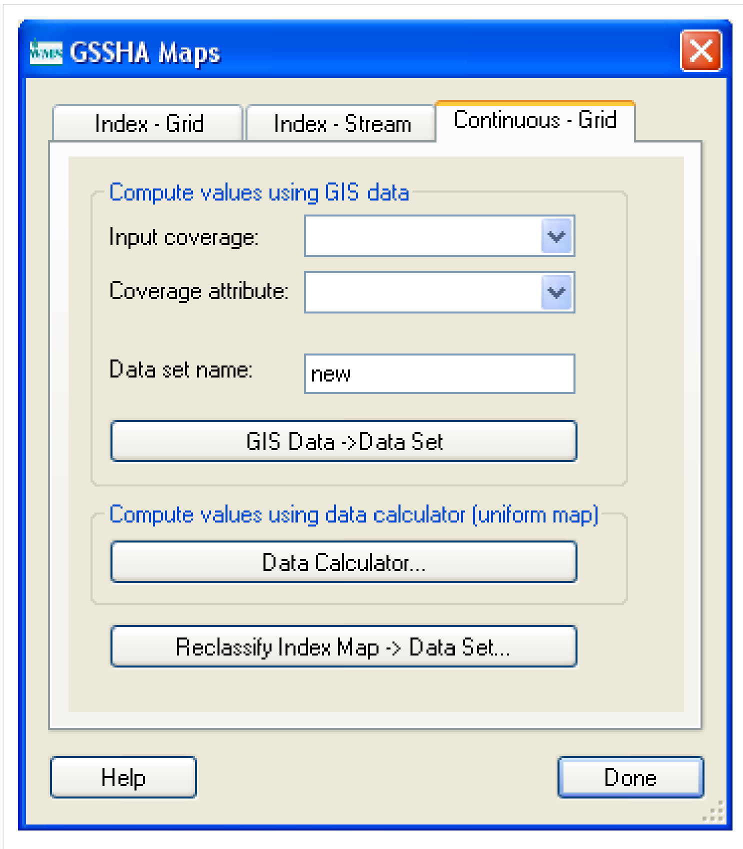
The basic process of assigning spatially distributed parameters consists of the following steps:

- Import a GIS coverage for land use, soil texture, or vegetation type (generally this should be in the ArcView® shapefile format).
- Map the land use, soil texture, or vegetation ID to the grid cells using a spatial overlay operation.
- Define parameter values (e.g., surface roughness, hydraulic conductivity, etc.) for the unique ID numbers.

A given soil texture/land use (STLU) index map can be used to assign multiple parameters. Since most of the grid cell parameters can be referenced to either land use or soil properties, a given simulation generally requires only a single index map of each. A combination land use and soil texture index map makes it possible to relate a parameter value to the combination of land use and soil texture (for example infiltration or erosion). Once the index maps are defined, parameter values are assigned to the IDs of the index maps. The combination of the index maps, with ID numbers, and the mapping tables, with the parameter values, are used by GSSHA to internally assign parameter values to each grid cell.

The principle means of modifying and creating index maps is in the *Index Map* dialog but the index maps are now also able to be accessed through the *Project Explorer*. When an index map has been created or read into WMS a folder appears in the *Project Explorer*, named *Index Maps*, that contains all of the index maps for the simulation. Index maps can then be treated like regular datasets; they can be contoured, renamed, deleted, and edited.

Continuous Maps



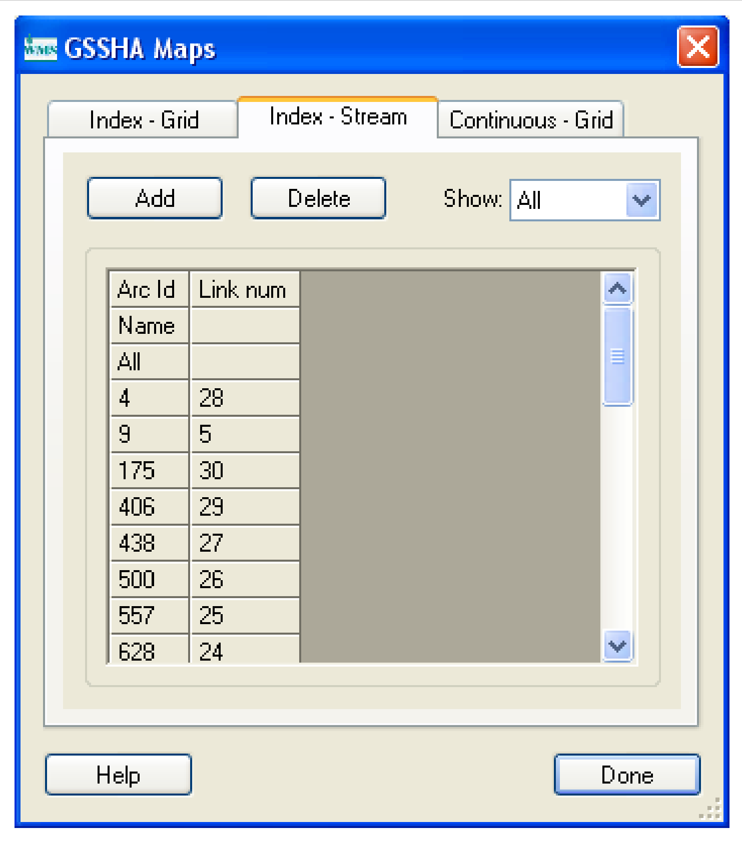
Some GSSHA input parameters change from cell to cell and cannot be characterized using index maps. Continuous maps, which are stored as datasets of a 2D grid in WMS, are used where index maps are not appropriate. Examples of GSSHA input parameters that use continuous maps include soil properties for the groundwater processes and ground surface, bedrock, and water table elevations. Contours for each of the continuous maps are displayed on the 2D grid according to the contour options selected for the dataset. Continuous maps are specified as GSSHA input in either the Sub-surface Parameters (Groundwater) dialog or in the *Continuous Maps* tab of the GSSHA Map Table Editor dialog.

The *Continuous – Grid* tab of the *GSSHA Maps* dialog has three options for generating continuous maps:

- Using GIS Data: Select an input coverage and attribute. Enter a name for the continuous map in the dataset name field. Click the **GIS Data → Dataset** button to create the dataset.
- Using the *Data Calculator*: This option is useful for mathematically manipulating existing continuous maps. Click the **Data Caluclator...** button to open the *Data Calculator* window.
- Reclassification: Reclassify index map values to create a continuous map, if appropriate. Click the **Reclassify Index Map→Dataset...** button to begin.

Stream Index Maps

Sometimes, you might have a stream-based index map instead of a grid-based index map. When you define a stream index map, define an index value greater than zero to each link in your model. There is not currently an option to automatically define stream index map values from grid or map module data. You can turn on the stream link IDs in the *Display Options* dialog and define an index for each of these IDs. You can assign index values to each of the stream links in your model. You can then use stream index maps to define Manning's roughness, nutrient values, and other values in the *GSSHA Map Table Editor*.



Related Topics

- GSSHA Overview

GSSHAWiki [2]

Primer [3]

- Assigning Parameter Values to Individual Grid Cells [1]
  - Index Maps [1]

User's Manual [5]

- Mapping Table File [2]
  - Index Maps [3]

Tutorials [7]

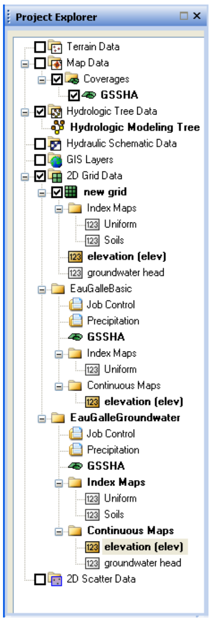
References

[1] http://gsshawiki.com/index.php?title=Mapping:Index_maps

GSSHA Multiple Scenarios

Overview

Multiple GSSHA models using the same 2D grid can be managed within a single instance of WMS. The models are displayed in the data tree, where the user can edit the job control, precipitation, GSSHA coverage, index maps, and continuous maps.



Job Control – The job control for the active GSSHA model can be accessed from the GSSHA menu item. The job control can also be opened by right-clicking on the *job control icon* that corresponds with the desired GSSHA model and selecting **Edit**.

Precipitation – The precipitation for the active GSSHA model can be accessed from the GSSHA menu item. The precipitation can also be opened by right-clicking on the *precipitation icon* that corresponds with the desired GSSHA model and selecting **Edit**.

GSSHA Coverage – When a GSSHA model is created, it is a assigned a GSSHA coverage (if a GSSHA coverage does not already exist, WMS creates one). To change the assigned GSSHA coverage, right-click on the coverage icon that corresponds with the desired GSSHA model and select the GSSHA coverage that you want to assign.

Index Maps – All index maps for all GSSHA models are shown in the data tree under the grid. To assign a map to a specific GSSHA model, right-click on the *Index Maps* folder under the desired GSSHA model, select **Assign**, and then select the index map that you want to assign. To unassign an index map, right-click on the map and select **Remove**.

Continuous Maps – All continuous maps for all GSSHA models are shown in the data tree under the grid. To assign the maps to a specific GSSHA model, right-click on the *Continuous Maps* folder under the desired GSSHA model, select **Assign**, and then select the index map that you want to assign. To unassign a continuous map, right-click on the map and select **Remove**.

Solution – Once a GSSHA model has successfully run, the solution is shown underneath the model in the data tree. If the solution already exists, it can be read in by right-clicking on the model's project name and selecting **Read Solution**.

Most of the options available under the GSSHA menu are also found in the right-click menu of the GSSHA project in the data tree. When selected from the GSSHA menu, they are performed on the active model. When selected from the right-click menu, they are performed on the corresponding model.

GSSHA Groups

Saving GSSHA Groups – Multiple GSSHA scenarios can be saved together as a group. To do this, right-click on the *2D Grid Folder* and select **Save GSSHA Group.** A dialog will appear to prompt you to select which scenarios you want to save. You will also need to pick a location for WMS to save the scenarios and the GSSHA Group File (*.ggp).

Opening a GSSHA Group – To open a group of GSSHA scenarios, right-click on the *2D Grid Folder* and select **Open GSSHA Group** and select the GSSHA Group File (*.ggp) that you wish to open. This will open each project that corresponds with this group. You also have the option to open the solutions, if they exist.

Running a GSSHA Group – To run multiple GSSHA scenarios, right-click on the *2D Grid Folder* and select **Run GSSHA Group.** A dialog will appear in which you can select which projects you want to run. These projects will be saved as a group and then WMS will run each GSSHA scenario in the group.

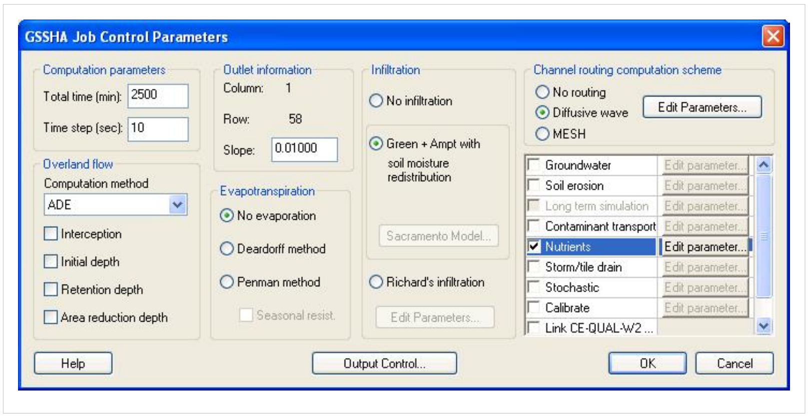
Related Topics

- GSSHA Overview

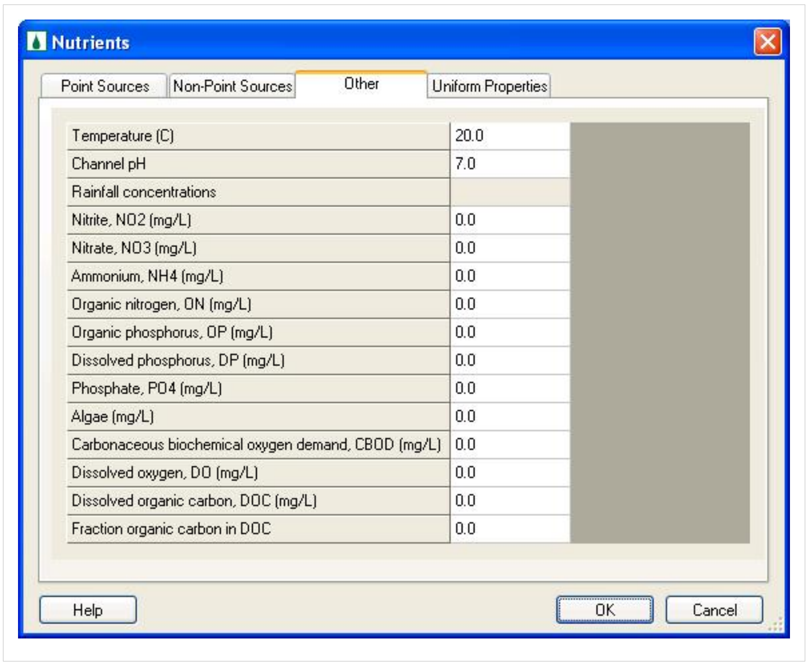
GSSHA Nutrients

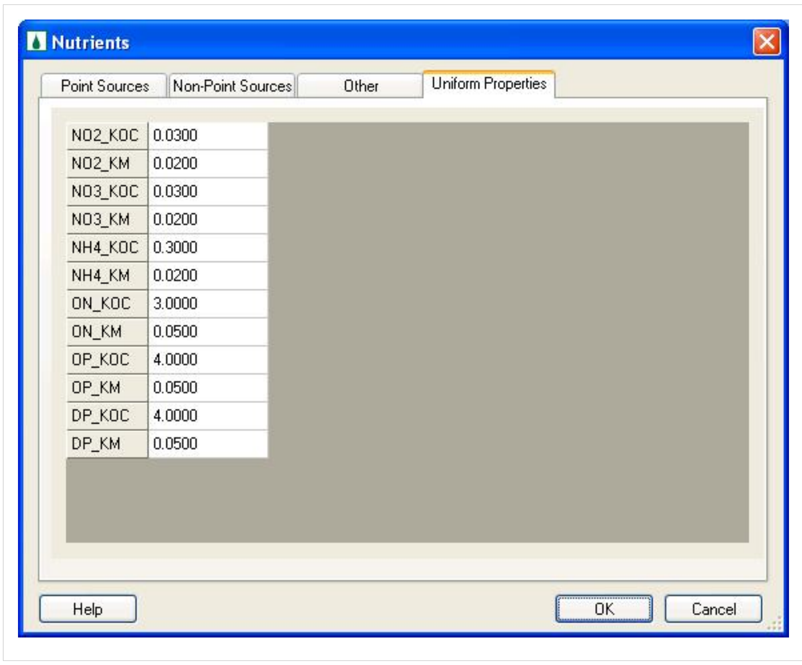
To model nutrients using GSSHA:

1. Toggle on the *Nutrients* option in the *Job Control*.

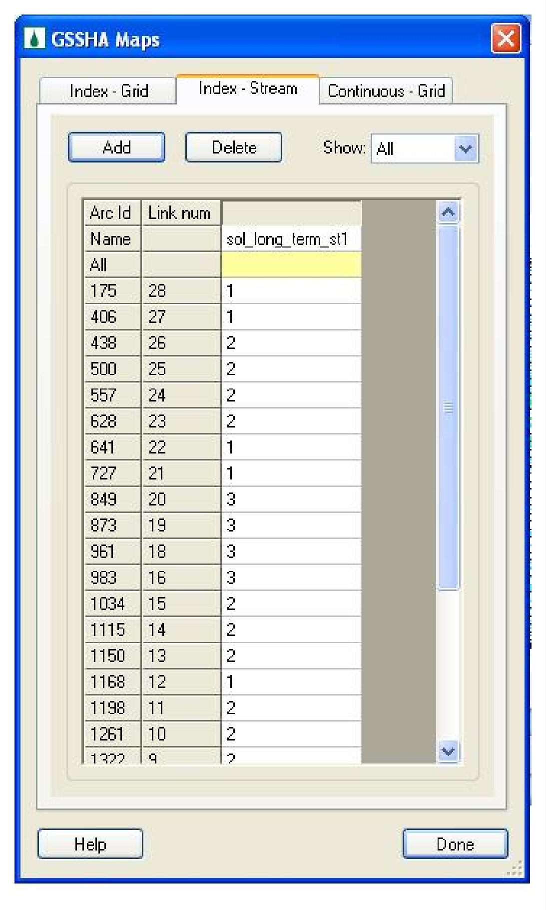


2. Open the *Nutrients* dialog to view the four nutrient tabs: *Point Sources*, *Non Point Sources*, *Other*, and *Uniform Properties.* Enter nutrient concentrations in the Other tab. These concentrations will be applied to rainfall input. The *Uniform* properties tab contains default values for uniform nutrient properties. You may adjust these values as you see fit.





3. Create stream and grid index maps. At least one stream and one grid index map is required for nutrient setup.



4. Set up the initial conditions for the stream and grid index maps using the Nutrients map tables in the GSSHA Map Table dialog. The following map tables are supported:
    - Aquatic kinetic constants
    - Dispersion
    - Nitrogen initial conditions
    - Phosphorus initial conditions
    - Carbon initial conditions
    - Other initial conditions
    - Soil nitrogen initial conditions
    - Soil phosphorus initial conditions
    - Soil carbon initial conditions
    - Soil uptake rates
    - Soil/water partitioning
    - Groundwater nitrogen initial conditions
    - Groundwater phosphorus initial conditions
    - Groundwater other initial conditions
5. Turn on any nutrient related output options in the *Output Control* dialog to view your desired datasets.
6. If you want to define Point Sources or Non-Point Sources, select the *Nutrients* button in the **GSSHA Job Control** dialog. If defining non-point sources, you need to define a grid-based or a stream-based index map associated with the non-point source. If defining a point source, you need to determine the *I, J* index of the cell if defining an overland point source or the *Link, Node* index if defining a stream point source. The I, J can be determined from the grid cell properties in the Properties window. The Link number can be determined by turning on the link number display option and viewing the link numbers in the WMS graphics window. Whether defining a point or non-point contaminant source, you need to define a mass curve or a concentration curve for any contaminant source you want to model.

Other important troubleshooting notes:

- GSSHA nutrients can only be modeled in long-term simulations
- The contaminants option must be turned on with at least one contaminant in order for nutrients to run. In this case, all contaminant parameters can be left at 0.
- You must turn on the initial depth option in the job control and specify an initial depth for any initial condition nutrient mapping tables to have any effect
- You must have at least one ID for all nutrient mapping tables for nutrients to run properly, even if the values for some IDs are all 0.

Related Topics

- GSSHA Maps

GSSHAWiki [2]

Primer [3]

User's Manual [5]

- NSM Project Cards [1]
- NSM Aquatic Kinetic Constants [2]
- Point/Non-point Sources [3]
- NSM Map Tables [4]

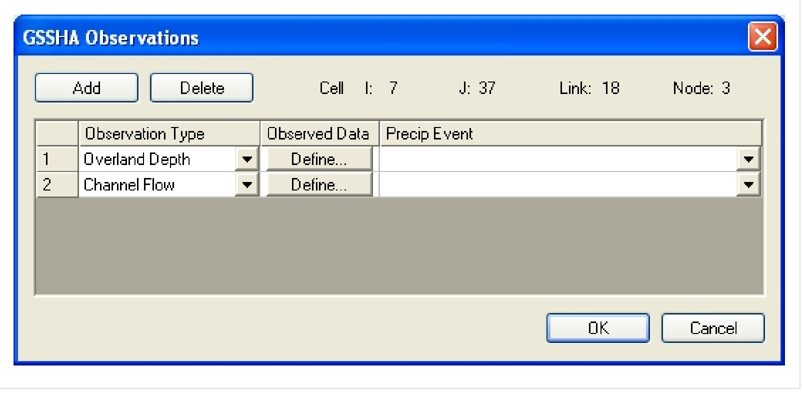
Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Constituents:Nutrient_Sub_Module_(NSM)_Constituents,_Project_Cards
[2] http://gsshawiki.com/index.php?title=Constituents:Nutrient_Sub_Module_(NSM)_Constituents,_Aquifer_Environment_File
[3] http://gsshawiki.com/index.php?title=Constituents:Point_and_Non-point_sources
[4] http://gsshawiki.com/index.php?title=Mapping_Table:NSM_Grid-based_Mapping_Tables

GSSHA Observation Points

Observed data are stored as part of the attributes of a feature point in the GSSHA coverage. Click on the button in the *Observations* column for any feature point/node in the *Feature Point/Node Properties* dialog to access the *Observations* dialog.



Observations are managed using the **Add** and **Delete** buttons in the dialog. Specify the observation type by choosing from any of the gridded or link/node datasets that GSSHA can output. Click on the **Define...** button to enter the observed data using the XY Series Editor. It is optional to associate the observation with a specific rain event (GSSHA rain gage coverage) if multiple events are defined. Both the link/node numbers and the I, J values of the cell encompassing the feature node are displayed in the dialog.

Related Topics

- GSSHA Calibration
- GSSHA Feature Nodes

GSSHAWiki [2]

Primer [3]

- Watershed Delinieation and Grid Construction [1]
- Lake and Channel Routing [1]

User's Manual [5]

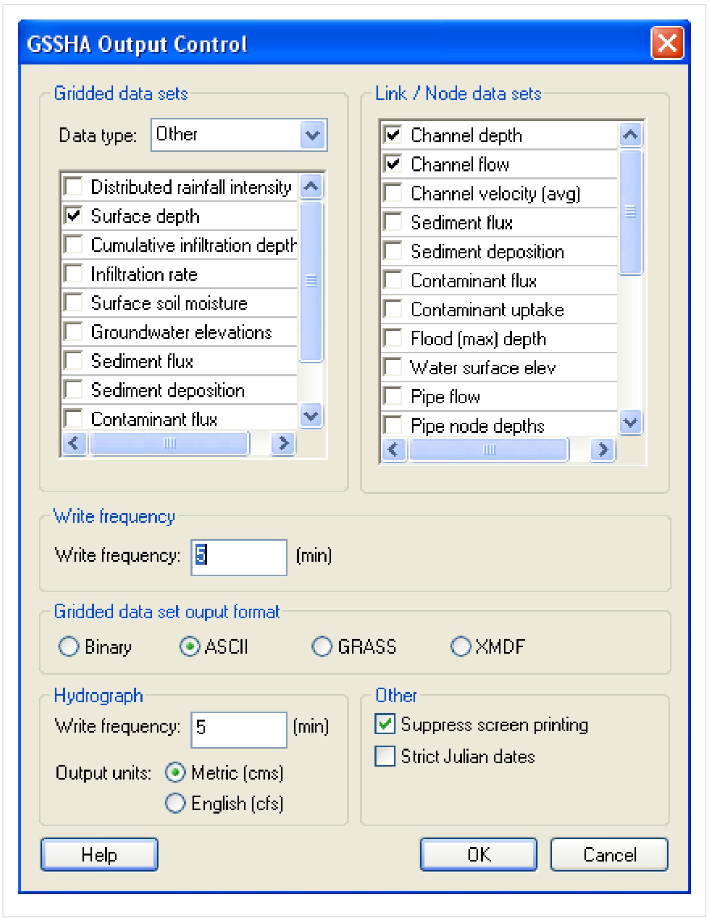
- Surface Water Routing [2]

Tutorials [7]

GSSHA Output Control

The *Output Control*, accessible from the *Job Control* dialog, is where most of the output options for a GSSHA simulation are set. GSSHA can write out two types of datasets: grid based and link/node (point) based. All of these data sets will be read in to WMS, if they exist, as part of a GSSHA solution.

Gridded Datasets



Because there are so many gridded datasets that GSSHA can output, they are separated into two categories: General and Nutrients – Overland.

General

- Distributed Rainfall Intensity
- Surface Depth – depth of water on the overland flow plane
- Cumulative Infiltration Depth – if an infiltration option is turned on
- Infiltration Rate – if an infiltration option is turned on
- Surface Soil Moisture – if an infiltration option with soil moisture redistribution is turned on
- Groundwater Elevations (Head)
- *Volume suspended sediment*
- Sediment Flux – maximum flux on the overland flow plane
- Net Sediment Transfer – erosion/deposition on the overland flow plane
- Flood (max) Depth – values in each grid cell may occur at different time steps

*WMS does not currently write the cards and file names associated with these output options to the GSSHA *.prj file.*

Nutrients – Overland

- Nitrite (NO2-)
- Nitrate (NO3-)
- Ammonium (NH4+)
- Organic Nitrogen
- Organic Phosphorus
- Dissolved Phosphorus
- Algae
- Carbonaceous BOD
- Dissolved Oxygen

Link/Node Datasets

Link/node datasets report values at the nodes along the links of the GSSHA stream/channel network. The files are written in a format unique to GSSHA.

- Channel Depth
- Channel Flow
- Channel Velocity (average)
- Sediment Flux
- *Net Sediment Transfer*
- Flood (max) Depth
- Water Surface Elevation
- Pipe Flow
- Pipe Node Depths
- Pipe Node Inflow/Outflow
- Nitrite (NO2-)
- Nitrate (NO3-)
- Ammonium (NH4+)
- Organic Nitrogen
- Organic Phosphorus
- Dissolved Phosphorus
- Algae
- CBOD
- Dissolved Oxygen

*WMS does not currently write the cards and file names associated with these output options to the GSSHA *.prj file.*

Write Frequency

Specify how often to write output values to the grid and the link/node output files. GSSHA will output data for a gridded dataset after the first computational time step and then using the specified write frequency. For example, a 2 hour simulation with a time step of 30 seconds and a write frequency of 30 minutes will produce output at 30 seconds, 30 minutes 30 seconds, 60 minutes 30 seconds, and so on. Data for link/node datasets are output before the first computation time step (initial values) and then according to the specified write frequency. Using the previous example link/node data sets are written at 0 seconds, 30 minutes 0 seconds, 60 minutes 0 seconds, and so on.

Gridded Dataset Output Format

- Binary
- ARC/INFO® ASCII
- GRASS ASCII
- XMDF

Hydrograph

GSSHA will output the outlet hydrograph to the *.otl file using the specified write frequency. Even though GSSHA performs all computations using metric units, it is possible to have GSSHA output flows to the outlet hydrograph file in either metric or English units.

Other

- Suppress Screen Printing – this option will not show output at each computational time step while GSSHA is running, but can significantly reduce the runtime required for a GSSHA simulation. This option can also be selected in the *GSSHA Run Options* dialog.
- Strict Julian Dates – write all dates to output files using strict Julian dates

Related Topics

- GSSHA Job Control
- GSSHA Overview
- GSSHA Read Solution

GSSHAWiki [2]

Primer [3]

- Post-Processing [1]
  - Output Control [2]

User's Manual [5]

- Output [3]

Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Post-Processing:Post-Processing
[2] http://gsshawiki.com/index.php?title=Post-Processing:Output_control
[3] http://gsshawiki.com/index.php?title=Output:Output

GSSHA Overland Soil Erosion

Sediments in WMS are built around a similar concept as the contaminants. Just like contaminants are individual processes to be modeled, so are sediments. Soil types are defined as having proportions of sediment types in them. To simulate sediments, the sediment option in the *Job Control* dialog must be turned on.

Sediments are set up in three steps:

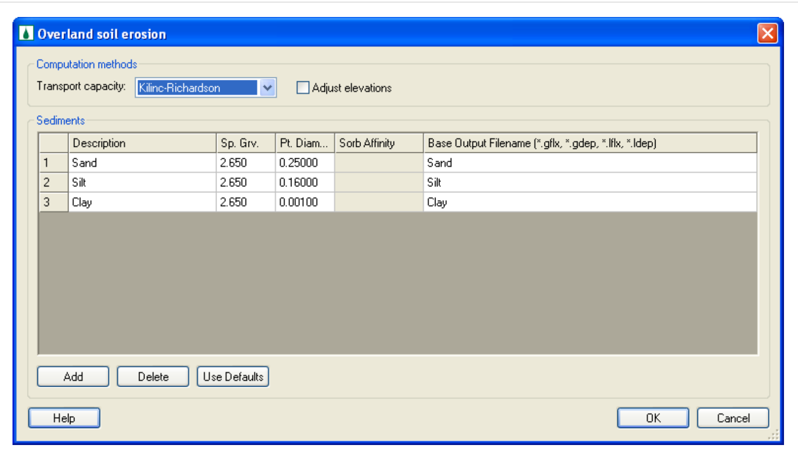
1. Create the sediment types,
2. Create a soil type index map,
3. Define sediment proportions and other relevant parameters for each soil type.

Create the sediment types

First, a set of sediments must be created. The *Sediments* dialog is accessible from the *GSSHA* menu if sediments are turned on in the *Job Control* dialog, or it is accessible directly from the *Job Control* dialog. As the soil types will be defined in terms of portions of sediments, the set of sediments must contain all major sediment types in the simulation. For example, if the only soil type in the simulation is a silty clay, then two sediment types must be defined, namely silt and clay.

Each sediment type has the physical properties of particle size and specific gravity, which are also specified in the *Sediments* dialog. Sediment types can be broken down into as fine or coarse of categories as desired. Selecting the **Use Defaults** button in the *Sediments* dialog will create four sediment types, a medium gravel, a medium sand, a medium silt, and a medium clay. The output filename parameter for each sediment type defines the name of the file that GSSHA will create showing the areas of deposition and scouring for each sediment type.

Allowing the user to create the sediment types offers a powerful means in GSSHA to simulate any sort of particulate transport, even solid contaminants such as lead or uranium. These are as simply set up as all the other sediment types. Once they are set up, an overlay soil type coverage can be combined with the real soil type coverage to generate the contaminant distributions.



The set of all sediments making up all the soil types for the simulation are defined in the sediments dialog. The **Use Defaults** button replaces the sediments already listed with the four types of sediments. For each sediment type, the particle specific gravity and average diameter must be given. An output file name must also be specified where GSSHA will write the results of the solution for that sediment type. The computation method should be specified in the *Overland Soil Erosion* dialog and the ADJUST_ELEV card will be written to the GSSHA project file if the *Adjust elevations* option is turned on. The ADJUST_ELEV card allows GSSHA to change the elevation in the model based on erosion and deposition and writes the adjusted elevations to a file.

Create a soil type index map

As with the contaminants, the next step in the process is to create the soil type index map. Creating this map is a fairly straightforward process. If no special sediment types are to be simulated, a soil type coverage imported from readily available files, such as those available from the EPA or the NRCS in the statsgo or ssurgo formats, is sufficient. Simply generate the index map from the coverage. Later, the proportions of sediments for each soil type will be set up.

If a particulate contaminant is being simulated then a contaminant distribution coverage must be defined (the actual coverage type should either be land use or soil type as there is no “contaminant distribution” coverage type), as is the case with the dissolved contaminants. The area defining the extent of each contaminant must be defined, as well as a polygon that covers the entire grid where no contaminant is present. When the index map is created, use both the soil type coverage and the contaminant distribution coverage.

Define sediment proportions and other relevant parameters for each soil type

In the GSSHA Index Map Table Editor, select the soil erosion properties table and assign the correct index map. After generating the IDs from the map, assign the parameter values for soil erodability and dispersivity. Below these two parameters is a list of all of the sediment types, which were created in the Sediments dialog. To each sediment type assign the proportion of that sediment that makes up the soil type. For example, if the soil type is a silty clay, with 20% silt and 80% clay, a 0.2 should be assigned to the silt and a 0.8 the clay, with 0.0 assigned to the rest of the sediment types (if any). If a sediment type has not been created that should have been, simply return to the Sediments dialog and add the sediment type. Note that the sum of the sediment type proportions for each soil type needs to add up to 1.0 before GSSHA is run. If sediment types are added or deleted after setting up the soil type parameters the percentages will need to be checked.

Related Topics

- GSSHA Overview
- Contaminant Transport
- GSSHA Maps
- Mapping Tables
- Job Control Dialog

GSSHAWiki [2]

Primer [3]

- Mapping [1]

User's Manual [5]

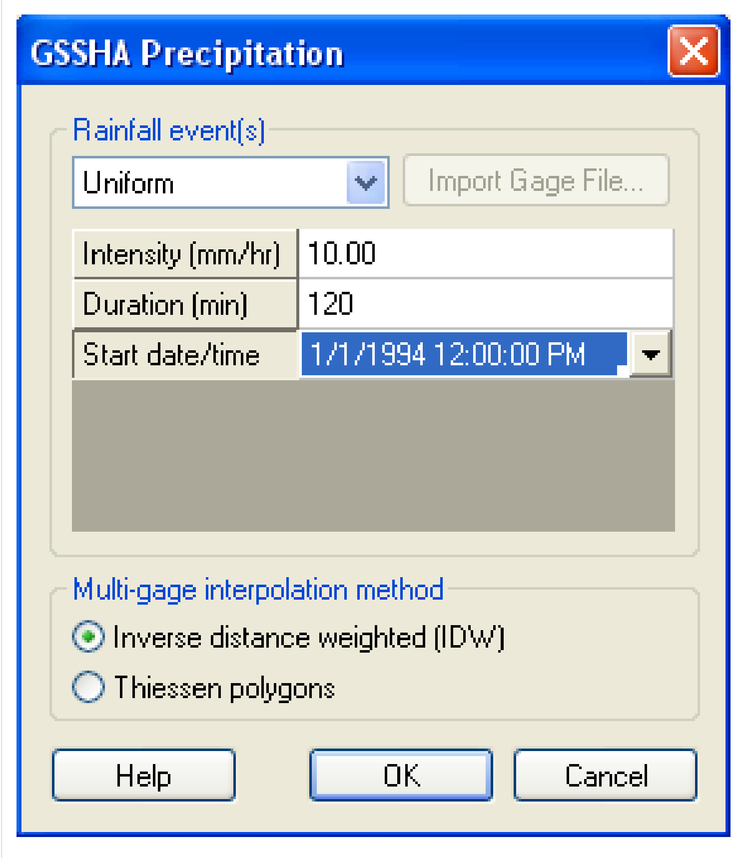
- Mapping Table File [2]
  - Index Maps [3]
  - Mapping Tables [4]

Tutorials [7]

GSSHA Precipitation

Rainfall can be input in one of the following formats:

- **Uniform** constant rainfall over the entire watershed.
- **Single** temporally varying rain gage.
- **Multiple** temporally varying rain gages.
- **User defined** hyetograph.
- **NEXRAD** radar data.



All rainfall types in GSSHA must be tied to a specific point in time by specifying the year, month, day, hour, and minute of each rainfall data point. The *GSSHA Precipitation* dialog in WMS allows you to specify the type of rainfall and enter the necessary data associated with the rainfall type. The *GSSHA Precipitation* dialog is accessed from the *GSSHA* menu in the 2D grid module.

Uniform Rainfall

The ability to assign uniform rainfall over the entire watershed is maintained largely as a trouble-shooting feature and is mostly used in initial model development. Real watersheds are modeled with temporally and spatially varying rainfall. For spatially and temporally constant rainfall, the input parameters are:

- Rainfall Intensity (mm/hr)
- Rainfall Duration (minutes)
- Start Time
    1. Year
    2. Month
    3. Day
    4. Hour
    5. Minute

Gage Rainfall

Select the *Gage* option to specify single or multi-gage rainfall. Rainfall distributions for all rain gages should already be entered on a rain gage coverage in WMS. Create a separate rain gage coverage for each precipitation event. Rain gage files constructed using an editor outside of WMS can be used in two ways: reference the file in the Manage Files dialog when saving the GSSHA project file or use the **Import Gage File...** button to read the file and generate rain gage coverages in WMS.

Temporally Varying, Spatially Uniform (Single Gage) Rainfall

Single-gage rainfall produces a time series of rainfall for the entire watershed. Toggle on the the rain gage coverages (containing a single gage) in the dialog that will be used for writing the *.gag file required by GSSHA.

Temporally Varying Multiple-Gage Rainfall

Select rain gage coverages with multiple gages defined to be used for writing the *.gag file required by GSSHA. Specify whether to use Inverse distance weighting (IDW) or Thiessen polygons for determining the spatial variation in rainfall for each grid cell.

Hyetograph

Select the *Hyetograph* option to enter a temporally varying, spatially uniform (single-gage) event without creating a rain gage coverage. Enter the average total depth (mm) of precipitation across the drainage area and the start date/time. Click on the **Define Distribution...** button to define the temporal distribution.

NEXRAD

Select the Nexrad Radar option and click on the Import Radar Data... button to process NEXRAD rainfall data for use in GSSHA. WMS will write the GSSHA rainfall (*.gag) file using the RADAR type. For RADAR type rainfall inputs, Thiessen polygons should be selected as the interpolation method.

Related Topics

- GSSHA Overview
- GSSHA Job Control

GSSHAWiki [2]

Primer [3]

- Spatially and Temporally Varying Rainfall [1]
  - Temporally Varying - Spatially Uniform Rainfall [2]
  - Temporally Varying Multiple Gage Rainfall [3]

User's Manual [5]

- Precipitation [4]
  - Spatially and Temporally Uniform Precipitation [5]
  - Spatially and Temporally Varied Precipitation [6]
  - Interpolation Between Gages [7]
  - Interception [8]

Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Rainfall:Spatially_and_temporally_varying_rainfall
[2] http://gsshawiki.com/index.php?title=Rainfall:Temporally_varying%2C_spatially_uniform_rainfall
[3] http://gsshawiki.com/index.php?title=Temporally_varying_multiple_gage_rainfall
[4] http://gsshawiki.com/index.php?title=Precipitation:Precipitation
[5] http://gsshawiki.com/index.php?title=Precipitation:Spatially_and_Temporally_Uniform_Precipitation
[6] http://gsshawiki.com/index.php?title=Precipitation:Spatially_and_Temporally_Varied_Precipitation
[7] http://gsshawiki.com/index.php?title=Precipitation:Interpolation_Between_Gages
[8] http://gsshawiki.com/index.php?title=Precipitation:Interception

GSSHA Read Solution

While solutions are not new to GSSHA, being able to work with them in WMS is a new feature. Accessible from the *GSSHA* menu in the 2D Grid module is the command **Read Solution** which looks for a GSSHA project file and then reads in all of the associated datasets and lumps them together into a solution folder in the Project Explorer. Solution folders are identified by a lowercase "s" on the folder. All of the datasets in the folder are treated as regular datasets. Organizing the datasets into a solution allows several solutions to be in memory at the same time. Several dialogs look for solutions and the associated datasets to set up and display output graphs. Along with the regular datasets, the summary file for the project is also accessible for each solution by double-clicking the summary file project explorer item under the solution folder.

Individual solution output time series data for a cell may be viewed from the *Solution Results* dialog accessed from the *Feature Point/Node Type* dialog. This dialog will only show the output dataset time series for the cell that underlies the feature point selected. To compare the solution output at a cell with observed data, see the GSSHA Observation Points section.

Related Topics

- GSSHA Overview
- Output Control
- GSSHA Observation Points
- GSSHA Solution Results

GSSHAWiki [2]

Primer [3]

- Running GSSHA [1]
  - Running GSSHA from WMS [2]
  - Project File [3]

User's Manual [5]

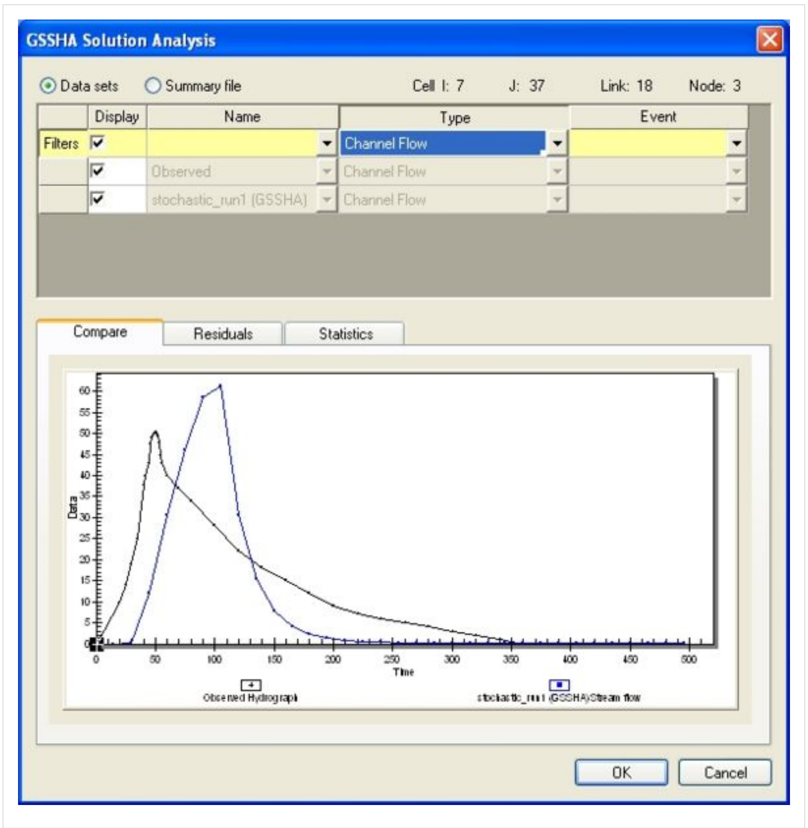
- Output [3]
  - Run Summary File [4]

Tutorials [7]

References

[1] http://gsshawiki.com/index.php?title=Running_GSSHA:Running_GSSHA
[2] http://gsshawiki.com/index.php?title=Running_GSSHA:Running_GSSHA_from_WMS
[3] http://gsshawiki.com/index.php?title=Running_GSSHA:Project_file
[4] http://gsshawiki.com/index.php?title=Output:Run_Summary_File

GSSHA Solution Analysis



The *GSSHA Solution Analysis* dialog is primarily used for manual calibration as well as visualizing solution results. There are two types of solution analysis:

- Datasets
- Summary file

Options specified in the spreadsheet in the upper portion of the dialog control what is shown in the tabs below. The *Filters row* in the spreadsheet is used to filter the results shown in the spreadsheet based on Name, Type, or Event. Use the toggle in the *Display* column to toggle on the display of all datasets shown in the spreadsheet.

Datasets

Both observations and gridded and link/node datasets that are part of all GSSHA solutions are shown in the spreadsheet. Anything shown in the spreadsheet with the *Display* toggled on is used for generating the *Compare* and *Residuals* plots and for computing Statistics using the residuals.

Summary file

After running the GSSHA simulation, GSSHA writes a summary file. The results for each event of the simulations is stored in WMS and shown in the spreadsheet. The observations are also listed in the spreadsheet, which if toggled on will be plotted with the solution results. This tool can be especially useful when running GSSHA in batch mode. The user can easily and quickly view a plot of the simulations in order to determine which of the runs produce results similar to the observed data.

Related Topics

- GSSHA Calibration
- GSSHA Observation Points
- GSSHA Read Solution

GSSHAWiki [2]

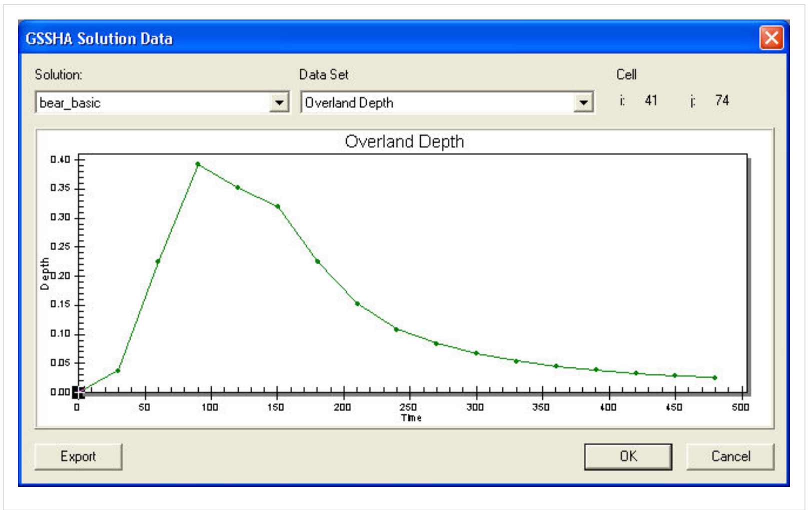
Primer [3]

User's Manual [5]

Tutorials [7]

GSSHA Solution Results

The *Solution Results* dialog, accessible from the *Feature Node Attributes* dialog, is a means to display the output of grid based and point based datasets produced by GSSHA during a simulation. For the point based datasets, only datasets originally associated with that feature node can be viewed at that node. For gridded datasets the cell underlying the feature node is used to extract a time series for the specified dataset.



Related Topics

- GSSHA Obervation Points
- GSSHA Feature Nodes

GSSHAWiki [2]

Primer [3]

- Running GSSHA [1]
  - Running GSSHA from WMS [2]
  - Project File [3]

User's Manual [5]

- Output [3]
  - Run Summary File [4]

Tutorials [7]

GSSHA Stream Arcs

WMS uses a conceptual model approach to building GSSHA simulations that allows for the use of GIS data objects. In WMS stream arcs are used to define both the spatial extent as well as the hydraulic characteristics of the streams for a GSSHA simulation. While the underlying process producing the stream flow in GSSHA has been significantly updated the process of creating the streams in WMS has not been significantly altered, only simplified. In WMS 6.1 a set of stream arcs were either created from a DEM or read in from a GIS shapefile. These arcs then had to be manipulated spatially to follow certain rules in order to allow WMS to correctly generate the GSSHA channel input file. In WMS 7.0 the stream arcs no longer need to be spatially manipulated. However, the streambed profile still needs to be checked and corrected using the *Smooth GSSHA Streams* dialog.

Stream Smoothing

An essential part of building the stream network is making sure that the streambed will flow as it should. Obtaining the stream network from a DEM leaves many artificial ridges and pits in the streambed profile that force water to pond. Often when the stream network is brought in from a shapefile the streams have been digitized from a topographical map and the streambed is lacking elevation values. The place to check for and solve these problems is in the *Smooth GSSHA Streams* dialog.

The *Smooth GSSHA Streams* dialog shows a profile of the bed elevations, allowing the individual vertices along the arc to be manipulated. There are five means to manipulate the streambed elevations in the *Smooth GSSHA Streams* dialog. The first way to manipulate the streambed elevations is by selecting a vertex (using the select vertex tool) and then adjusting the streambed elevation value in the stream elevation edit field. The second means of modifying the streambed elevations also uses the select vertex tool; the select vertex tool can also drag the selected vertex along a vertical line to adjust the elevation. Third, the **Offset Stream Elevations By Constant** button will ask for a value by which to offset all of the streambed elevations. Positive offset values are downward. The *Smooth GSSHA Streams* dialog will not allow the surface elevations to be higher than the cell surface elevations. The cell surface elevations are shown in a step-wise fashion along with the streambed profile. If the cell elevations are not visible the most likely cause is that the streambed elevations are too close to the cell surface elevations so that the points and lines representing the streambed profile are drawn on top of the cell surface elevations.

The two smoothing options are the principal means of modifying the streambed profile to allow water to flow, as it naturally should. The fourth method of modifying the streambed profile is by using the **Smooth Stream Elevations To Smoothed Grid** button. When this button is pushed a warning comes up explaining that the stream arc is about to be dramatically altered. What will occur is that the stream arc will be conformed to lie on the surface of the grid. To accommodate the interpolated smoothness of the grid many extra vertices will be added to the arc at locations where the grid changes slope. All of these extra vertices make manually manipulating the stream arc much more difficult and hence the reason for the warning message. The arc will not change spatially; the new vertex (x,y) locations are linearly interpolated from the existing vertices.

The final method of modifying the streambed elevation, the **Interpolate Stream Elevations** button, was implemented in WMS 6.1 and is a modified form of the stream-smoothing algorithm proposed by Ogden, Saghafian, and Krajewski. [Ogden et. al 1994] The modification constrains junction nodes to be lower than the next upstream vertex on the offshoot branch. The yellow dots on the streambed profile are these elevations, the elevations of the next upstream vertex of each stream arc attached to the stream arcs being displayed.

Stream Arc Topology Rules

There are still several topological rules for stream arc connectivity that must be adhered to in order produce a valid channel input file. The stream arcs must still be connected from downstream to upstream, and there should not be any braids or multiple downstream channels from an upstream reach. New for GSSHA 2.0 is that there may be more than two upstream reaches from any particular reach (e.g. three reaches may converge to become a single reach).

Related Topics

- Embankment Arcs
- GSSHA Feature Arcs

GSSHAWiki [2]

Primer [3]

- Watershed Delinieation and Grid Construction [1]
- Lake and Channel Routing [1]

User's Manual [5]

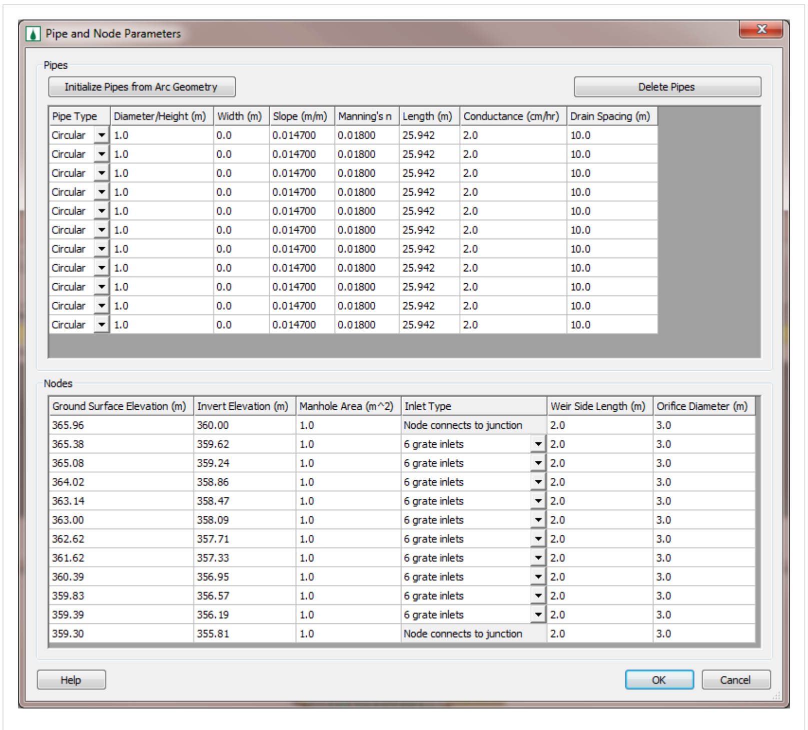
- Surface Water Routing [2]

Tutorials [7]

Pipe and Node Parameters

Overview

The *GSSHA Pipe and Node Parameters* dialog is accessed by selecting one or more feature arcs, going to the *GSSHA Feature Arc Properties* dialog, and changing the arc type to Pipe. This dialog is used for building and editing the pipes and nodes associated with Super-Links when running a GSSHA Storm and Tile drain simulation.

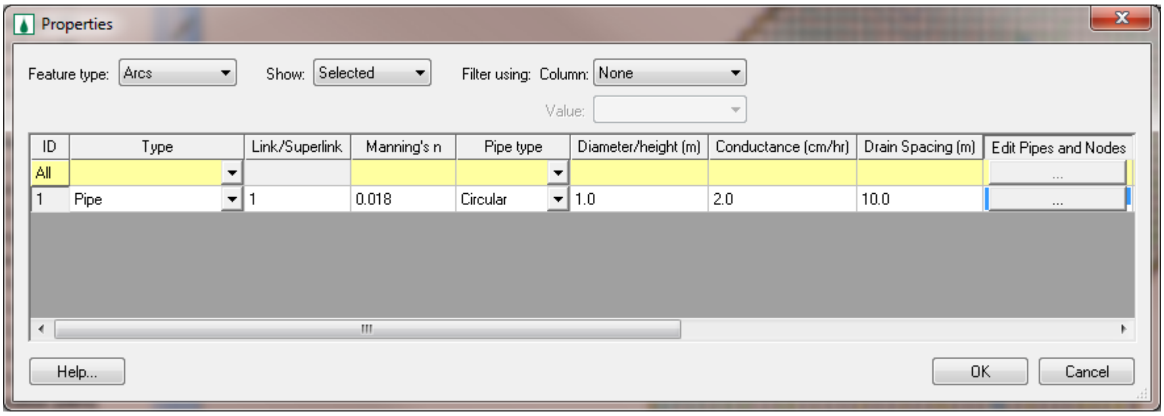


Help

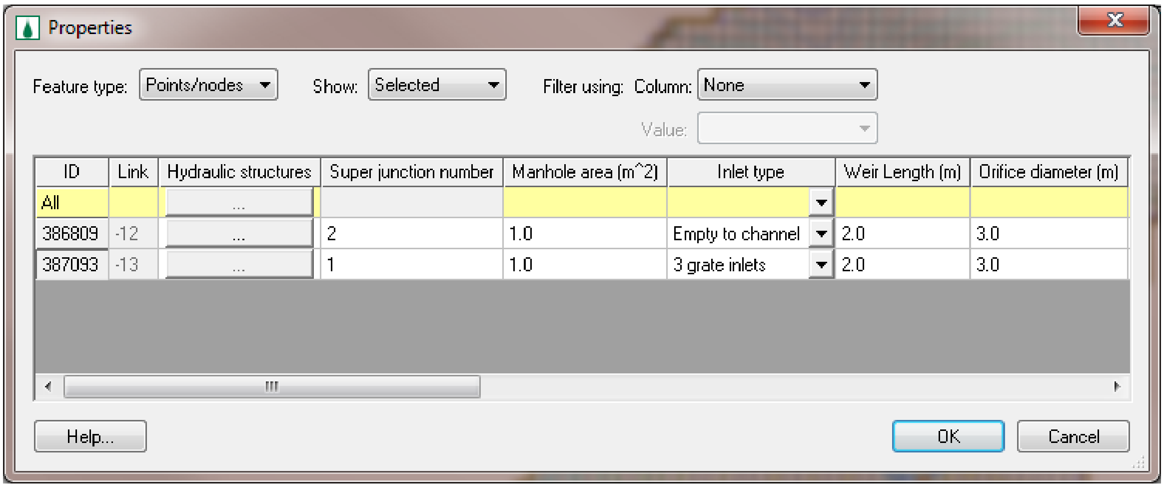
When running a GSSHA Storm and Tile drain simulation, it is a good idea to add more than one pipe (2 nodes) between Super-Junctions. In WMS, Super-Links are represented as arcs and Super-Junctions are represented by the 2 nodes at the end of each arc. Each link may contain 1 or more pipes and 2 or more nodes. Many of the attributes between Super-Junctions and nodes between Super-Junctions overlap. In WMS, you define nodes along a

Super-Link (between Super-Junctions) by adding vertices along an arc. You can redistribute the vertices on the arc using the Redistribute command or you can manually add vertices along an arc using the *Create Vertex* tool. Normally, you would follow the following steps to define storm or tile drains and their attributes in WMS:

- Define your storm or tile drain geometry using arcs.
- Define the storm or tile drain attributes for each arc (Super-Link) using the *GSSHA arc properties* dialog.



- Define the storm or tile drain attributes and pipe invert elevations for the nodes at each end of the arcs (Super-Junctions) using the *GSSHA point/node properties* dialog.



- For each arc (Super-Link), redistribute the vertices along the arc using the redistribute vertices command and/or manually add vertices along each arc. The spacing and number of vertices on an arc define the pipes and nodes on the arc and the lengths and slopes of the pipes.
- Define the attributes of the pipes and nodes for each arc (Super-Link) by going to the *GSSHA arc Properties* dialog and clicking on the **Edit Pipes and Nodes** button.
- From the *Pipe and Node Parameters* dialog, you can Initialize your Pipes from the Arc Geometry. Clicking on this button creates a pipe for each arc segment on your selected arc and creates a node for each node or vertex on the arc. Attributes associated with the Super-Link (arc) and Super-Junctions (arc nodes) are transferred to the generated pipes and nodes, but you are free to edit these transferred data values. Node ground surface elevations are extracted from the 2D grid elevations at the location of each node or vertex on the selected arc. These pipes and nodes are written to the GSSHA Storm Pipe Network (*.spn) file when the GSSHA project is written. It is important to define pipe and node parameters for all the pipe arcs in your storm or tile drain network. Deleting the pipes will delete the pipes and nodes from your arc, but you should re-initialize the pipes from the arc geometry after making any changes that need to be made or WMS may not write the correct pipe and node attributes for the selected arc.

Refer to the GSSHA Subsurface Tile and Storm Drain tutorial for more information about how to setup a storm or tile drain model.

Radar Rainfall

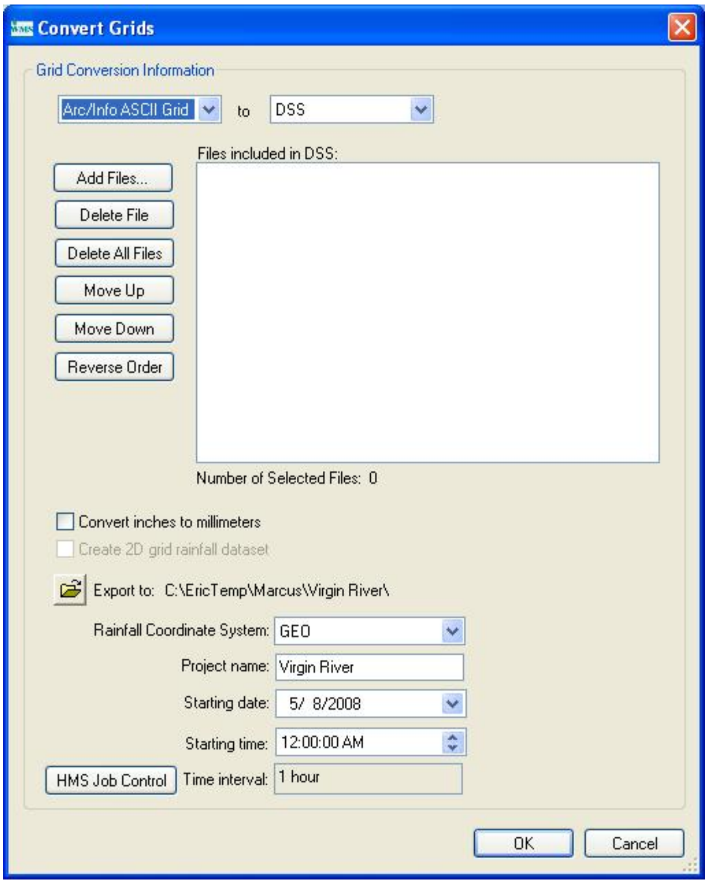
Overview

WMS 8.1 and later allows you to read and utilize radar rainfall data from the National Weather Service. You can use data with either the HMS or GSSHA hydrologic models.

Radar Rainfall with HMS

Steps to use Radar Rainfall with HMS

To use radar rainfall data with HMS, create an HMS model as you normally would. You can use the hydrologic modeling wizard. After you have created your model, define your precipitation in the HEC-HMS Meteorological Parameters dialog. Once you are here, you will need to do the following steps.



HMS Radar Rainfall Interface

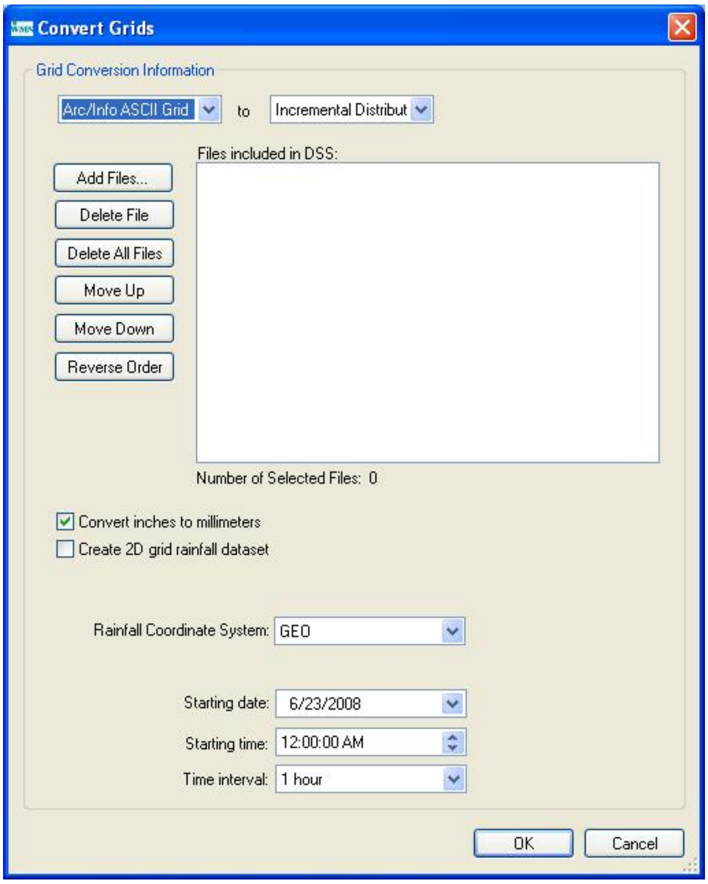
1. Change precipitation method to either *User Gage Weighting* or *Gridded Precipitation*. "User Gage Weighting" will only allow you to create rain gages from the radar data, while "Gridded Precipitation" will allow you to create gridded precipitation in a DSS file as well as creating rain gages.
    1. To use "Gridded Precipitation", you must have a 2-D Grid defined.
2. Click on the button, which depending on your selection will be entitled *Radar Data->Rain Gages...* or *Convert ASCII or XMRG files to DSS...*. This will bring up the Convert Grids dialog.
3. Obtain your rainfall data. You can follow the instructions on the GSDA site to obtain Nexrad Arc/Info ASCII grids.
4. Select the datatype you want to convert your data to: *Incremental Distribution Rain Gages*, *Total Storm Rain Gages*, or *DSS*.
5. Click *Add Files..* and select your radar precipitation files.
    1. If you select the last file first, and select the first file **while holding shift** the data will come into WMS in the proper order.
    2. It is very important that you arrange the data in chronological order! Once you have brought the data into WMS, you can arrange the data's order through the use of the *Move Up*, *Move Down*, and *Reverse Order*.
    3. If you are using the suggested data from the NCDC website (link available at the GSDA site), then the filename has the date and time. The last numbers of the filename is the year, month, day, hour, and minute that the data was collected. For more information, please see NCDC Radar Frequently Asked Questions (FAQ) [1].

4. Note that the dialog can be resized to view the entire filename.

6. If you are creating a DSS file, select the folder you wish to export the DSS file by clicking on the select folder button. You should also change the *Project name*.
7. Select the starting date and time of the precipitation grids you are using.
8. You can change the time interval of the rainfall grids by clicking on the *HMS Job Control* and changing the project's *Time interval*. The Rainfall timestep should match the project's timestep.
9. If you wish to view the data in milimeters rather than inches, check the *Convert inches to millimeters*. If you are creating rain gages, then you may also create 2D grid rainfall dataset. This will allow you to visualize your data, but will take additional computational time. Selecting this option will also create a *NEXRAD RADAR DATA SUMMARY REPORT* with a basin average hyetograph.
10. Click *OK* and wait for your computer to finish processing the radar rainfall data.
11. Click *OK* to exit the *HMS Meteorological Model* dialog.

Radar Rainfall with GSSHA

Steps to use Radar Rainfall with GSSHA



GSSHA Radar Rainfall Interface

To use radar rainfall data with GSSHA, create an GSSHA model as you normally would. You can use the hydrologic modeling wizard. After you have created your model, define your precipitation in the *GSSHA precipitation* dialog. Once here, you will do the following steps to define Radar precipitation:

1. Change your precipitation type to *Nexrad Radar*
2. Select *Import Radar Data...*
3. Obtain your rainfall data by following the instructions on the GSDA site
4. Select the *Add Files...* button
5. Select the datatype you want to convert your data to: *Incremental Distribution Rain Gages* or *Total Storm Rain Gages*.
   1. Incremental Distribution Rain Gages corresponds to a GSSHA *GAGE* type rain gage.
   2. Total Storm Rain Gages corresponds to a GSSHA *ACCUM* type rain gage.
6. Click *Add Files..* and select your radar precipitation files.
   1. If you select the last file first, and select the first file **while holding shift** the data will come into WMS in the proper order.
   2. It is very important that you arrange the data in chronological order! Once you have brought the data into WMS, you can arrange the data's order through the use of the *Move Up*, *Move Down*, and *Reverse Order*.

   3. If you are using the suggested data from the NCDC website (link available at the GSDA site), then the filename has the date and time. The last numbers of the filename is the year, month, day, hour, and minute that the data was collected. For more information, please see NCDC Radar Frequently Asked Questions (FAQ) [1].
   4. Note that the dialog can be resized to view the entire filename.
7. Select the starting date and time of the precipitation grids you are using.
8. You can change the time interval of the rainfall grids by clicking on the *HMS Job Control* and changing the project's *Time interval*. The Rainfall timestep should match the project's timestep.
9. You should leave the toggle box *Convert inches to millimeters* checked. GSSHA will be expecting the precipitation in millimeters.
10. If you wish to view the data as a 2D grid rainfall dataset, check the toggle box. This will allow you to visualize your data, but will take additional computational time. Selecting this option will also create a *NEXRAD RADAR DATA SUMMARY REPORT* with a basin average hyetograph.
11. Click *OK* and wait for your computer to finish processing the radar rainfall data.
12. Click *OK* to exit the *HMS Meteorological Model* dialog.

What happens when I click OK in the Convert Grids dialog

This depends if you are converting to rain gages or Gridded Precipitation (including DSS). The following explains what occurs for each step of the process.

Rain Gages

1. Read each radar file: This step will also convert the data from inches to millimeters, if that option is selected.
2. Prepare Data: Sometimes the user will select data that stretches beyond one storm. The radar precipitation grids are cumulative, but only for the storm duration. After the defined storm has passed, the grids reset to zero. This step will make the grids cumulative for the time period selected by the user. Please note that it takes at least 0.5% of the grid to be less than the previous grid to be counted as a reset. WMS will not be tricked by a few incorrect values in the grid.
3. Create Initial Grid: If the user does not select the first timestep of the storm, the first grid will contain precipitation that did not occur during the time period selected by the user. This step creates a copy of the initial values of the data selected by the user.
4. Remove Initial Grid: This step removes the precipitation from all the timesteps that occurred prior to the first time step selected by the user.
5. Converting grid to features: This step finds which points lie within the basin, and then creates a rain gage at that location. This step does not read any values for these gages.
6. Assign time series Data: This step assigns values to each rain gage.

Gridded Precipitation (including DSS)

1. Read each radar file: This step will also convert the data from inches to millimeters, if that option is selected.
2. Prepare Data: Sometimes the user will select data that stretches beyond one storm. The radar precipitation grids are cumulative, but only for the storm duration. After the defined storm has passed, the grids reset to zero. This step will make the grids cumulative for the time period selected by the user. Please note that it takes at least 10% of the grid to be less than the previous grid to be counted as a reset. WMS will not be tricked by a few incorrect values in the grid.
3. Create Initial Grid: If the user does not select the first timestep of the storm, the first grid will contain precipitation that did not occur during the time period selected by the user. This step creates a copy of the initial values of the data selected by the user.

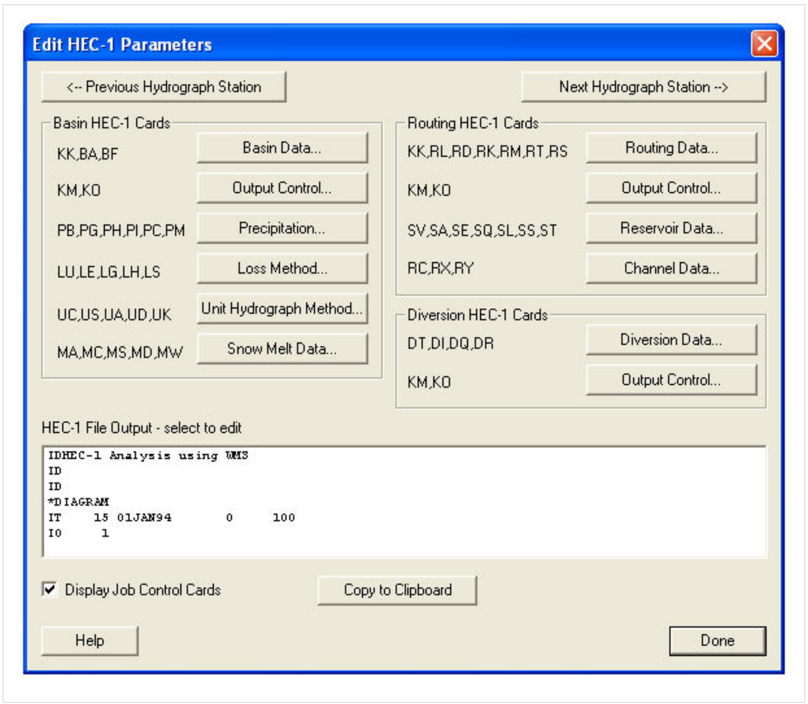
4. Remove Initial Grid: This step removes the precipitation from all the timesteps that occurred prior to the first time step selected by the user.
5. Create 2D Scatter Set: This step finds which points lie within the basin, and then creates a 2D Scatter point at that location. This step does not read any values for these points.
6. Assign 2D Scatter Set Data Values: This step assigns values to each 2D Scatter point for every time step.
7. Interpolate *#####.####* timestep: This step will convert the 2D Scatter Set to the 2D Grid. The process uses 2D scatter points to preserve accuracy to convert the precipitation grids because converting coordinate systems will change grid size and shape. Converting coordinate system through scatter points preserves the location and value of the measured precipitation. The most accurate way to convert this data back to a grid is through interpolation.
8. Creating Incremental Dataset, in three steps: This step will convert the current cumulative rainfall dataset to an incremental dataset.
9. Write 2D Grid dataset as ASCII Files for each time step: This step is necessary to create a DSS file. The 2D grid Rainfall dataset is exported to a 2D Arc/Info ASCII Grid at the same location as the DSS file. Each time step is exported separately. This step will create a command prompt for each time step.
10. Convert ASCII Grids to DSS file: This will take each grid exported in the last step and create a DSS file.

References

[1] http://www.ncdc.noaa.gov/nexradinv/faq.html#FORMAT

6.3. Hydrologic Engineering Center-1 (HEC-1)

HEC-1 Parameters



Attributes or parameters for all HEC-1 hydrograph stations are defined and/or later edited using the *Edit HEC-1 Parameters* dialog. This dialog is accessed by selecting the **Edit HEC-1 Parameters** command from the *HEC-1* menu or by double-clicking on basin, outlet, or diversion icon from the *Graphics Window*.

If a basin, outlet, or diversion is selected before issuing the command then data for that object appears in the text window at the bottom of the dialog. The top portion of the dialog lists the HEC-1 cards that can be edited by selecting the corresponding button. When a hydrograph station is selected (basins/outlets/diversions) only the buttons that edit parameters associated with that hydrograph station are active, all others are dimmed. In addition to using the appropriate button, HEC-1 attributes can be edited by clicking on the HEC-1 card in the text display window. Using this method, job control parameters can be edited by first toggling their display using the *Display Job Control Cards* toggle box, and then selecting a job control card from the text display window.

Once the dialog appears it becomes part of the main screen until you select the **Done** button. Therefore, you can continue to select additional, or other hydrograph stations so that data for that object may be edited without exiting the dialog. You can use the previous and next hydrograph station buttons to cycle through hydrograph stations in the order they are computed by HEC-1. Since the dialog is part of the main screen, all menu commands are active while this dialog is present.

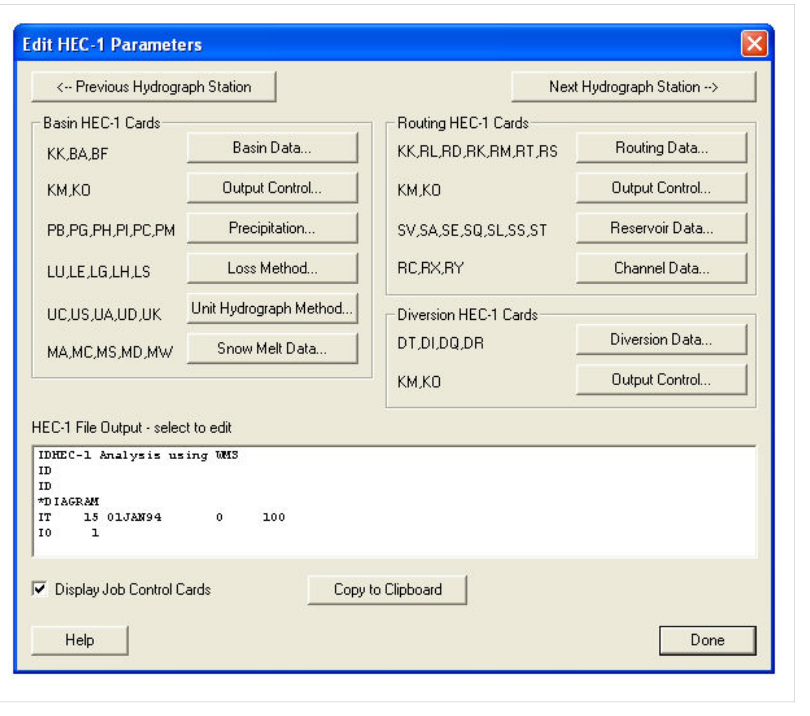
- Basin HEC-1 Cards
- Routing HEC-1 Cards
- Diversion HEC-1 Cards

Related Topics

- Job Control Data
- Model Check
- Running an HEC-1 Analysis

Basin HEC-1 Cards

Basin Data...



Basin Name (KK)

Each hydrograph station should be identified with a unique name. This name appears as part of the KK record for that station in the input file. The name should not be more than six characters long. By default WMS uses the basin ID number followed by a "B" for the name, but a descriptive name is generally more useful.

Basin Area (BA)

When a terrain model is used, basin areas and slopes can be computed automatically using the **Compute Basin Data** command from the *Drainage* menu of the TIN or DEM module, or the **Update Basin Data** command in the *Feature Objects* menu of the Map module. Otherwise, areas and slopes must be entered interactively using the topological tree as a map. Areas should be entered in either square miles or square kilometers.

Base Flow (BF)

Base flow parameters can be defined for a basin by selecting the *Enter base flow* check box. The input parameters for base flow are as follows:

- $STRTQ$ – Flow at the start of the storm in cfs (cms for metric units).
- $QRCSN$ – Flow in cfs (cms) below which base flow recession occurs in accordance with the recession constant $RTIOR$. In other words, it is that flow where the straight line (in semilog paper) recession deviates from the falling limb of the hydrograph.
- $RTIOR$ – The ratio of the recession flow $(QRCSN)$ to that flow occurring one hour later (Must be greater than or equal to 1).

Output Control...

For each hydrograph station (basin hydrographs, combined hydrographs, and routed hydrographs) different output controls can be specified. This dialog is accessed by selecting the *Output Control* dialog button from the *Edit HEC-1 Parameters* dialog. Entries which can be defined in this dialog are described below.

Routed and Combined Hydrographs at Outlets

In WMS an outlet point is used to represent locations where hydrographs are both combined and then routed. Therefore, if an outlet is selected before choosing the *Output Control* dialog, a radio group at the top of the dialog appears so that you may specify whether the options should be applied to the combined or routed hydrograph. If a basin is selected the radio group at the top of the dialog does not appear.

Comment Lines (KM)

Individual comments can be defined for each hydrograph station. These comments can be used to identify unique characteristics about a particular basin or outlet point. A new comment can be defined by selecting the new button in the *Output Control* dialog and then entering the comment in the text entry. When more than one comment card has been defined, the up and down arrow buttons can be used to scroll through the list of comments for that hydrograph station. When using WMS, comment cards always appear directly after the KK cards for each hydrograph station.

Output Control (KO)

These controls determine what information about a given hydrograph station is written to the HEC-1 ASCII output file. By default the IO record information is used. However, you may wish to print out a more (or less) complete summary for individual hydrograph stations.

By default, the option to write a hydrograph to the TAPE22 file is specified. This is the file read by WMS for display of hydrographs. Therefore, this option should only be changed to suppress particular hydrographs.

Precipitation...

Precipitation patterns are assigned to basins by first selecting the appropriate basin(s) and then clicking on the precipitation button in the *Edit HEC-1 Parameters dialog*. If multiple basins are selected then the defined parameters will apply to all selected basins.

> **NOTE:** If no basins are selected, the parameters can be applied to all basins.

HEC-1 No Precipitation

If no precipitation for a given basin is chosen, then the program will use the precipitation pattern of the most recently defined basin. In other words, if the same precipitation pattern is to be used for each basin, specify precipitation at the upper-most basin and let all other basins "inherit" this same pattern.

Basin Average Precip (PB)

With this method, a time distribution can be entered to create a PI or PC card. The distribution is entered via the XY Series Editor, refer to the chapter titled Using the XY Series Editor. Several standard storm distributions can be loaded automatically from this editor. In addition, distributions can be saved and later restored from a file. When creating PI or PC records, an IN record needs to be defined to specify the beginning time and date of the storm. WMS computes the values for this IN card based on the first time increment in the precipitation (PI or PC) record defined in the *XY Series Editor*. Because only one IN card defining the precipitation time increment is defined for each raifall distribution, **the precipitation time increments defined for a single rainfall series in the XY series editor must all be equal**. The time/date parameters entered on the IT card (in the HEC-1 Job Control dialog) are used for the start time/date of the precipitation.

Precipitation Gage (PG)

Gages can be used with or without a terrain model. If drainage basins have been defined using a TIN, the appropriate gage weights (using the Thiessen polygon method) for each basin are automatically computed when the **Compute (or Update) Basin Data** command is executed. If the HEC-1 model is defined using only the tree, or to change any of the computed values, the gage weights can be changed/assigned by clicking on the gage weights button in the *Precipitation* dialog. The gage weights dialog will display a list of all defined gages and their station type. Choose from this list when defining gage weights to the selected basin. Storm total stations are written on PT/PW records whereas recording stations are written on PR/PW records. You must have at least one PR/PW record combination for each basin.

When using a terrain model (DEM, TIN, or Feature Objects) a rain gage coverage can be defined and used to establish the positions of gages by using the graphical creation/selection tools available in the Map module.

Hypothetical Storm (PH)

A hypothetical storm may also be used to define the precipitation pattern for the runoff simulation. The XY Series Editor is used to define the necessary rainfall values for the appropriate times. The storm frequency in percent is entered in the frequency edit field. Rainfall will be converted to an annual-series for fifty, twenty, and ten percent storms. No conversion is made for any other frequency storms. A storm area to be used in computing reduction of point rainfall depths is entered in the area edit field. If 0 is entered for the area then the basin area (or area from JD card for depth/area storms) will be used as a default.

Probable Maximum Precipitation (PM)

Defining precipitation using the probable maximum precipitation option allows for the computation of the probable maximum storm according to the outdated Hydrometeorological Report No. 33 (HMR 33). This does use an outdated method and has been retained in HEC-1 for now in order to be able to reproduce results according to the old HMR 33 method.

The following variables must be defined:

- PMS – The probable maximum index precipitation from the HMR 33.
- $TRSPC$ – Precipitation adjustment (between 0 and 1.0) based on drainage area size. If this value is set at zero HEC-1 will default it to the appropriate value based on the HOP Brook Adjustment Factor as described in the HEC-1 manual.
- $TRSDA$ – The drainage area in square miles for which the storm is transposed.
- SWD – This value can be set to the EM 1110-2-1411 criteria or the Southwestern Division criteria.
- $R6, R12, R24, R48, R72, R96$ – Maximum precipitation at the specified hourly intervals as a percentage of the probable maximum storm. The $R48$, $R72$, and $R96$values are optional

Loss Method...

One of several different loss methods can be chosen when generating synthetic hydrographs. A loss method is assigned to a basin by first selecting the basin and then choosing the **Loss Method** button in the *Edit HEC-1 Parameters dialog*. As with other basin data the same parameters can be assigned to several basins by selecting multiple basins before accessing the *Loss Method* dialog.

When defining a kinematic wave model, it may be necessary to define a separate set of loss parameters for the two different UK records (generally corresponding to pervious and impervious area). This second set of loss parameters is defined from within the *Unit Hydrograph Method* dialog.

Uniform Loss Method (LU)

This loss method uses an initial value and a uniform value to define infiltration losses. Input parameters are as follows:

- $STRTL$ – Initial rainfall/snow melt loss in inches (mm) for snow free ground.
- $CNSTL$ – Uniform rainfall/ loss in inches/hour (mm/hour) which is used after the starting loss $(STRTL)$has been satisfied.
- $RTIMP$ – Percentage of drainage basin that is impervious.

Losses (LM)

Losses are used in conjunction with the uniform (LU) or exponential (LE) loss methods. The parameter descriptions are as follows:

- $STRKS$ – The starting value of the loss coefficient on the exponential recession curve for losses in in/hour (mm/hour) when used with the exponential loss rate (LE) or the uniform melt water loss rate (in/hour) when used with the uniform loss rate (LU).
- $RTIOK$ – Rate of change of the loss-rate parameter computed as the ratio of $STRKS$to a value of $STRKS$after ten inches of accumulated loss when used with the exponential loss rate or not used when using the uniform loss rate.

Exponential Loss (LE)

Parameters for the exponential loss method are as follows:

- STRKR – The starting value of the loss coefficient on the exponential recession curve for rain losses.
- DLTKR – The amount in inches (mm) of initial accumulated rain loss during which the loss coefficient is increased.
- RTIOL – Parameter computed as the ratio of STRKR to a value of STRKR after ten inches (ten mm) of accumulated loss.
- ERAIN – Exponent of precipitation for rain loss function that reflects the influence of the precipitation rate on basin-average loss characteristics.
- RTIMP – Percentage of drainage basin that is impervious.

Green & Ampt (LG)

Green-Ampt infiltration loss parameters are as follows:

- IA – Initial loss (abstraction) in inches (mm).
- DTHETA – Volumetric moisture deficit. If this value is 0, then the method reduces to the initial loss equal to IA and a constant loss equal to XKSAT.
- PSIF – Wetting front suction in inches (mm). If this value is 0, then the method reduces to the initial loss equal to IA and a constant loss equal to XKSAT.
- XKSAT – Hydraulic conductivity at natural saturation in inches/hour (mm/hour).
- RTIMP – Percentage of drainage basin that is impervious.

Using methods defined by the Maricopa County Flood Control District, Green-Ampt parameters can be determined from GIS data layers automatically in WMS.

Holtan (LH)

Parameters used to define the Holtan loss method:

- FC – Holtan's long term equilibrium loss rate in inches/hour (mm/hour) for rainfall/losses on snow free ground.
- GIA – Infiltration rate in inches/hour per inch $*BEXP$(mm/hour per mm </math>*BEXP</math>) of available soil moisture storage capacity.
- SAI – Initial depth in inches (mm) of pore space in the surface layer of the soil which is available for storage of infiltrated water.
- $BEXP$ – Exponent of available soil moisture storage.
- $RTIMP$ – Percentage of drainage basin that is impervious.

SCS Loss Method (LS)

The SCS curve number method uses the following parameters:

- $STRTL$ – Initial rainfall abstraction in inches (mm) for snowfree ground. If value is 0, then initial abstraction will be computed as $0.2 * (1000 - 10 * CRVNBR)/CRVNBR$.
- $CRVNBR$ – SCS curve number for rainfall/ losses on snowfree ground.

NOTE: Composite Curve Numbers can be computed automatically when this method for computing losses is chosen and a terrain model is present.

- $RTIMP$ – Percentage of drainage basin that is impervious.

Unit Hydrograph Method...

One of several different unit hydrograph methods can be chosen when generating synthetic hydrographs. A method is assigned to a basin by first selecting the basin and then choosing the **Unit Hydrograph Method** button from the *Edit HEC-1 Parameters* dialog. As with other basin options the same parameters can be assigned to several basins by selecting multiple basins before accessing the *Unit Hydrograph Method* dialog.

Clark Unit Hydrograph (UC)

The parameters for the Clark method are as follows:

- T_C – Time of concentration in hours for the unit hydrograph. Several different equations exist for determining the time of concentration. The list of basin geometric attributes computed automatically when basins have been delineated from a terrain model can be useful in many of these equations. These attributes can be viewed from within the *Unit Hydrograph Method* dialog by choosing the **View Basin Geometrical Attributes** button. Time of concentration can be computed from one of several equations using these attributes, or by using a time

computation coverage. These options are accessed from the Compute Parameters – Basin Data and Compute Parameters – Map Data buttons respectively.
- R – The Clark storage coefficient in hours.
- TIME AREA CURVE – The time area curve defines the area of the watershed contributing runoff to the basin outlet as a function of time. This curve is defined by selecting the check box and then activating the *XY Series Editor* with the adjacent button. The time area curve can be computed automatically from a TIN (this method will not work for watersheds delineated from DEMs or Feature Objects) using the **Compute Time Area Curves** button.

Snyder (US)

Parameters for the Snyder unit hydrograph are as follows:

- TP – Lag time in hours. Several different equations have been published to determine the lag time of a basin. Many of these use some of the geometric attributes computed automatically when a terrain model is present. These attributes can be viewed by choosing the **View Basin Geometrical Attributes** button. Lag time can be computed from one of several equations using these attributes, or by using a time computation coverage. These options are accessed from the Compute Parameters – Basin Data and Compute Parameters – Map Data buttons respectively (see Computing Travel Times).
- CP – Peaking coefficient.
- TIME AREA CURVE – The time area curve defines the area of the watershed contributing runoff to the basin outlet as a function of time. This curve is defined by selecting the check box and then activating the *XY Series Editor* with the adjacent button. The time area curve can be computed automatically from a TIN (this method will not work for watersheds delineated from DEMs or Feature Objects) using the **Compute Time Area Curves** button.

SCS Unit Hydrograph (UD)

Parameters for generating a unit hydrograph using the SCS dimensionless method include:

- TLAG – SCS lag time in hours. Several different equations have been published to determine the lag time of a basin. Many of these use some of the geometric attributes computed automatically when a TIN is present. Lag time can be computed from one of several equations using these attributes, or by using a time computation coverage. These options are accessed from the Compute Parameters – Basin Data and Compute Parameters – Map Data buttons respectively.

Kinematic Wave (UK)

Distributed outflow from a basin may be obtained by utilizing combinations of three conceptual elements: overland flow planes, collector channels, and a main channel. These elements can be defined if the kinematic wave option is specified.

The first and second kinematic wave records can be used to distinguish between different properties such as pervious/impervious (grass/pavement). For each record, the following parameters can be supplied.

- L – Overland flow length.
- S – Representative slope.
- N – Manning's roughness coefficient.
- A – Percentage of sub-basins area that this record represents (The total of the two records must sum to 100).

Losses – A loss method must be defined for each plane. Choosing the **Define Loss** button will present the *Standard Loss Method* dialog and allow a method to be chosen and parameters defined.

In addition to the kinematic wave records, collector channels and a main channel must be defined. Either kinematic wave (RK) or Muskingum-Cunge (RD) routing can be specified by selecting the appropriate radio button. A dialog for defining the channels is accessed by choosing the **Define Channels** button. The main channel must be defined, whereas the two collector channels are optional. The following parameters are used for each channel:

- L – Channel length.
- S – Channel slope.
- N – Manning's roughness coefficient for the channel.
- CA – Contributing area to the channel.
- $SHAPE$ – The characteristic shape of the channel.
- WD – Channel bottom width or diameter.
- Z – Side slopes if the channel type requires it.

For the main channel, only an eight point cross section as defined with the RC, RX, RY cards can be used.

A flag for routing upstream hydrographs can be specified for the main channel from within this dialog as well.

Derived Unit Hydrograph (UI)

A given unit hydrograph determined from a separate analysis can be input using the XY Series Editor. The given unit hydrograph must be derived for the same time interval as is specified on the IT record in the *Job Control* dialog.

Snow Melt Data...

When snow needs to be considered in the runoff analysis, snow melt data for a basin needs to be defined. HEC-1 has two different methods for computing snowfall/melt simulations: the Degree-Day method, and the Energy-Budget.

To define data for a selected basin, choose the **Snow Melt Data** button from the *Edit HEC-1 Parameters* dialog. The toggle at the top of the dialog turns snow calculations on. For both methods the elevation or zone data, the coefficients, and temperature data must be defined.

The Degree-Day method is set up once these parameters have been defined. If the Energy-Budget method is toggled on then the Dew point, Short-wave radiation, and Wind speed data must be defined as well. Losses should be defined when either method is used. These losses are used in conjunction with the LU or LE cards for normal basin losses. The check box at the bottom of the dialog allows losses to be turned on or off for a given simulation.

Elevation Zone Data (MA)

Snow computations are accomplished in HEC-1 using separate, equally incremented, elevation zones within each basin. The number of elevation zones for which data must be defined is determined by specifying the base elevation of zone 1 and zone interval in the appropriate edit fields. The default values correspond to the lowest elevation and the range between the highest and lowest elevation (i.e. one elevation zone). More zones can be created by decreasing the interval, or lower elevations can be excluded from calculations by increasing the base elevation. Once the base elevation and interval are set, elevation zone data is defined by choosing the **Define MA Data** button. The number of zones which need to be defined is automatically determined and the appropriate edit fields are unhighlighted. The elevation zone parameters are as follows:

- AREA – The drainage area associated with this elevation zone.
- SNOPACK – The snow pack depth.
- AVEPRECIP – The normal annual precipitation in inches (mm) for this zone.

Areas for the elevation zones can be computed and supplied automatically using the **Compute Areas** button. The elevation fields are not part of the HEC-1 input.

Radiation

These three data records (dew point, radiation and wind speed) are only defined for the Energy-Budget method. Like the temperature time series, these three HEC-1 records are defined using the XY Series Editor. Dates for IN records can be specified using the appropriate edit fields and the beginning time and time increment are defined using the *XY Options* dialog from within the *XY Series Editor*.

Temperature (MT)

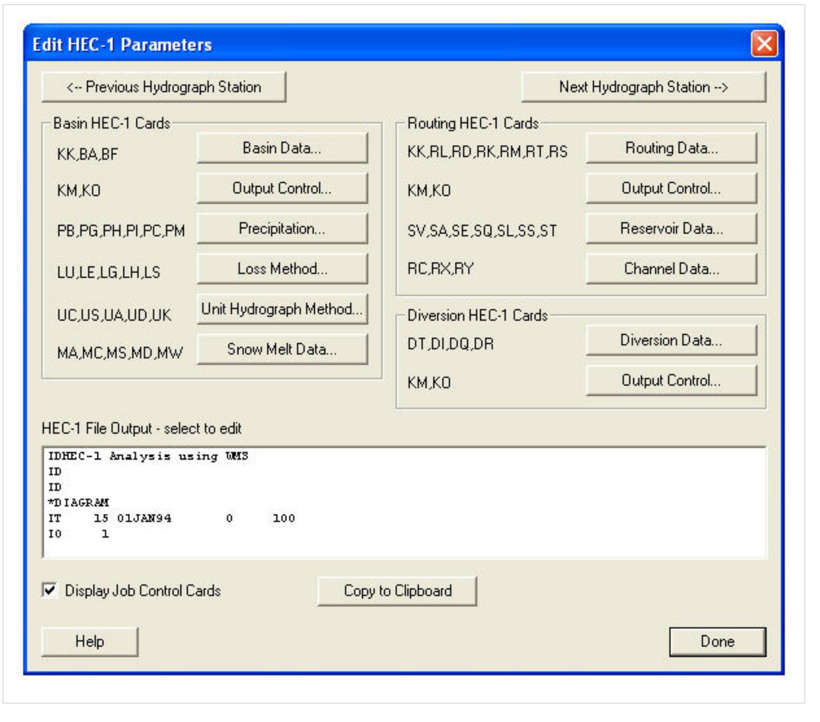
The temperature time series is entered using the *XY Series Editor* where each value corresponds to the air temperature at the bottom of the lowest elevation zone for that interval. The starting date is determined from the IN record values in the edit fields corresponding to the temperature data. The starting time and time increment (also part of an IN record) are specified in the XY Series Editor using the *XY Options* dialog.

Related Topics

- HEC-1 Parameters
- Computing Composite CN
- Computing Travel Times
- Computing the Area Between Contours
- XY Series Editor
- Obtaining Precipitation Data

Routing HEC-1 Cards

Routing Data



Outlet points are used to define locations where hydrographs are combined and then routed downstream. The appropriate combined hydrograph (HC cards) stations are generated automatically when writing a HEC-1 file. However, routing data must be entered in order to simulate the movement of a flood wave through the river reaches or reservoirs. The effects of storage and flow resistance are accounted for in the shape and timing of the flood wave. In addition to these changes, volume may be lost due to channel infiltration. Most of the routing methods available in HEC-1 are based on the continuity equation and some relationship between flow and storage or stage.

Routing data is entered by selecting an outlet and then selecting the **Routing Data** button from the *Edit HEC-1 Parameters* dialog.

Outlet Names (KK)

Since outlets are used for both types (combining and routing) of hydrograph stations in the HEC-1 input file, a separate name for each type of hydrograph must be entered. The name should be six characters or less and correspond to the name used on the KK card to represent the appropriate hydrograph station. By default WMS uses the basin ID number followed by a "R" for the name, but a descriptive name is generally more useful.

Channel Losses (RL)

Constant channel losses may be defined by defining values for the RL record. These parameters include the following:

- $QLOSS$ – Constant channel loss in entire routing in $cfs(cms)$. This value is subtracted from every ordinate of the inflow hydrograph.
- $CLOSS$ – Ratio of remaining flow (after $QLOSS$) which is lost for entire routing. After subtracting $QLOSS$each inflow hydrograph ordinate is multiplied by $(1 - CLOSS)$.
- $PERCRT$ – Percolation rate $cfs/acre(cum/sec - acre)$for wetted surface area of channel. This option is used in conjunction with storage routing and requires SA or SV/SE records to be defined.
- $ELVINV$ – Average invert elevation of channel Lused to compute flow surface area for $PERCRT$.

Kinematic Wave (RK), Muskingum Cunge (RD) and Convex (RV)

The Kinematic wave, Muskingum Cunge, and Convex are defined with essentially the same parameters. A brief description follows.

- L – Channel length.
- S – Channel slope.
- N – Manning's roughness.
- $Shape$ – Characteristic channel shape.
- WD – Base width of the channel.
- Z – Side slope of channel.

If the Muskingum-Cunge method is selected, define the channel geometry using an eight point cross-section by specifying the appropriate radio button and selecting the **Define RC Record** button.

NSTPS

NSTPS is the number of routing sub-reaches used to route the hydrograph in the Muskingum (RM), Straddler Stagger (RT), and Storage (RS) routing methods. To insure the Muskingum method's computational stability and the accuracy of computed hydrograph, the routing reach should be chosen so that:

$$\frac{1}{2(1-X)} \leq \frac{AMSKK}{NSTPS \times \Delta t} \leq \frac{1}{2X}$$

Where $X$is the Muskingum weighting factor $(0 \leq X \leq 0.5)$, $AMSKK$is the travel time through the reach in minutes (multiply by 60), and $\Delta t$is the computation time interval defined in the *HEC-1 Job Control* dialog in minutes.

For storage routing, $NSTPS$is usually about equal to (reach length / average velocity) / Δt, where $\Delta t$is described above. Set $NSTPS$to 1 for a reservoir.

Muskingum (RM)

The Muskingum method is dependent primarily upon an input weighting factor. The parameters along with short descriptions of their meanings follow:

- $NSTPS$ – The number of integer steps for the Muskingum routing.
- $AMSKK$ – Muskingum Kcoefficient in hours for entire reach.
- X – Muskingum $x$coefficient.

Using the basin data computed by WMS when a TIN or DEM is used to delineate the watershed, the $AMSKK$ and $NSTPS$coefficients can easily be estimated. $AMSKK$is essentially the travel time for the reach, which can be estimated by noting the length of the stream segment (displayed in the *Muskingum Cunge* dialog even though it is dimmed) and multiplying by an assumed channel velocity (1-5 ft/s would be appropriate for most natural channels). Of course you will need to convert the estimated travel times from seconds to hours before entering it into the *AMSKK edit* field. The $NSTPS$value is the number of time steps the flood wave is in the channel and can be determined by dividing $AMSKK$by the computational time step found in the Job Control dialog (again, be sure that units are consistent).

The Muskingum method computes outflow from a reach using the following equation:

$$Q_{OUT}(2) = (CA - CB) \times Q_{IN}(1) + (1 - C_A) \times Q_{OUT}(1) + CB \times QIN(2)$$
$$CA = \frac{2 \times \Delta t}{2 \times AMSKK \times (1 - X) + \Delta t}$$
$$CB = \frac{\Delta t - 2 \times AMSKK \times X}{2 \times AMSKK \times (1 - X) + \Delta t}$$

where $QIN$is the inflow to the routing reach in $cfs(m^3/sec)$, $QOUT$is the outflow from the routing reach in $cfs(m^3/sec)$, $AMSKK$is the travel time through the reach in hours, and $X$is the Muskingum weighting factor $(0 \leq X \leq 0.5)$. The routing procedure may be repeated for several subreaches (designated as $NSTPS$) so the total travel time through the reach is $AMSKK$.

Straddler Stagger (RT)

Parameters used to define the Straddler/Stagger or Tatum routing method are defined below.

- $NSTPS$ – Should be one for Straddler/Stagger method or integer number of routing steps to be used for Tatum method.
- $NSTDL$ – Integer number of intervals hydrograph is to be lagged in the Straddler/Stagger method or 0 if using Tatum method.
- LAG – Integer number of ordinates to be averaged in the Straddler/Stagger method or 2 if using Tatum method.

Storage Routing (RS)

Storage-discharge routing can be used to define either channel or reservoir routing. When this routing option is specified the appropriate data items are dimmed and additional radio buttons are used to determine whether channel or reservoir routing is to be used.

The following parameters must be defined regardless of the storage routing option specified.

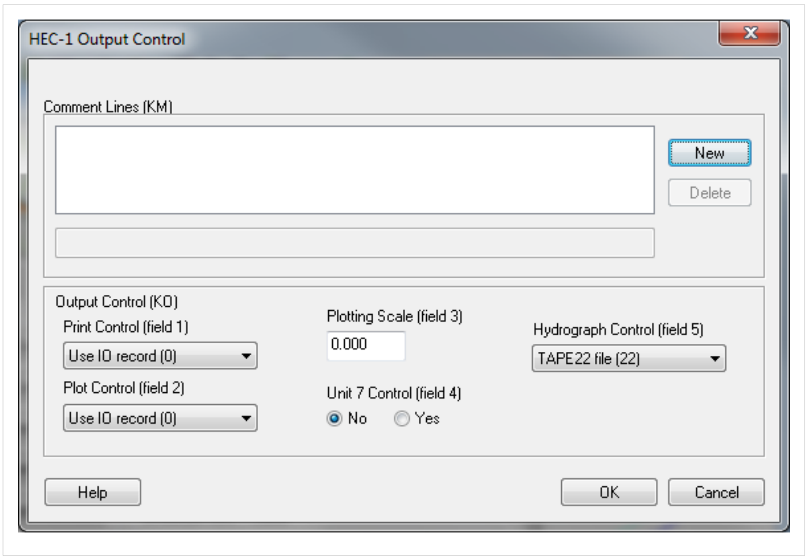
- $NSTPS$ – Number of steps to be used in the storage routing. Typically this is approximately equal to $(reachlength)/(averagevelocity * timeinterval(NMIN))$. $NSTPS$is usually equal to 1 for a reservoir.
- $ITYP$ – The next parameter $RSVRIC$can be entered in one of three different ways:
  - $STOR$ – Storage in acre-feet (1000 cu m).
  - $FLOW$ – Discharge in cfs (cms).
  - $ELEV$ – Elevation in feet (m).

- $RSVRIC$ – Storage, discharge, or elevation as defined by $ITYP$corresponding to the desired starting condition at the beginning of the first time period as specified on the IT record in the *Job Control* dialog.

HEC-1 No Routing (RN)

By default there is no routing at an outlet point. This allows for hydrographs to be combined without considering routing effects.

Output Control



For each hydrograph station (basin hydrographs, combined hydrographs, and routed hydrographs) different output controls can be specified. This dialog is accessed by selecting selecting the outlet, basin, reservoir, or diversion for which you want to edit data and then the **Output Control** dialog button from the *Edit HEC-1 Parameters* dialog. Entries which can be defined in this dialog are described below.

Routed and Combined Hydrographs at Outlets

In WMS an outlet point is used to represent locations where hydrographs are both combined and then routed. Therefore, if an outlet is selected before choosing the *Output Control* dialog, a radio group at the top of the dialog appears so that you may specify whether the options should be applied to the combined or routed hydrograph. If a basin is selected the radio group at the top of the dialog does not appear.

Comment Lines (KM)

Individual comments can be defined for each hydrograph station. These comments can be used to identify unique characteristics about a particular basin or outlet point. A new comment can be defined by selecting the new button in the *Output Control* dialog and then entering the comment in the text entry. When more than one comment card has been defined, the up and down arrow buttons can be used to scroll through the list of comments for that hydrograph station. When using WMS, comment cards always appear directly after the KK cards for each hydrograph station.

Output Control (KO)

These controls determine what information about a given hydrograph station is written to the HEC-1 ASCII output file. By default the IO record information is used. However, you may wish to print out a more (or less) complete summary for individual hydrograph stations.

By default, the option to write a hydrograph to the TAPE22 file is specified. This is the file read by WMS for display of hydrographs. Therefore, this option should only be changed to suppress particular hydrographs.

Reservoir Data

Reservoirs in HEC-1 can be defined in a few different ways, depending on the storage routing techniques that need to be modeled. The tutorial on creating topologic trees outlines the different methods that can be used to represent reservoirs. The parameters required to define the reservoir are the same in all cases. The main difference is whether the reservoir stands alone by itself or whether the routing option of the outlet is used to define the reservoir.

Reservoir Routing

If the reservoir routing option is specified then one method for volume and one method for outflow must be defined.

- Known volume – Define a known volume (SV) record using the XY Series Editor. Optionally, you can define the elevations (SE) which correspond to the known volumes.
- Computed volume – By defining an area (SA) elevation (SE) relationship the volume can be computed automatically by HEC-1. Both * records are defined using the XY Series Editor.

If a TIN has been used to define the watershed a storage capacity curve can be generated and the information used to set up the volume/elevation (SV, SE) or area/elevation (SA, SE) data. See Storage Capacity Curves for more information.

The available methods for outflow include:

- Known outflow – Define a known outflow (SQ) record using the XY Series Editor.
- Computed Weir Spillway – A combination of data records are used to define spillway characteristics (SL, SS, ST). Parameter description for these different records are as follows:

Low-Level Outlet (SL)

- $ELEVL$ – Center line elevation of downstream end of low-level outlet.
- $CAREA$ – Cross-sectional area in square feet (square m) in the low-level outlet orifice equation.
- $COQL$ – Discharge coefficient in orifice outlet equation.
- $EXPL$ – Exponent of head in orifice equation.

Spillway Characteristics (SS)

- $CREL$ – Spillway crest elevation. This value must be less than the highest elevation on the SE card for HEC-1 to run properly.
- $SPWID$ – Spillway length.
- $COQW$ – Discharge coefficient in the spillway weir flow equation.
- $EXPW$ – Exponent of head in the weir spillway flow equation, usually equals 1.5.

Dam Overtopping (ST)

- $TOPEL$ – Elevation of the top of the dam at which overtopping begins.
- $DAMWID$ – Length of the top-of-dam which is actively being overtopped.
- $COQD$ – Discharge coefficient in the weir equation.
- $EXPD$ – Exponent of head in the weir equation.

Channel Data

Constant channel losses may be defined by defining values for the RL record. These parameters include the following:

- $QLOSS$ – Constant channel loss in entire routing in cfs (cms). This value is subtracted from every ordinate of the inflow hydrograph.

- $CLOSS$ – Ratio of remaining flow (after $QLOSS$) which is lost for entire routing. After subtracting $QLOSS$each inflow hydrograph ordinate is multiplied by $(1 - CLOSS)$.
- $PERCRT$ – Percolation rate cfs/acre (cu m/sec-acre) for wetted surface area of channel. This option is used in conjunction with storage routing and requires SA or SV/SE records to be defined.
- $ELVINV$ – Average invert elevation of channel L used to compute flow surface area for $PERCRT$.

Channel Routing

If the channel routing option is specified, the Define button will access a dialog which allows you to choose between Normal depth and Modified-Puls methods. If Normal depth is specified, the following parameters must be defined for the RC record.

- Manning's coefficients – Manning roughness coefficients for the channel and left and right overbanks.
- Length – The length of the river reach.
- Slope – The slope of the river reach.
- Max Elevation – The maximum elevation for which storage and outflow values are to be computed.

In addition to these parameters an eight point cross section must be defined using the XY Series Editor. The first two points define the left overbank, the third point defines the left bank, the fourth and fifth points define the channel itself, the sixth point defines the right bank, and the last two points define the right overbank.

If the modified-Puls method is chosen the volume (SV) outflow (SQ) relationship must be defined. Both records are defined using the XY Series Editor.

Hydrographs

Direct Input Hydrograph (QI)

Hydrographs can be input directly using the QI record, and then routed downstream using the different routing options. To do this, select the *Direct Input Hydrograph* option and define the QI record using the XY Series Editor.

Pattern Hydrograph (QP)

This option is used to input a pattern hydrograph for an optimization job (OR record). A QP hydrograph can be used in conjunction with a QI and QO hydrograph to optimize routing parameters. This hydrograph can be input by selecting the check box and then defining the hydrograph using the XY Series Editor.

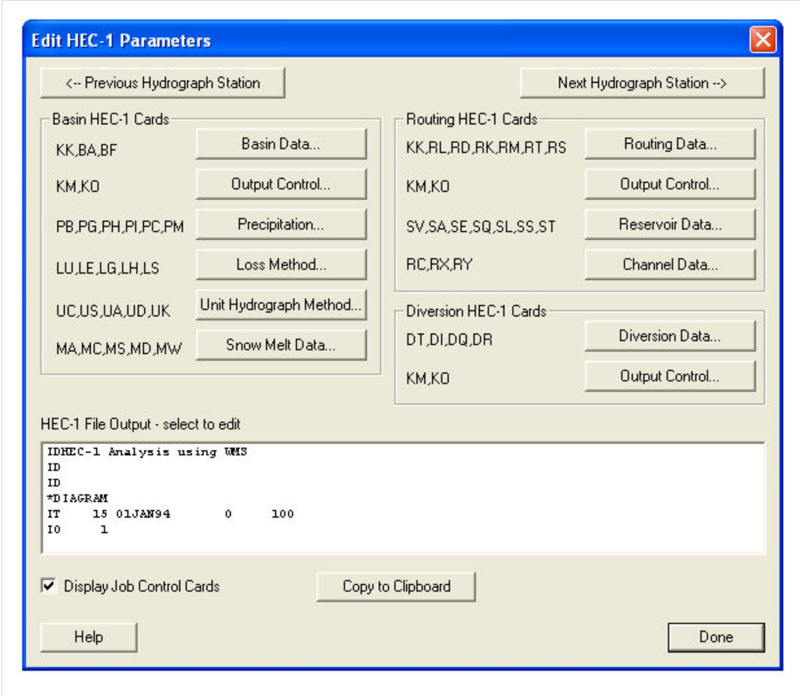
Related Topics

- HEC-1 Parameters
- XY Series Editor

Diversion HEC-1 Cards

Diversion Data...

HEC-1 allows flow to be diverted from an outlet or drainage basin. This flow can be thought of as leaving the normal drainage system at that point. It can be retrieved at a downstream outlet where the diverted flow then contributes to the flow at that outlet. If no downstream retrieval outlet point is specified, the flow simply leaves the system at the diverted outlet point and never returns.



Editing Diversion Data (DT)

Diversions are created using a combination of the **Add | Diversion** and **Retrieve Diversion** commands found in the *Tree* menu. Once created, data for the diversion can be defined and/or edited by selecting the **Diversion Data** button from the *Edit HEC-1 Parameters* dialog.

The following data should be defined for a diversion:

- Name – The name identification string of the diversion as used on the DT record. It is important to assign a unique name to each diversion in a given model because this name is used by WMS and by HEC-1 to identify the diversion.
- Max Volume – Maximum volume of diverted flow in acre-feet (1000 cu m).
- Peak Flow – Peak flow that can be diverted in any computation period in cfs (cms).
- Outflow name – Name used on KK record where flow is diverted.
- Inflow name – Name used on KK record where flow is retrieved.
- The flow capacity of a stream flow diversion is specified using an inflow (DI) and outflow (DQ) tables. These tables are defined with the XY Series Editor by clicking on their respective define buttons.

Output Control...

For each hydrograph station (basin hydrographs, combined hydrographs, and routed hydrographs) different output controls can be specified. This dialog is accessed by selecting the *Output Control* dialog button from the *Edit HEC-1 Parameters* dialog. Entries which can be defined in this dialog are described below.

Routed and Combined Hydrographs at Outlets

In WMS an outlet point is used to represent locations where hydrographs are both combined and then routed. Therefore, if an outlet is selected before choosing the *Output Control* dialog, a radio group at the top of the dialog appears so that you may specify whether the options should be applied to the combined or routed hydrograph. If a basin is selected the radio group at the top of the dialog does not appear.

Comment Lines (KM)

Individual comments can be defined for each hydrograph station. These comments can be used to identify unique characteristics about a particular basin or outlet point. A new comment can be defined by selecting the new button in the *Output Control* dialog and then entering the comment in the text entry. When more than one comment card has been defined, the up and down arrow buttons can be used to scroll through the list of comments for that hydrograph station. When using WMS, comment cards always appear directly after the KK cards for each hydrograph station.

Output Control (KO)

These controls determine what information about a given hydrograph station is written to the HEC-1 ASCII output file. By default the IO record information is used. However, you may wish to print out a more (or less) complete summary for individual hydrograph stations.

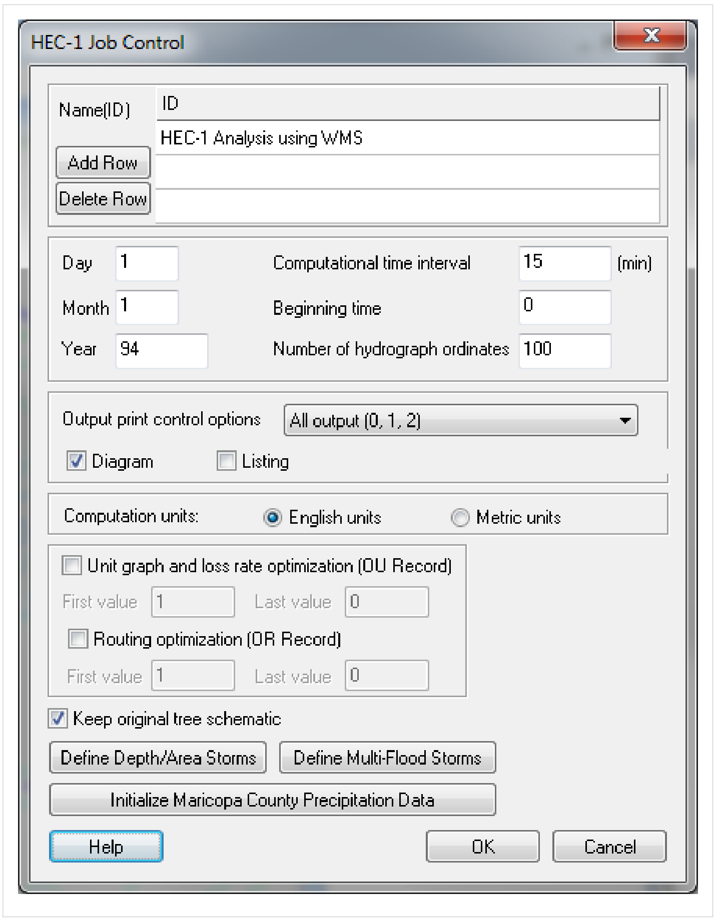
By default, the option to write a hydrograph to the TAPE22 file is specified. This is the file read by WMS for display of hydrographs. Therefore, this option should only be changed to suppress particular hydrographs.

Related Topics

- HEC-1 Parameters
- Running an HEC-1 Analysis
- Adding Diversions
- XY Series Editor

HEC-1 Job Control

The *Job Control* dialog is used to define general information about the HEC-1 model. Selecting the **Job Control** command from the *HEC-1* or *OC Hydrograph* menu accesses this dialog. The *Job Control* dialog can also be accessed by toggling on the display of Job Control cards in the *Edit HEC-1 Parameters* dialog and then clicking on a Job Control related card in the text display window.



Computational Time Interval

The computational time interval entered here will be used to computed the effective precipitation and unit hydrograph.

Expected Value Analysis

Toggle this checkbox on to do an expected value analysis. Effective precipitation values and losses will be automatically updated for each sub-area if they have already been computed.

HEC-1 Job Control Parameters

The *Job Control* dialog is used to define general information about the HEC-1 model. Selecting the **Job Control** command from the *HEC-1* menu accesses this dialog. The *Job Control* dialog can also be accessed by toggling on the display of Job Control cards in the Edit HEC-1 Parameters dialog and then clicking on a Job Control related card in the text display window.

Name (ID)

Enter a name and/or project description identifying the model. Three different name records up to 78 characters each can be entered. The name records will appear at the top of the HEC-1 input file.

Beginning Time

The beginning time of the simulation is defined in this entry. The time should be specified as a single integer number defining the hours and minutes. For example 7:45 am would be entered as 745 and 1:15 PM would be 1315. Do not place a colon between the hours and minutes.

Computation Time Interval (IT)

The computational time interval defines the length in time between hydrograph ordinates. The interval should be specified in minutes, with 1 being the minimum. The total simulation time of the model will be the number of ordinates minus one times the computational time interval.

Day Month Year (IT)

The day, month, and year fields correspond to the date of the first computational time interval. The year is entered as the last two digits only.

NOTE: The date should be consistent with dates defined on IN records for precipitation and other time series data.

Number of Ordinates (IT)

The number of hydrograph ordinates computed during the simulation is defined in this entry. The length of the simulation is determined by the number of ordinates times the computational interval. If a simulation is run and a complete hydrograph is not developed, then the number of ordinates can be increased to increase the length of simulation. Similarly the number of ordinates can be decreased if the simulation continues long past the falling limb of the hydrograph.

Output Control Options (IO)

In the *HEC-1 Job Control* dialog, there is an option to set *Output print control options*. HEC-1 generates an ASCII output file that can be printed or read into any standard word processing program. The information output is controlled by the *Output print control options* in the *HEC-1 Job Control* dialog.

Listing (*LISTING)

Causes HEC-1 to echo print input data.

Tree Diagram (*DIAGRAM)

The diagram option causes a diagram of the stream network to be printed to the HEC-1 output file. This diagram should appear similar to the tree representation created by WMS.

Units (IM)

By default HEC-1 performs computations in English units, however metric unit calculations can be specified.

Basin Data Optimization (OU)

Select this option if you want to optimize unit hydrograph and loss rate parameters in this HEC-1 simulation so the calculated hydrograph will match the observed hydrograph.

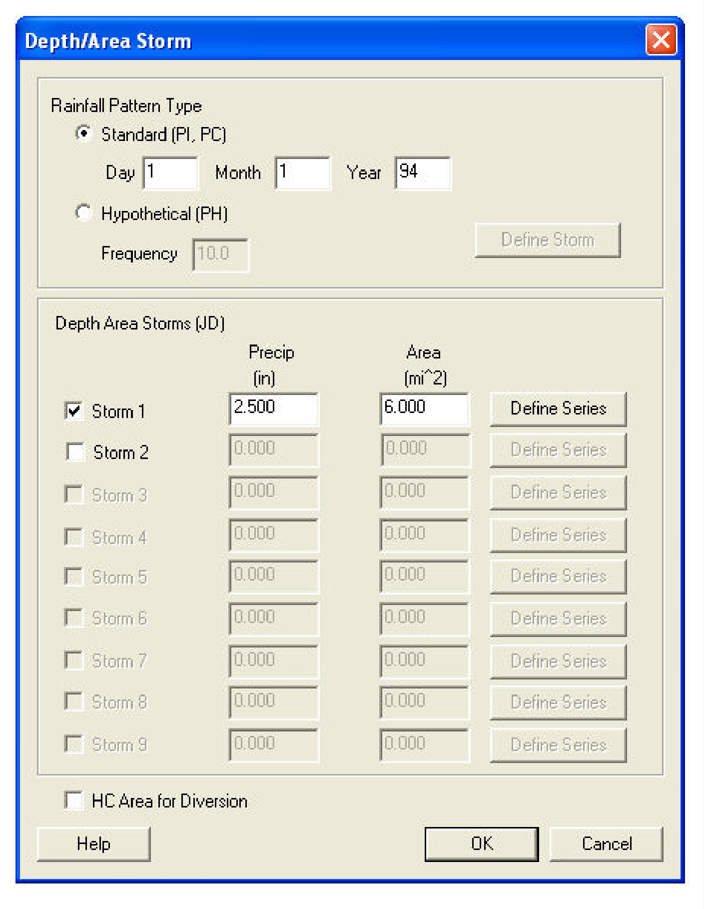
Routing Optimization (OR)

Select this option if you want to optimize routing parameters using observed inflow and outflow hydrographs and a pattern lateral inflow hydrograph for the routing reach.

Depth Area Storms (JD)

Depth Area Storms

Runoff simulations which use a consistent depth/area relationship as defined in the HEC-1 User's Manual[1] can be defined by clicking on the **Define Depth/Area Storms** button to bring up the dialog shown below.



Rainfall Pattern Type

Each storm (JD record) may be defined with a set of PI/PC cards giving the precipitation pattern to be used for that depth and area. If no pattern is given for the second through ninth storms then the previously defined precipitation pattern will be used.

Alternatively hypothetical storms may be used to specify precipitation patterns. In such cases only a single storm using the PH record needs to be defined.

The rainfall pattern type is specified using the radio group at the top of dialog. As a new storm is activated, data fields which require input will be undimmed. Data fields which do not apply to the specified pattern type will remain dimmed.

Depth Area Storms

Different depth/area storms can be defined by toggling on a storm from within the Depth/Area Storms portion of the dialog.

Precipitation

The average precipitation is entered in this data field.

Area

The applicable area for this storm is entered in this data field.

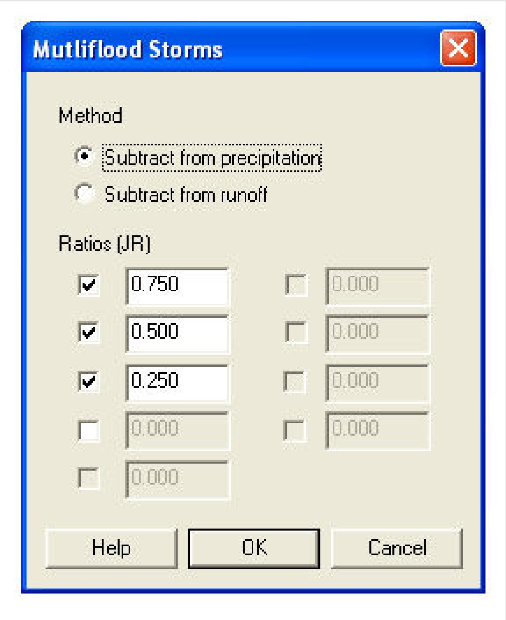
Defining a Series

If a standard storm type has been chosen, a rainfall pattern must be defined using the XY Series Editor. Activate the XY Series Editor by choosing the Define Series button. A series for the first storm must be defined. If a series is not defined for the second through ninth storms, the previously defined storm will be used.

MultiFlood Storms (JR)

Multiple Flood Storms

Multiple ratios of a given storm event can be specified by bringing up the Multi-flood dialog using the **Define Multi-Flood Storms** button. The data input options are described below.



Method

Input ratios can be specified using either precipitation or runoff.

Ratios

Up to nine different storm ratios can be specified by toggling on the appropriate check box. All hydrograph or precipitation ordinates are multiplied by the specified value.

Each storm is analyzed during execution of HEC-1 and the resulting TAPE22 file will contain as many hydrographs for each station as there are ratios defined. Display of different storm ratios can be toggled on and off in the *Hydrologic Modeling* tab of the *Display Options* dialog.

Related Topics

- Edit HEC-1 Parameters

References

[1] http://www.hec.usace.army.mil/software/legacysoftware/hec1/hec1-documentation.htm

HEC-1 Input Hydrograph (QI)

If a hydrograph is known for a given basin, there is no need to compute a synthetic hydrograph. This hydrograph can be input by selecting the check box and then defining the hydrograph using the XY Series Editor.

Precipitation, base flow, loss rates, and unit hydrograph methods for each hydrograph must be specified, regardless of whether or not a TIN is being used, before a complete HEC-1 file can be created. Selecting all of the basins enters data for one or more basins.

NOTE: If no basins are selected, the information entered is applied to all basins.

Related Topics

- Editing HEC-1 Parameters
- Basin Data
- Observed Hydrograph

HEC-1 Model Check

The **Model Check** command should be issued once you feel that all necessary HEC-1 data has been defined. It will report any possible errors/inconsistencies in your model so that corrections can be made prior to executing HEC-1. Two types of information are provided as a result of this command. The first type is simply informational and provides things such as the starting time, time step, and total time of the simulation. You should verify that these parameters are what are intended. The second type of information messages are errors and must be corrected before an accurate HEC-1 analysis can be performed. The list of checks made is not complete and just because no errors are reported does not insure that a successful and/or accurate analysis will be completed. We encourage you to report any additional checks that might be made as you work through various problems.

Related Topics

- Editing HEC-1 Parameters
- Running an HEC-1 Analysis
- Saving HEC-1 Files

HEC-1 Observed Hydrograph (QO)

This record is used to input an observed hydrograph for an optimization job (OU record). This hydrograph can be input by selecting the check box and then defining the hydrograph using the XY Series Editor.

Related Topics

- Editing HEC-1 Parameters
- Basin Data
- Input Hydrograph
- XY Series Editor

HEC-1 Reading Existing Files

WMS is capable of reading HEC-1 files that have been manually created using a text editor or some other program. However, there are a couple of problems which need to be considered, and may have to be altered either before or after reading in one of these files.

- There can be no blank fields in a file read in by WMS. If a field is left blank, HEC-1 assumes the value of this field is 0. However, errors will occur or data will be lost if a file with a blank field is read into WMS.
- Names for all KK (Hydrograph Station identifying card) records must be unique. This problem won't surface until you run HEC-1 with a new file created by WMS and try to read the hydrographs. In such cases all hydrographs will displayed at the first hydrograph station (KK record) with the duplicate name. This can be changed either inside or outside of WMS.
- WMS reads hydrograph results from the TAPE22 file. Many existing HEC-1 files will not specify output to this file and you may need to define it for all hydrograph stations before you will be able to read in the modeling results. This can be done by selecting all basins/outlets and bringing up the respective output control dialog.
- Only the first three ID cards are read into WMS.
- All KM cards (comments) are placed directly after the KK record for a hydrograph station. WMS will read KM cards from any position within the HEC-1 file, but always writes them out directly following the KK card.

WMS writes out a few other comment cards (preceded by an *) that are ignored by HEC-1. These cards are not necessary, but you should be aware of differences you might see from your original file and the one created by WMS.

Related Topics

- Saving HEC-1 Files

HEC-1 Run Analysis

The version of HEC-1 distributed with WMS can be run directly from WMS by using the **Run HEC-1** command in the *HEC-1* menu. Before running an HEC-1 simulation you should run the model checker which will help you identify serious and potential problems that should be corrected before a successful run of HEC-1 can be made.

The **Run HEC-1** command will bring up a dialog allowing you to specify three files which are necessary to run HEC-1. The first file is the HEC-1 input file. The second is an ASCII output file generated by HEC-1 and can be used to extract specific results values. It also contains important information which can be used to correct problems encountered when running HEC-1. The third file is a new name for the TAPE22 file, and will contain hydrograph results for basins and outlets. You can view these results by reading this file with the **Open** command from the *Hydrographs* menu.

Once these files have been defined and you select **OK**, HEC-1 will be executed. A separate window will appear and information about the HEC-1 simulation will be reported.

If HEC-1 is not executed successively when issuing this command then for PC computers be sure that the path to your HEC1.EXE file is included in AUTOEXEC.BAT.

If HEC-1 does not run to a successful completion you can view the ASCII output file using the **View File** command in the *File* menu.

Related Topics

- Saving HEC-1 Files
- Model Check
- HMS

HEC-1 Saving Files

Once a topologic tree has been created and all of the necessary data entered, an HEC-1 input file can be generated by selecting the **Save HEC-1 File** command from the *HEC-1* menu. When writing the file, the proper order for computing, combining, and routing hydrographs is automatically determined. HEC-1 can be run without any further editing of the input file generated by WMS. Because WMS does not allow input for all HEC-1 options, it may be necessary to modify the file somewhat before execution. Hydrograph names defined on KK cards should not be changed, as they are needed to correctly read hydrographs generated by HEC-1 back into WMS for post-processing.

WMS can read HEC-1 input files so that data previously entered can be restored for basins and outlets. Names on the KK cards must match the basin or outlet names when reading the file for an existing terrain model.

Existing files generated outside of WMS can be read into WMS and a separate topological tree will automatically be generated for the watershed described in the file. Since WMS does not support all possible HEC-1 card types, there may be some incomplete information. However, the basic structure of the watershed will be created and all possible data will be retained. Parameters from unrecognized cards, and/or hydrograph names, are ignored.

Related Topics

- Running an HEC-1 Analysis
- Reading Existing Files

Computing Area Between Elevations

The **Compute Area Between Elevations** command is useful for determining areas in different elevation zones as part of a snow melt analysis. This operation can also be done when defining snow melt parameters for HEC-1. Model units are assumed to be either in feet or meters and subsequent areas are converted to square miles or square kilometers according to the metric flag set in the HEC-1 Job Control dialog.

This same procedure is also useful for determining storage capacity curves.

Related Topics:

- Snow Melt Simulations
- Storage Capacity Curves

Weighted Average Precipitation

Once a watershed has been delineated, follow the steps below to use rain gage data to compute a basin average precipitation with HEC-1.

1. Switch to the "Drainage Module".
2. Select the *Select drainage unit or basin* tool and double click on the watershed which you would like to assign precipitation based on rain gages.
3. Select the **Precipitation** button.
4. Select the *Gage (PG)* option. Select **OK**, then **Done**.
5. Right click on the *Coverages* folder and select **New Coverage** to create a new coverage.
6. Change the "Coverage type" to *Rain Gage* and select **OK**.
7. Create your rain gages with the *Create Feature Point* tool (you can also add rain gages later on)
8. Right click on a rain gage with the *Select Feature Point/Node* tool and select **Attributes** (you can also double click on the gage). The *Rain Gage Properties* window will appear. You can now do the following (NOTE: make sure the Show combo box in the upper right is set to All in order to view all gages):
    - Add additional gages with the **Add Gage** button
    - Edit the X, Y coordinates of each gage
    - Enter the amount of precipitation at each gage
    - Assign a temporal distribution to each gage by activating the *Temporal distribution* check box and selecting **Define**.
9. Select **OK** when finished adding rain gages.
10. Switch to the *Hydrologic Modeling Module*. Select the **Select Basin** button.
11. Double click on the basin.
12. Select the **Precipitation** button.
13. Select the *Gage (PG)* option. Select **OK** and then **Done**.
14. Select **HEC-1 | HEC-1 Rain Gages**.
15. Select the rain gage coverage in the *Coverage:* combo box. Select **OK**.
16. Select **DEM | Compute Basin Data** while in the *Drainage Module*. Enter a *Gage weight computation cell size* and the select **OK**.

6.4. Hydrologic Engineering Center-Hydrologic Modeling System (HEC-HMS)

HEC-HMS

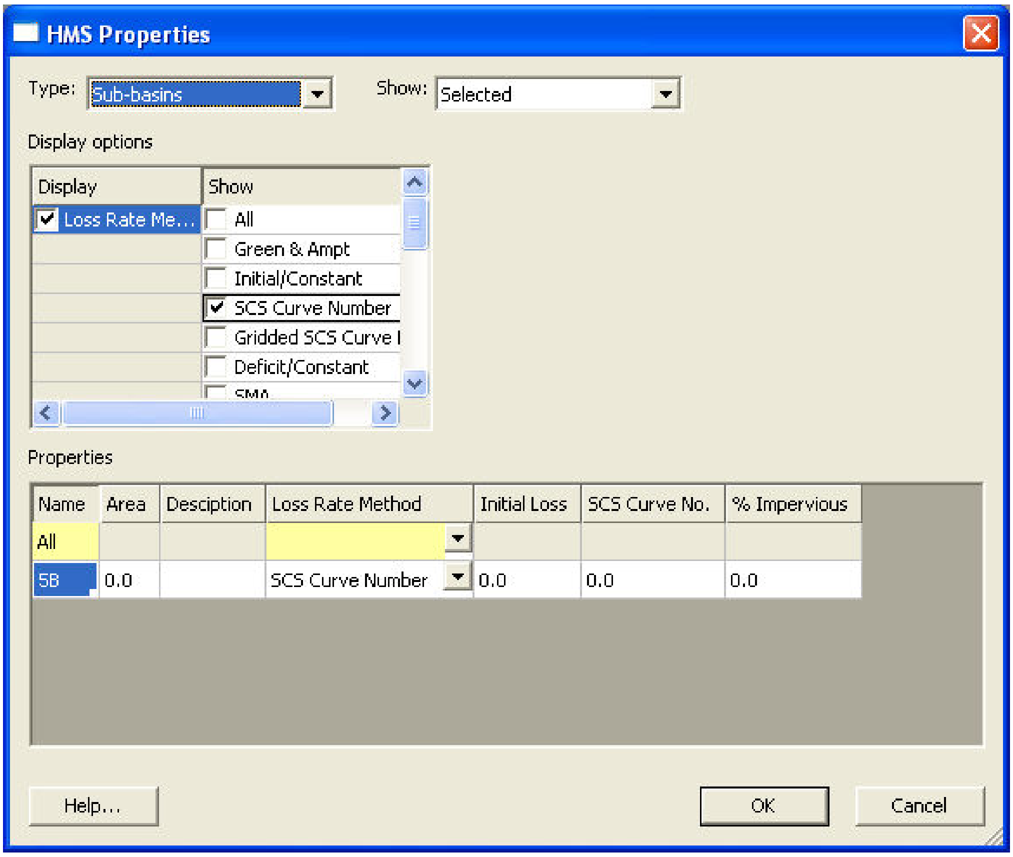
Overview

The Hydrologic Modeling System (HMS), written by the Hydrologic Engineering Center (HEC), simulates a watershed's rainfall-runoff process. It is designed to replace HEC-1, and has similar options to HEC-1, but incorporates some advances in hydrologic engineering. These advances include (1) a linear quasi-distributed runoff transform (called ModClark) for use with gridded precipitation, (2) continuous simulation capabilities with either a one-layer or more complex five-layer soil moisture method, and a parameter estimation option. HMS also incorporates a graphical user interface (GUI) for entering hydrologic data, where HEC-1 data was entered using text input files. WMS 7.1 and later versions provide tools for setting up, computing data for, and entering data for HMS models. These models can then be saved to HMS format and the model can be run in HMS. The results from these models can then be read into and viewed in WMS.

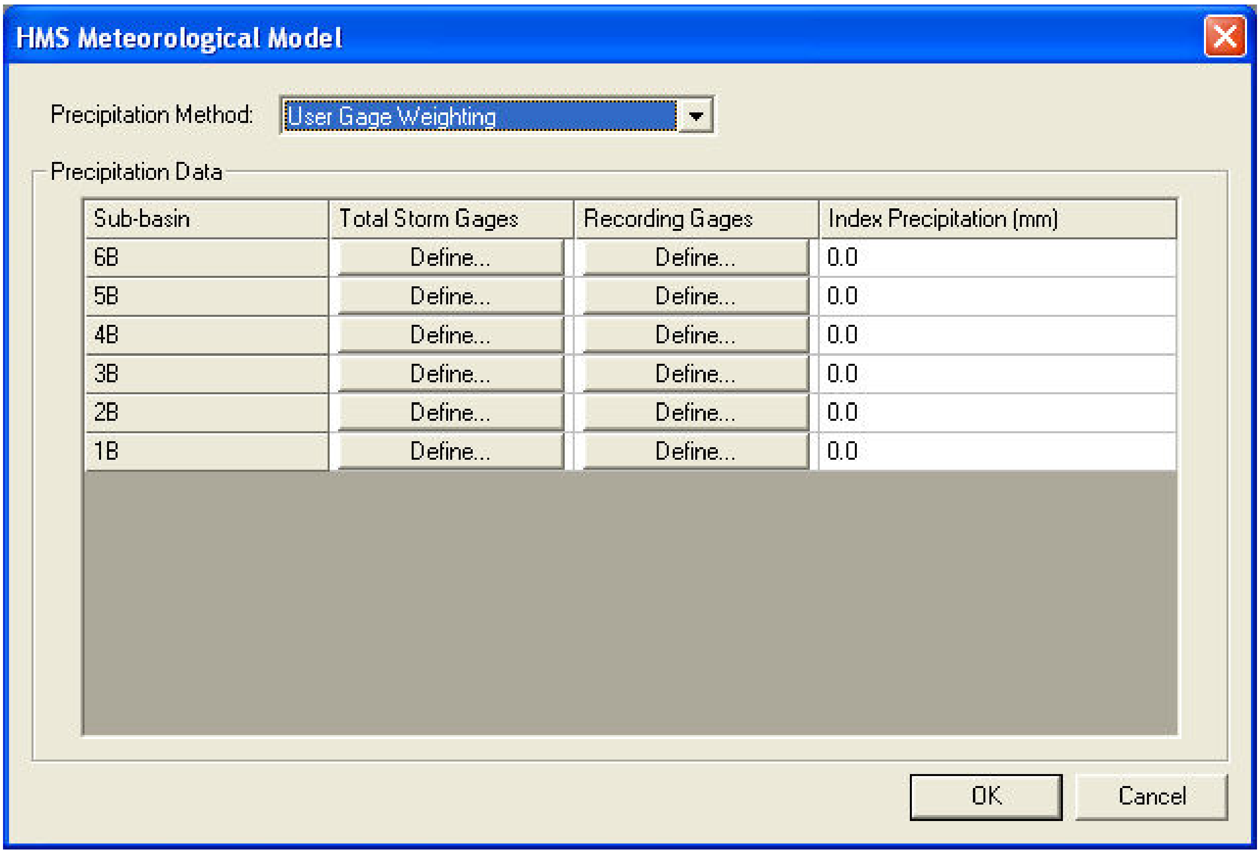
You may wonder why you would set up and run an HMS model in WMS when you can just do it in HMS. The reason you would do this is because WMS provides GIS, watershed modeling, and HMS parameter computation capabilities all in one place, while HMS does not provide this.

The HEC Data Storage System (DSS) is a format that was previously unsupported in WMS. Since HMS uses this format for importing and exporting time series data, WMS 7.1 and later versions now support importing and exporting DSS files, allowing for reading and writing time series data from and to HMS.

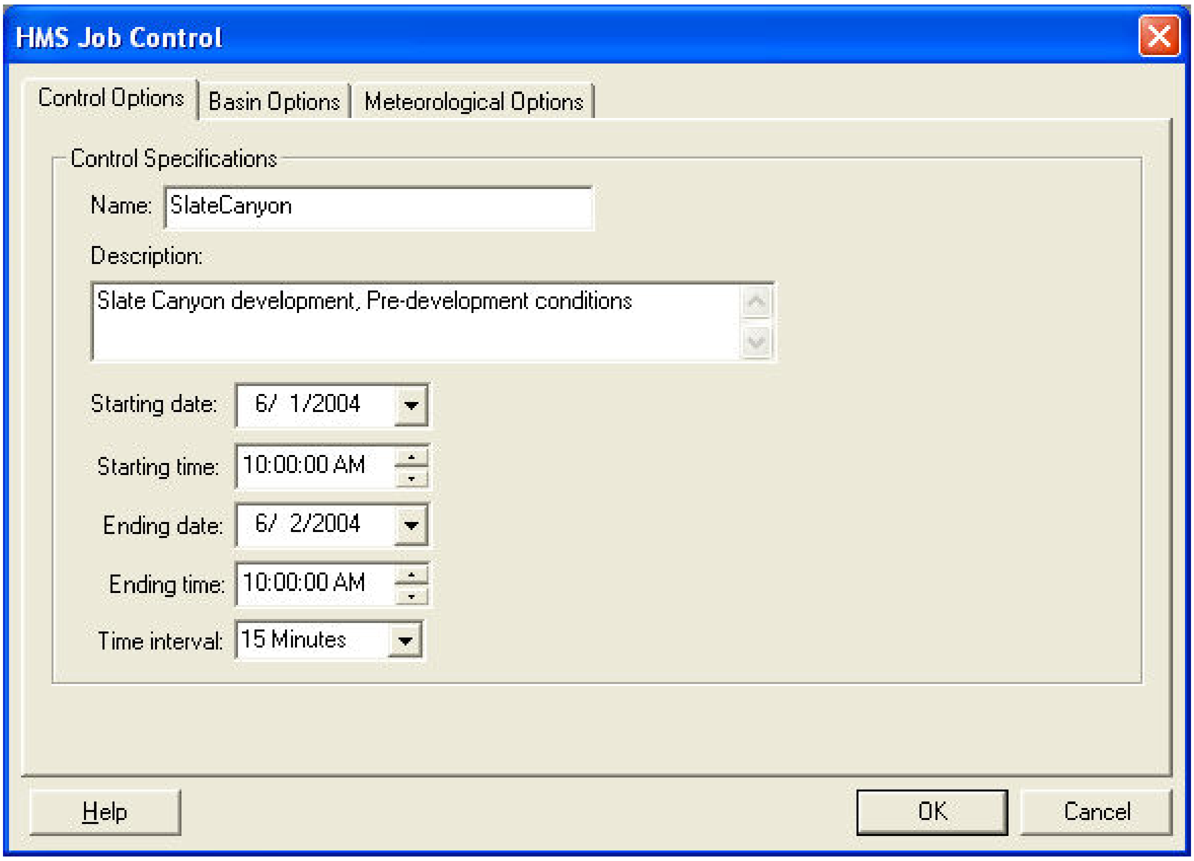
All the HMS data is entered using one of three dialogs. These dialogs are meant to correspond to the three different types of data entered in HMS: The HMS meteorologic data, the control data, and the basin data. The first dialog used in WMS is the *HMS Properties* dialog, shown below:



The second is the *HMS Meteorologic Model* dialog, shown below:



The final dialog is the *HMS Job Control* dialog:



HEC-1 / HEC-HMS Comparison

HMS includes most of the same capability as HEC-1, but has dropped some of the lesser used functions and added others. You can use WMS to delineate watershed data and define most parts of the HMS model. The Export HMS Basin File found in the *HEC-1* menu will create the input file necessary to use data derived in WMS to perform modeling in HMS.

There are five different files that are (or can be) exported as part of the HMS simulation.

1. The Project file is like the WMS super file in that it is used to keep track of the other files that make up the HMS simulation.
2. The Control file includes the job control data, or simulation global parameters such as time step, number of ordinates, id cards, etc.
3. The Basin file has all of the parameters for each hydrologic unit (basin, outlet, diversion, etc.).
4. The Precipitation file contains the information used to define the precipitation event for the simulation.
5. The Map file is a map or trace of the watershed and sub-basin boundaries. It is not a terrain model and therefore cannot be used to extract information such as area or runoff distance; it is only a "picture" of the watershed that is placed as a backdrop to the HMS schematic. If you have a watershed derived in WMS from a digital terrain model then you can export it as part of your HMS project. In order to export the map you must convert it to feature objects. If you create your watershed from feature objects or a DEM then it will already be in this format. If you create your watershed from a TIN then you will need to use the **Drainage Data → Feature Objects** command found in the *Drainage* menu prior to exporting your HMS files.

Related Topics:

- Post Processing HMS Results
- Running HMS
- Saving an HMS File
- HMS Properties Dialog
- Job Control
- Saving HEC1 Files
- Running an HEC-1 Analysis

HEC-HMS Base Flow

Shallow groundwater that returns quickly (relative to deep groundwater flow) to contribute to stream flow during a precipitation event is classified as base flow. Four different base flow methods are available for estimating this return of infiltrated precipitation to the channel.

Recession

The recession method is generally more applicable to shorter duration periods for watersheds where the volume and timing of the base flow is strongly influenced by the precipitation event itself. The input parameters for the recession base flow method are as follows:

- Initial flow at the start of the storm in cfs or cfs/sq. mile (cms or cms/sq. km for metric units).
- Recession constant describing the rate of base flow decay. The constant represents the ratio of base flow at the present time to the flow one day earlier and consequently ranges between 0 and 1.
- Threshold flow in cfs (cms) below which base flow recession occurs in accordance with the recession constant and corresponds to the point on the hydrograph where base flow replaces overland or runoff flow as the source from the sub-basin. This can be specified as an absolute value or as a ratio of the peak flow.

Constant Monthly

The constant monthly method uses a constant base flow in cfs (cms) at all simulation time steps that fall within a given month. The method is intended for use in long term simulations and requires a separate value for each month that is part of the overall simulation time.

Linear Reservoir

The linear reservoir method computes base flow from groundwater storage and can only be used in conjunction with the soil moisture accounting loss method. Available water from each groundwater layer (there are two layers defined) is converted to base flow through a specified number of linear reservoirs. The storage coefficient and number of linear reservoirs is required for each of the two layers.

Bounded Recession

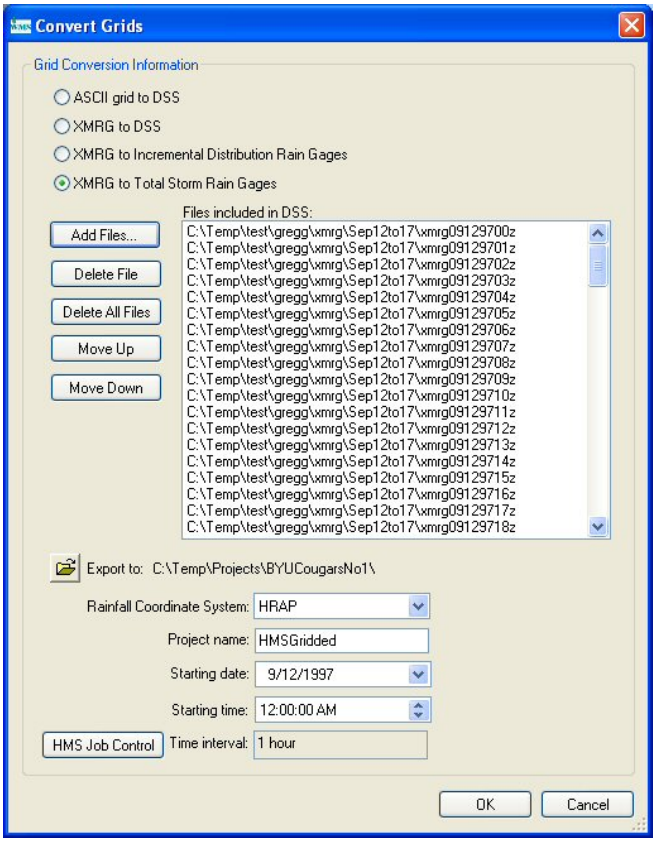
The bounded recession method is similar to the recession, but includes a bounding recession value for each month so that it can handle individual events within a long term simulation better.

Related Topics

- Editing HMS Properties
- Sub-Basins
- Loss Rates
- Transform

HEC-HMS Convert Grids

The *convert grids* dialog is used to manage the process of converting NEXRAD RADAR rainfall grids in XMRG or ASCII grid format [1] to a format that is usable with WMS or HMS. This dialog is available if the *Gridded Precipitation* option or the User Gage Weighting option is selected in the HMS Meteorological Model dialog.



The following commands are available in this dialog:

- **ASCII grid to DSS –** This option converts rainfall grids in ArcInfo ASCII grid format to a gridded DSS file. The *Export to* directory defines the export directory and the *Project name* defines the DSS filename. Since the DSS file also contains date and time information, the specified *Starting date* and *Starting time* will determine the starting time of the first grid in the file list. Each additional grid will be saved to the DSS file based on the time interval specified in the *HMS Job Control* dialog and listed at the bottom of this dialog.
- **XMRG to DSS –** This command first converts XMRG grids to Arc/Info ASCII grid format, and then converts the grids to a gridded DSS file. Everything else is done in the same manner as the **ASCII grid to DSS** command.

- **XMRG to Incremental Distribution Rain Gages –** This command converts the points in an XMRG file to incremental distribution rain gages in a rain gage coverage. These gages can then be tied to a basin in a hydrologic model. When the points in the rain gage coverage are created, their coordinates are converted to the current coordinate system, if one is defined. This command will not function unless the current coordinate system is defined.
- **XMRG to Total Storm Rain Gages –** This command converts the points in an XMRG file to total storm rain gages in a rain gage coverage. The coordinate system of the points is converted to the current coordinate system.

Related Topics:

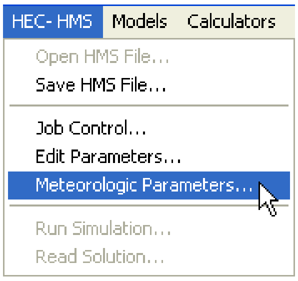
- Defining the Meteorological Model
- Gages
- Rain Gage Coverage

References

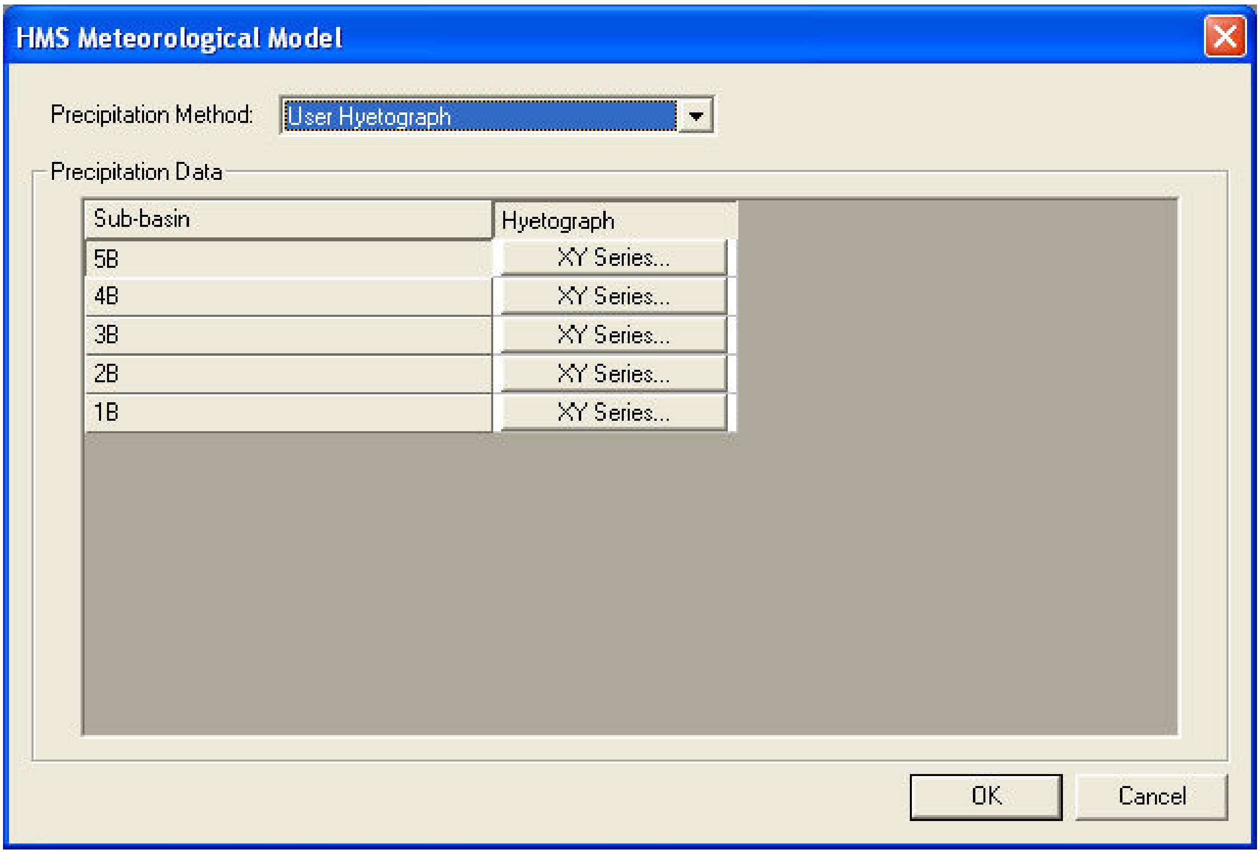
[1] http://dipper.nws.noaa.gov/hdsb/data/nexrad/nexrad.html

HEC-HMS Defining the Meteorological Model

You can define meteorologic data for your model by selecting the *Meteorologic Parameters* menu item from the *HEC-HMS* menu.



Selecting this menu item will bring up the *HMS Meteorological Model* dialog. This dialog is used to assign precipitation data to all the basins in the model.



The same precipitation method must be assigned to all the basins in your HMS model. The precipitation method you choose should be based on the type of analysis you want to run. If you have rainfall gage data for your watershed, you can use the user hyetograph or the user gage weighting method. If you want to create a synthetic storm with a known exceedance probability, select the frequency storm method. If you want to model a standard project storm, select the standard project storm or the SCS hypothetical storm method. You should only use the *No Precipitation* option if your HMS model does not have any basins. See the topics below for more detailed information about the *HMS Meteorological Model* dialog.

Related Topics:

- Frequency Storm
- Gages
- Convert Grids (NEXRAD Radar rainfall grids)
- SCS Hypothetical Storm
- Standard Project Storm
- User Hyetograph
- Meteorological Options
- HMS Overview

HEC-HMS Diversions

HMS allows flow to be diverted from an outlet or drainage basin. This flow can be thought of as leaving the normal drainage system at that point. It can be retrieved at a downstream outlet where the diverted flow then contributes to the flow at that outlet. If no downstream retrieval outlet point is specified, the flow simply leaves the system at the diverted outlet point and never returns (similar to a sink).

Diversion Data

Diversions are created using a combination of the Add and Retrieve Diversion commands found in the *Tree* menu. Once created, data for the diversion can be defined and/or edited by selecting the *Edit Parameters* option from the *HEC-HMS* menu.

The following data should be defined for a diversion:

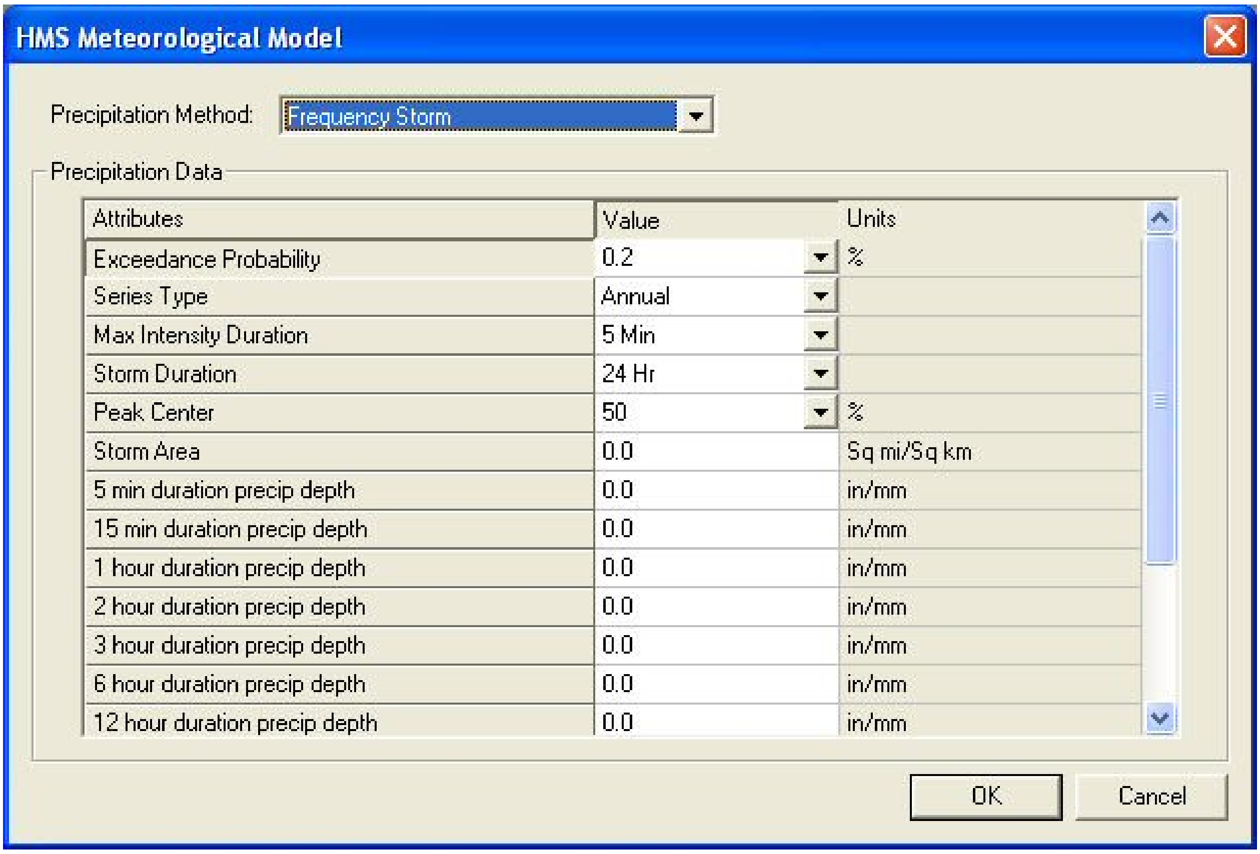
- Name – The name identification string of the diversion as used on the DT record. It is important to assign a unique name to each diversion in a given model because this name is used by WMS and by HEC-1 to identify the diversion.
- Max Volume – Maximum volume of diverted flow in acre-feet (1000 cu m).
- Peak Flow – Peak flow that can be diverted in any computation period in cfs (cms).
- The flow capacity of a stream flow diversion is specified using an inflow and outflow table. This table is defined with the XY Series Editor.

Related Topics

- Editing HMS Properties
- Adding Diversions
- Retrieving Diversions

HEC-HMS Frequency Storm

"The frequency storm method can be used to create a balanced, synthetic storm with a known exceedance probability. Automatic adjustments for storm area and series type are based on the exceedance probability. Depth-duration data are usually obtained from publications such as TP-40 (National Weather Service, 1961)." (HEC-HMS User's Manual) The options for defining a frequency storm are shown below:



Follow the following steps (from the HEC-HMS User's Manual) to set up a frequency storm:

1. Select a storm exceedance probability from the list (Exceedance probability = (1 / Return Period) * 100%).
2. Set the series type for the desired output.
3. Select the maximum intensity duration and the total storm duration.
4. Enter the precipitation depths corresponding to the selected exceedance probability for the durations between the maximum intensity and storm durations.
5. Select the percentage of the storm duration that occurs before the peak intensity.
6. Enter the storm area. This is equal to the total drainage area at the point where the exceedance probability will be inferred for the computed flow.

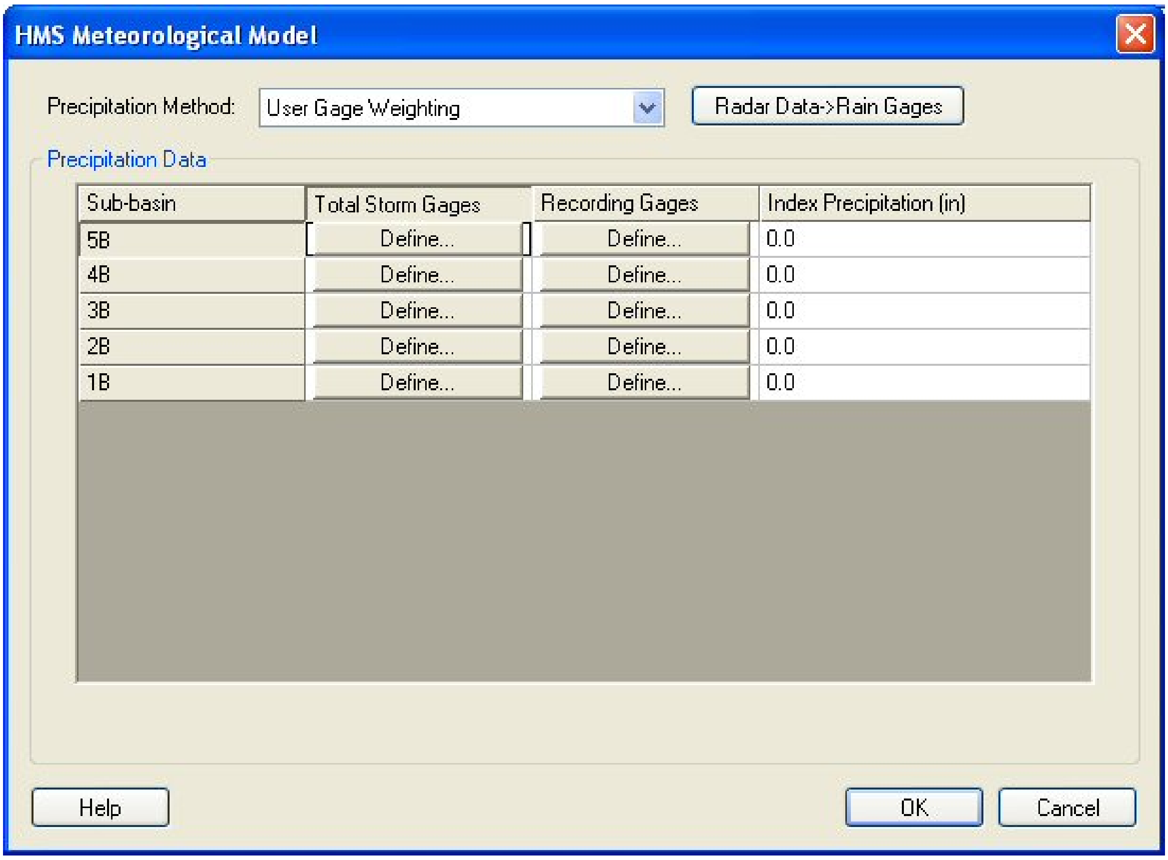
Related Topics:

- Defining the Meteorological Model

HEC-HMS Gages

The User Gage Weighting method allows you to define one or more gages for each sub-basin in your watershed model.

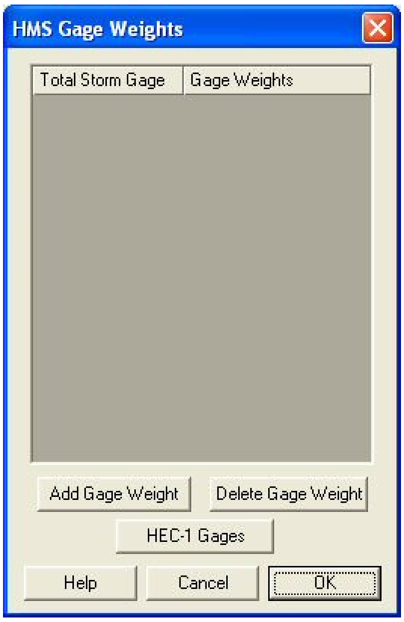
The method of defining gages in HMS is similar to the method of defining gages in HEC-1. For each basin, you must define one or more total storm gages and one recording gage. WMS has the capability to automatically determine gage weights for each sub-basin using the Thiessen polygon method. The data required for the User Gage Weighting method are shown below:



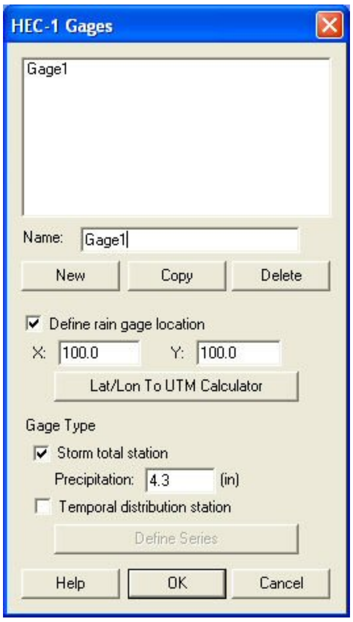
If you have a Map or DEM-based watershed associated with a coordinate system, you can import NEXRAD RADAR grids in XMRG format [1] and WMS will convert the radar data to gages in a rain gage coverage in your current coordinate system. The RADAR grids are read by selecting the Radar Data->Rain Gages button and defining your grids and the time step associated with your grids.

Here is how you would define meteorological data for each sub-basin using the user gage weighting method:

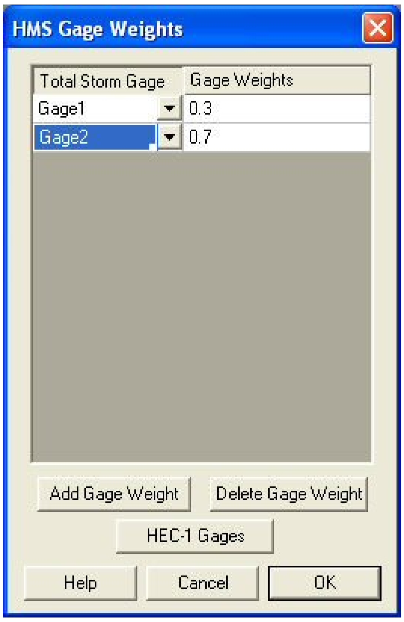
1. Select the **Define** button under the total storm gages column. The following dialog will appear:



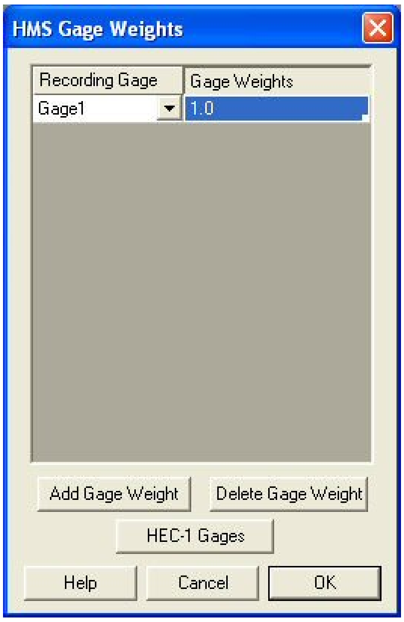
2. Define any rain gage data you have by selecting the **HEC-1 Gages** button. The *HEC-1 Gages* dialog will appear. Enter gage data for any storm total stations (total storm precipitation values) and at least one temporal distribution station (XY series of precipitation values for the entire storm).



3. After you are done defining all the gage data, go back to the *HMS Gage Weights* dialog and add one or more gage weights associated with a total storm gage station. Enter the weights for each of these gages, as shown below.



4. Go back to the *HMS Meteorological Model* dialog, select the **Define** button under *Recording Gages*, and define one recording gage for the basin. This is a gage which defines the temporal distribution of the rainfall. Normally, you would define only a single recording gage, though you can define more than one.



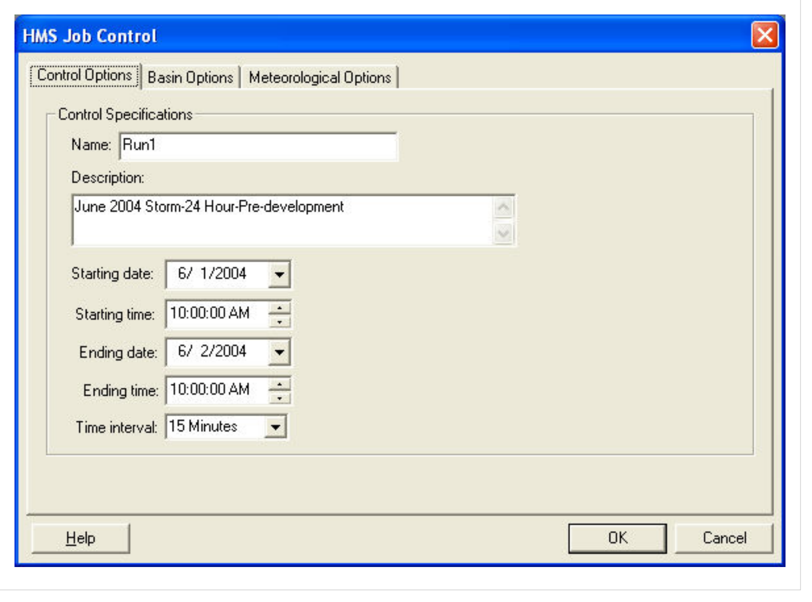
5. Go back to the *HMS Meteorological Model* dialog and (optionally) enter index precipitation values for each sub-basin. This index precipitation accounts for regional precipitation variation between sub-basins.

Related Topics:

- Defining the Meteorological Model
- Gages

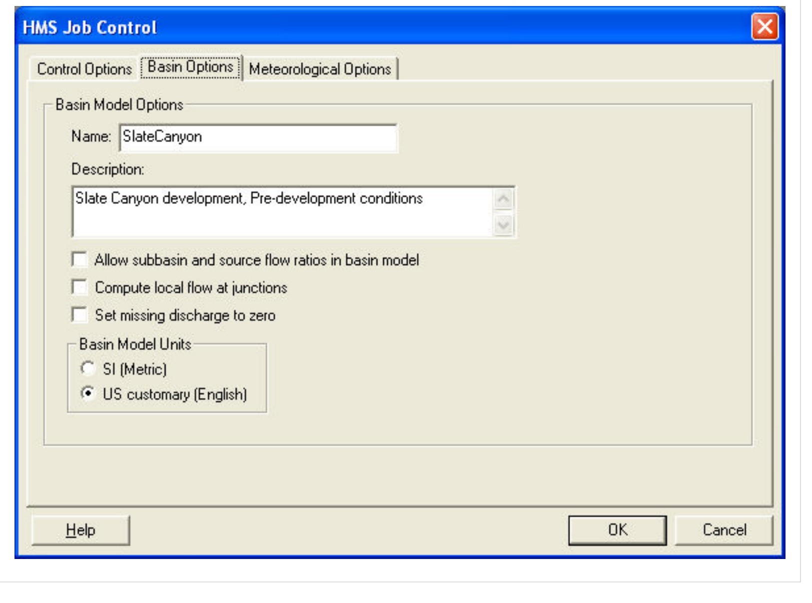
HEC-HMS Job Control

The control option data entered in WMS is exported to the HMS project and used to set the appropriate HMS parameters when you read the project into HMS. After reading the project into HMS, the data entered in the *Control Options* tab shows up in the *Control Specifications* of the HMS model. The data included in the basin model include the name, description, and the other items in the *Control Options* tab shown below:



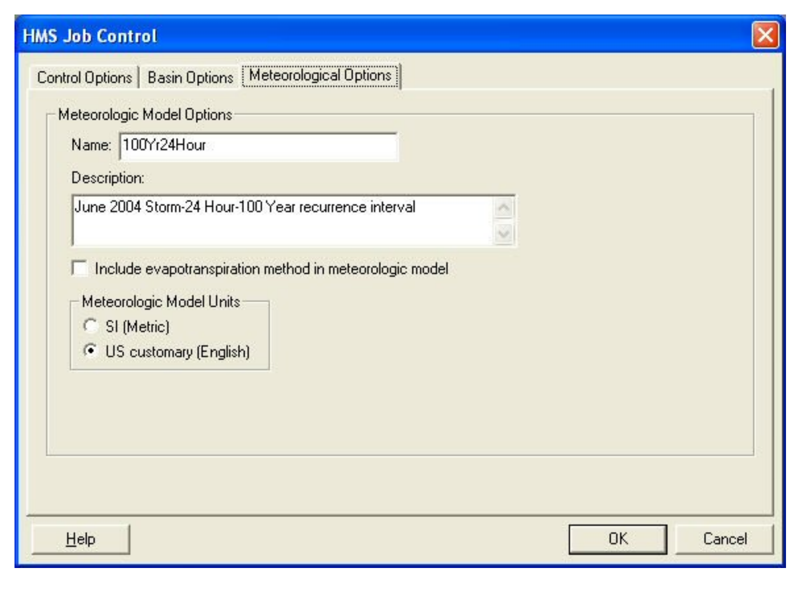
Basin Options

The basin option data entered in WMS is exported to the HMS project and used to set the appropriate HMS parameters when you read the project into HMS. The data included in the basin model include the name, description, and the other items in the *Basin Options* tab shown below:



Meteorological Options

The meteorological option data entered in WMS is exported to the HMS project and used to set the appropriate HMS parameters when you read the project into HMS. The data included in the basin model include the name, description, and the other items in the *Meteorological Options* tab shown below:



You can define all other meteorological data in the Defining the Meteorological Model dialog.

Related Topics

- Saving an HMS File
- Defining the Meteorological Model
- HMS Overview

HEC-HMS Junctions

"A junction is an element with one or more inflows and only one outflow. All inflow is added together to produce the outflow by assuming zero storage at the junction. It is usually used to represent a river or stream confluence." (HEC, 2002)

WMS treats outlets as junctions. This means that when you create an outlet, you are actually creating a junction. You are also creating a reach when you create an outlet. The only data associated with junctions is a description. This description is exported to the HMS file when you save the file. A reach is also associated with each outlet.

Related Topics

- Editing HMS Properties
- Reach Routing

HEC-HMS Loss Methods

The loss rate method defines the equations used in the HMS simulation to separate precipitation volumes from runoff excess. Each of the following methods require one or more input parameters to be defined.

Green & Ampt

The Green and Ampt method requires the following variables to be defined:

- Initial loss (abstraction) in inches (mm).
- Volumetric moisture deficit. If this value is 0, then the method reduces to the initial loss equal to IA and a constant loss equal to XKSAT.
- Wetting front suction in inches (mm). If this value is 0, then the method reduces to the initial loss and a constant loss equal to the saturated hydraulic conductivity.
- Hydraulic conductivity at natural saturation in inches/hour (mm/hour).
- Percentage of drainage basin that is impervious.

Using methods defined by the Maricopa County Flood Control District, Green-Ampt parameters can be determined from GIS data layers automatically in WMS.

Initial/Constant

This loss method uses an initial value and a uniform value to define infiltration losses. Input parameters are as follows:

- Initial rainfall/snow melt loss in inches (mm) for snow free ground.
- Uniform rainfall/loss (in inches/hour or mm/hour) which is used after the starting loss has been satisfied.
- Percentage of drainage basin that is impervious.

SCS Curve Number

The SCS curve number method uses the following parameters:

- Initial rainfall abstraction in inches (mm) for snowfree ground. If value is 0, then initial abstraction will be computed as 0.2*(1000/CRVNBR-10).
- SCS curve number for rainfall/ losses on snowfree ground.

  NOTE: Composite Curve Numbers can be computed automatically when this method for computing losses is chosen and a terrain model is present.
- Percentage of drainage basin that is impervious.
- Composite curve numbers can be computed from digital land use and soils files with an accompanying CN value mapping table.

Gridded SCS Curve Number

With the gridded SCS curve number method a grid defining the CN value must be defined. An initial abstraction ratio must also be defined as well as the potential retention scale factor. The default initial abstraction ratio as originally suggested by the SCS is 0.2, but later research has shown that for many watersheds this value could be as small as 0.05. This method should only be used with the ModClark unit hydrograph transform method.

Deficit/Constant

The Deficit/Constant method is much like the Initial/Constant uniform where a total losses volume and an initial losses volume are used to specify an "initial" value. This method uses the following parameters:

- Initial deficit in inches (mm).
- Constant loss rate in in/hr (mm/hr).
- Maximum deficit in inches (mm).
- Percentage of drainage basin that is impervious.

SMA

Soil moisture accounting uses a five layer model that includes evapotranspiration calculations. The five layers include:

- Canopy storage layer
- Surface depression storage layer
- Soil profile layer
- Groundwater 1 layer (shallow groundwater)
- Groundwater 2 layer (deep percolation groundwater)

To define the SMA method you must first define the capacity of each layer as well as the initial storage as a percent of that capacity. In addition, infiltration rates for the soil and groundwater layers have to be defined (based on soil types) and the tension zone capacity for the soil profile layer and storage coefficients for the groundwater layer must also be defined. Composite area-weighted parameters can be computed for each drainage basin by overlaying soil and land use coverages with the drainage basins using the GIS calculator.

Gridded SMA

The gridded SMA method uses the same parameters as the SMA, except that they are defined on a gridded basis rather than by sub-basin. This method should only be used with the ModClark unit hydrograph transform method. Use the Compute HMS Grid Parameters... menu command to compute the parameters required for each grid cell.

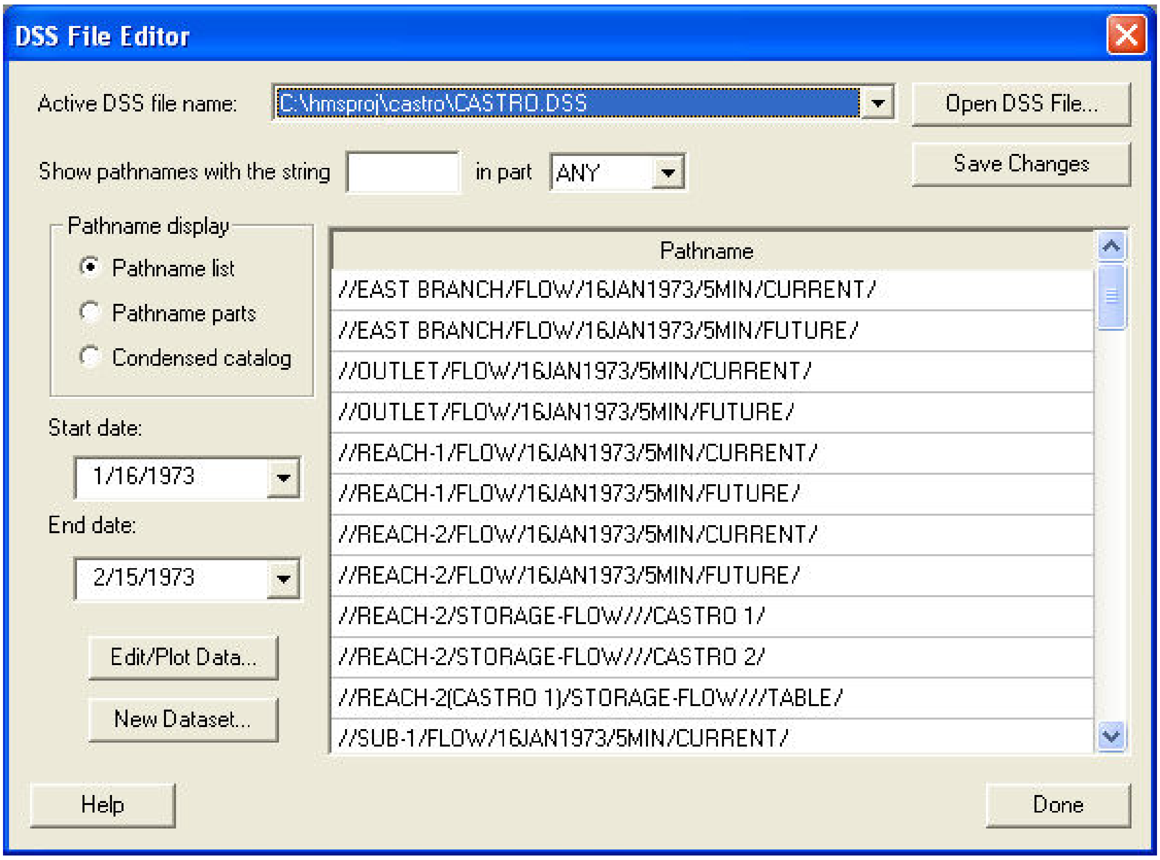
Related Topics

- Editing HMS Properties
- Sub-Basins
- Base Flow
- Transform

HEC-HMS Post Processing

To run an HMS simulation, you first need to save the HMS file from WMS. Then, start HMS, read the HMS file, and run the simulation in HMS. Finally, you can read the solution back into WMS by opening the HMS DSS solution file from WMS.

To read the solution into WMS, open the DSS file associated with the name of the basin. For example, if the basin name is "Subbasin-3", and the HMS project file name is "castro.hms", you would find a file called "castro.dss" in the same directory as "castro.hms". In WMS, select **File | Open** and go to the directory where the DSS file that you want to view is located. Open the DSS file and a dialog like the following will show up.

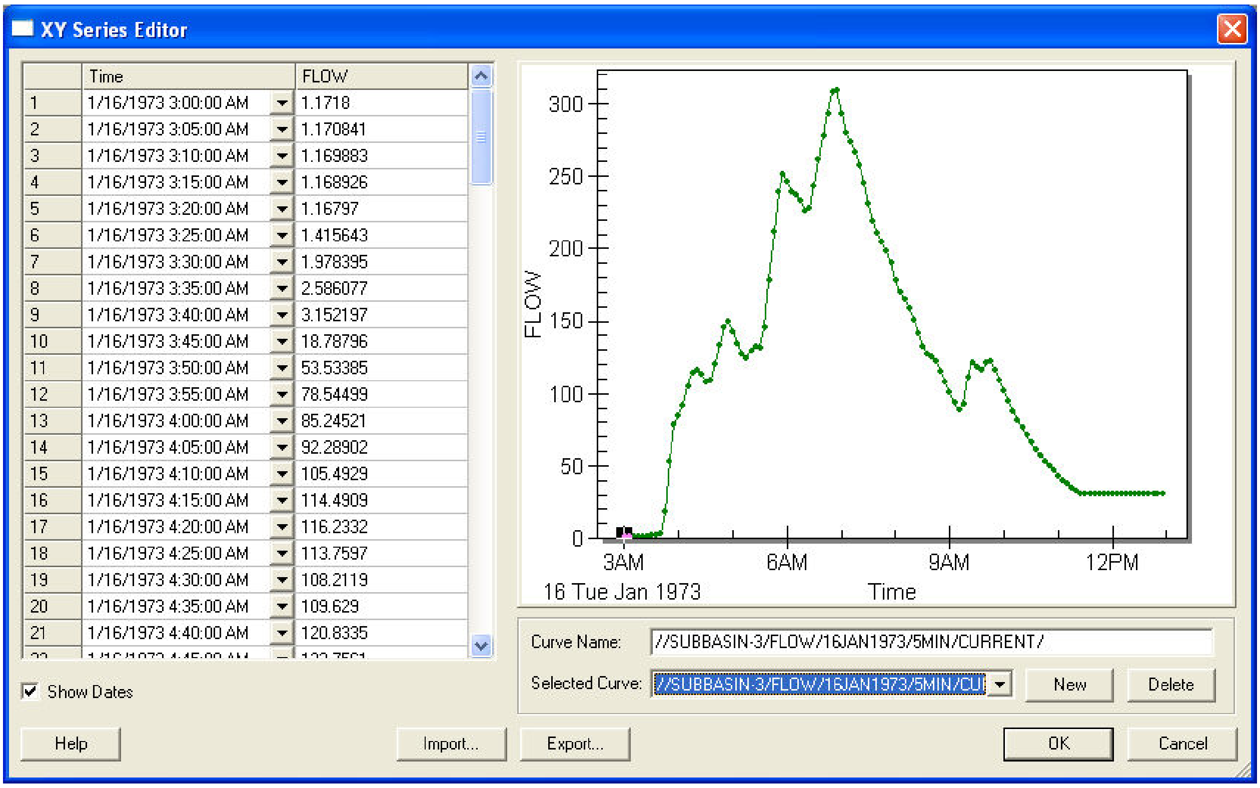


Most hydrographs computed in HMS will have the following information for each part of the pathname:

- Part A: Nothing
- Part B: The sub-basin name ("SUBBASIN-3" in our case)
- Part C: The parameter value ("FLOW" will correspond with the hydrograph for the basin)
- Part D: The start date for the hydrograph
- Part E: The time increment for the hydrograph
- Part F: The HMS Run ID for the hydrograph ("CURRENT" in our case)

Therefore, the entire pathname that we are looking for is "//SUBBASIN-3/FLOW/16JAN1973/5MIN/CURRENT/".

In this dialog, scroll down to find the DSS pathname associated with the basin that you are interested in and select the **Edit/Plot Data...** button. This will bring your data into the XY series editor where you can see a plot of your output results as shown below.



Related Topics

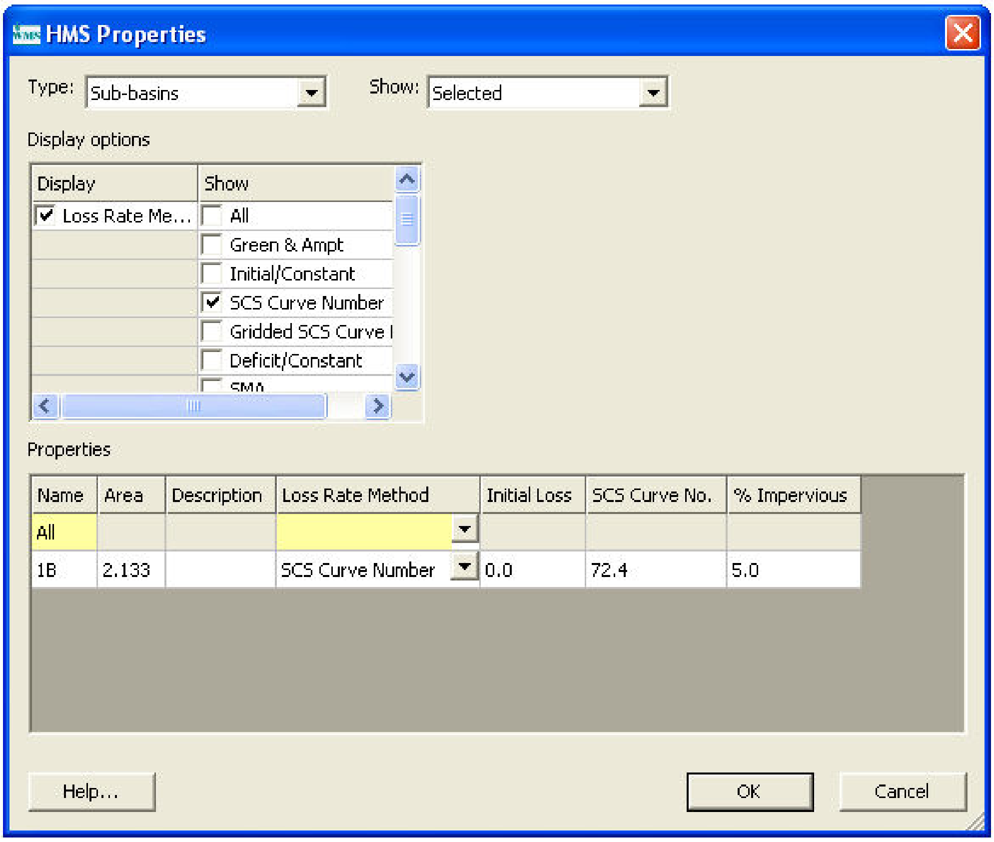
- HMS Overview
- Running HMS

HEC-HMS Properties

HMS includes the following hydrologic units which can be used to define a model:

- Basins
- Junctions
- Reaches
- Diversions
- Reservoirs
- Sources
- Sinks

All of the parameters (properties) are edited from the same *HMS Properties* dialog shown below.



The HMS documentation can be reviewed for detailed model formulation and behavior of the various properties, but different functions of the dialog are defined below:

Type – Specifies which type of hydrologic unit parameters you wish to view/edit.

Show – You can choose to show/edit the parameters of the selected or all hydrologic units of the specified type.

Display Options – The display options determine which properties (Display column) and which methods (Show column) are displayed/edited in the Properties spreadsheet.

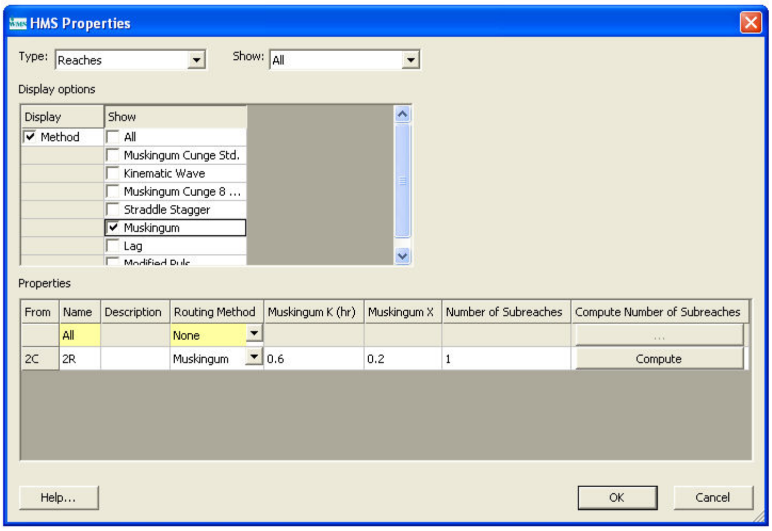
Properties – The properties of the selected (or all) hydrologic units are displayed as a spreadsheet for editing in this area. Only the properties and methods as determined in the display options are visible for review and editing. While more than one method may have properties displayed, a method for simulation must be chosen as one of the properties.

Related Topics

- Sub-Basins
- Junctions
- Reach Routing
- Reservoirs
- Diversions
- Sources
- Sinks

HEC-HMS Reach Routing

Reach routing will lag and attenuate the hydrograph computed (combined) at an outlet according to the reach properties defined. To define reach parameters, you should choose the **Edit Parameters** command from the *HEC-HMS* menu to obtain the *HMS Properties* dialog, shown below, and then select the *HMS Property type* to be *Reaches*. You can either show/edit properties for only the selected reach(es), or for all reaches in the model. You select a reach in WMS by selecting the upstream outlet of the reach since it is from this point hydrographs are routed.



You can choose to *Display properties of the reach* and then show one or more methods in the properties table. The method that HMS will use and the associated parameters are all edited from the properties table.

Lag

The Lag method simply lags the hydrograph without any attenuation. The only parameter for this method is the lag time with its accompanying units (minutes or hours).

Muskingum

The Muskingum method is dependent primarily upon an input weighting factor for the reach. The required parameters are as follows:

- Number of sub-reaches (time steps that hydrograph will be in the reach).
- Muskingum K coefficient in hours for entire reach.
- Muskingum x (weighting) coefficient.

Using the basin data computed by WMS when a TIN or DEM is used to delineate the watershed, the K coefficient and number of sub reaches can easily be estimated. K is essentially the travel time for the reach, which can be estimated by noting the length of the stream segment (you can see this by displaying in the Muskingum Cunge method) and multiplying by an assumed channel velocity (1-5 ft/s would be appropriate for most natural channels).

Of course you will need to convert the estimated travel times from seconds to hours before entering it into the K property field. The sub-reaches value is the number of time steps the flood wave is in the channel and can be determined by dividing K by the computational time step found in the Job Control dialog (again be sure that units are consistent). A button exists in the Muskingum K property field so that these computations can be done directly within WMS.

Modified Puls

The modified Puls method uses a storage routing technique, or level-pool routing. You must enter the storage, outflow relationship as well as the number of sub-reaches and initial condition.

Muskingum Cunge Std., Muskingum Cunge 8 Point, or Kinematic Wave

The Muskingum-Cunge and Kinematic Wave methods are defined with essentially the same parameters.

- Channel length.
- Energy grade slop (generally you can use channel slope).
- Manning's roughness.
- Characteristic channel shape.
  - Type, base width and side slope of a prismatic cross section or.
  - 8 point cross section defining the right over bank, center, and left over bank.

Straddle Stagger

This is a seldom used method which requires the number of ordinates to lag and the duration.

Related Topics

- Editing HMS Properties
- Sub-Basins
- Reservoirs
- Junctions
- Diversions

HEC-HMS Reservoirs

Reservoir routing is similar to the Modified-Puls reach routing method. The difference is that relationships between elevation-storage and elevation-outflow can be used to determine the storage-outflow curve. This input can either be in the form of:

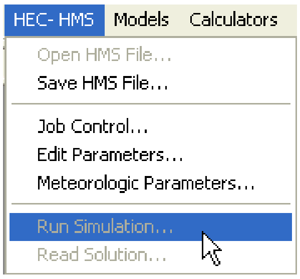
- Storage-Outflow (same as reach routing)
- Elevation-Storage-Outflow
- Elevation-Area-Outflow

You will also have to establish the initial conditions (whether there is storage), and the number of sub-reaches.

Related Topics

- Defining HMS Properties
- Reach Routing
- Sub-Basins
- Junctions

HEC-HMS Run Simulation



WMS Version 8.0 does not support running HMS simulations directly from WMS. Therefore, this menu item is dimmed. The WMS developers will support this capability in future versions or in later updates of WMS 8.0. To run a simulation, you first need to save the HMS file from WMS. Then, start HMS and run the simulation in HMS. Finally, you can read the solution back into WMS by opening the HMS DSS solution file from WMS.

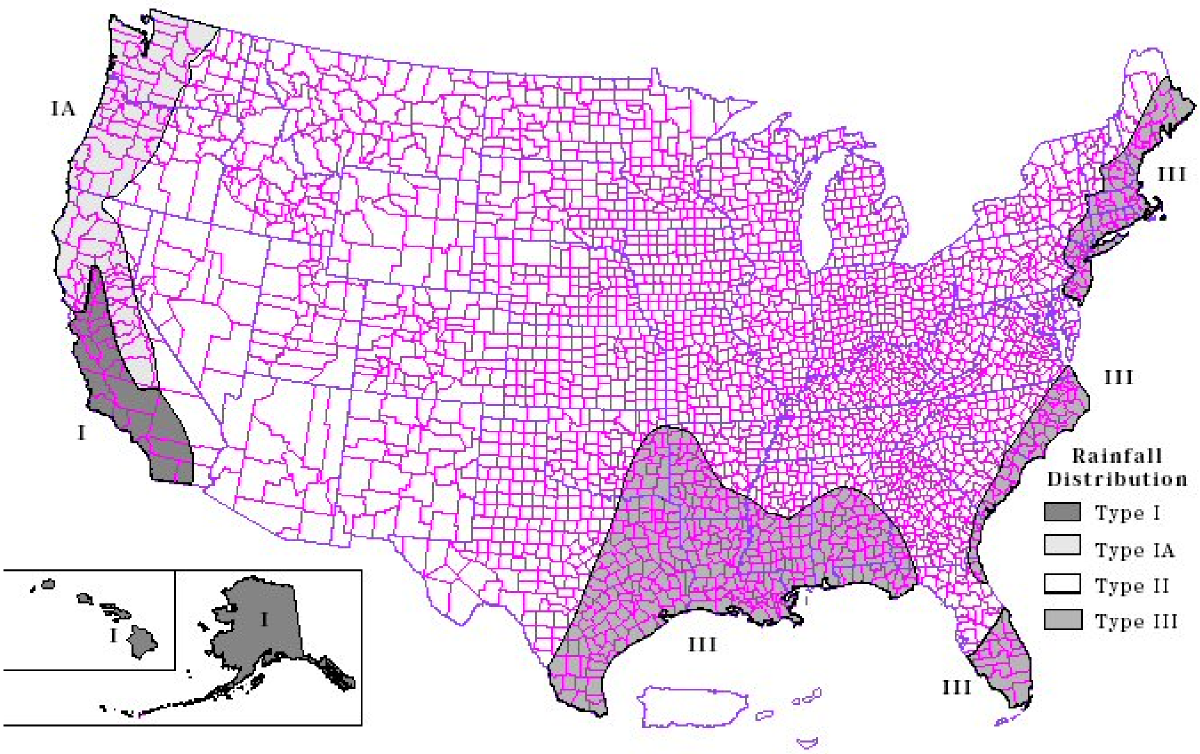
Related Topics

- HMS Overview
- Post Processing HMS Results

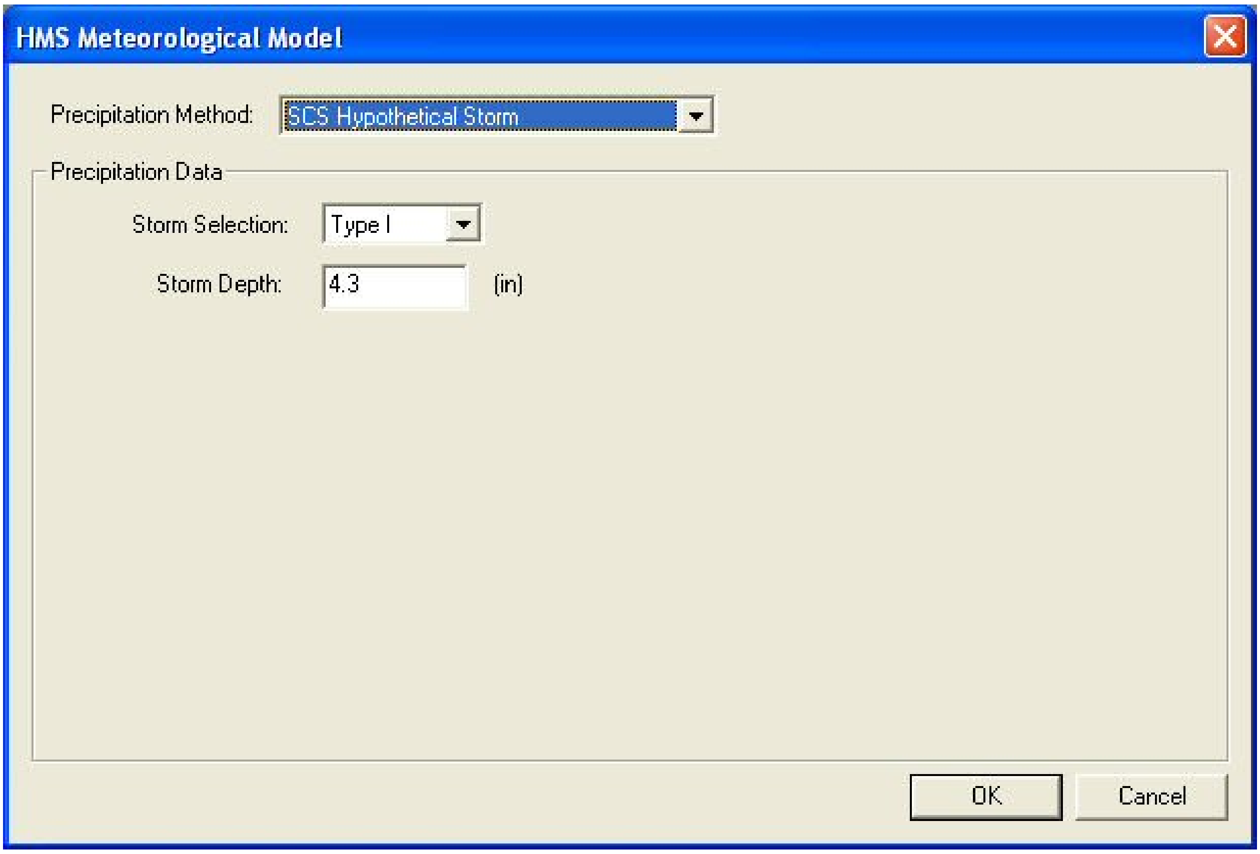
HEC-HMS SCS Hypothetical Storm

"The SCS hypothetical storm method implements the four synthetic rainfall distributions developed by the Natural Resources Conservation Service (NRCS) from observed precipitation events. Each distribution contains rainfall intensities arranged to maximize the peak runoff for a given total storm depth. The four distributions correspond to different geographic regions (Soil Conservation Service, 1986)." (HEC-HMS User's Manual)

The only data requirements for this method are the storm type and the storm depth. The storm depth can be determined for different areas of the United States by looking at the following map (Figure B-2 from Appendix B of the TR-55 manual).



This method does not perform any depth-area reductions. The HMS Meteorological Model dialog for this method is shown below:



Related Topics:

- Defining the Meteorological Model

HEC-HMS Saving an HMS File

Saving an HMS file is a simple task. To save an HMS file, just select the **Save HMS File** command from the *HEC-HMS* menu. Enter a file name ending in ".hms". The HMS project file and all the files (*.basin, *.control, *.map, and *.met) will be saved based on the data you have entered in WMS.

HEC-HMS Sinks

If you wish to run a model that is interior to a larger watershed then you can define a sink at the outlet of your a model. A sink can be defined either as a hydrograph or a constant flow rate. Sinks can only be defined at outlet points that do not have downstream reaches.

Before you can define a sink the *HMS properties* dialog you must first add the sink to the outlet. This is done using the **Add→Sink** command from the *Tree* menu.

Related Topics

- Defining HMS Properties
- Adding a Sink
- Sources
- Building Trees

HEC-HMS Sources

If you wish to run a model that is interior to a larger watershed then you can define a source at the headwaters of a stream within your a model. A source can be defined either as a hydrograph or a constant flow rate. Sources can only be defined at outlet points that do not have upstream reaches or basins defined, i.e. the stream headwaters.

Before you can define a source in the *HMS properties* dialog you must first add the source to the outlet. This is done using the **Add→Source** command from the Tree menu.

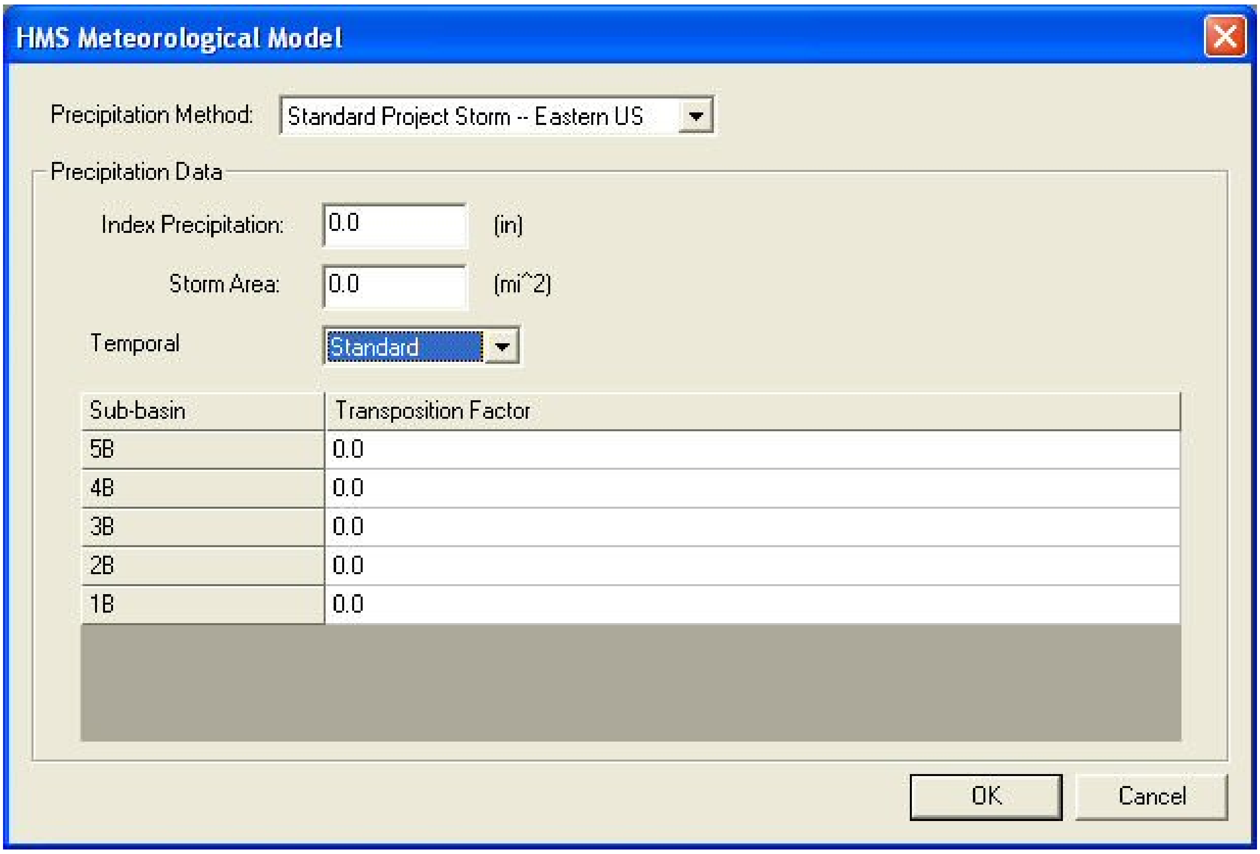
Related Topics

- Defining HMS Properties
- Adding a Source
- Sinks
- Building Trees

HEC-HMS Standard Project Storm

"The standard project storm method can be used to compute precipitation for estimating the standard project flood (Corps of Engineers, 1952). The method is appropriate for watersheds in the United States east of 105° west longitude with an area less than 1,000 square miles. New methods in risk-based analysis have generally replaced the standard project flood criteria." (HEC-HMS User's Manual [1])

The *HMS Meteorological Model* dialog for the Standard Project Storm method is shown below:



You can get the index precipitation value (also known as the probable maximum index precipitation) from HYDROMET Report 33. It can also be estimated from plates contained in EM 1110-2-1411 (US Army Corps of Engineers) and represents the total precipitation depth for the storm. The storm area is the total drainage area at the point where the standard project flood is to be estimated. For the temporal distribution type, the standard storm distributes the precipitation according to the criteria outlined in EM 1110-2-1411. The "SWD" option distributes precipitation according to the Southwestern Division criteria (table 3.1 of the HEC-1 reference manual).

Each basin requires a transposition factor. The transposition factor for each subbasin can range from 0.80 to 1.40 and is multiplied by the index precipitation to determine the mean areal precipitation. The transposition factor can be determined from the SPS isohyetal pattern contained in EM 1110-2-1411 and a map of the watershed.

Reference

USACE (1952) Standard project flood determinations. EM 1110-2-1411. Washington, DC.

Related Topics:

- Defining the Meteorological Model

References

[1] http://www.hec.usace.army.mil/software/hec-hms/documentation/HEC-HMS_Users_Manual_3.5.pdf

HEC-HMS Sub-Basins

Sub-basins are one of the basic hydrologic units that can be defined in an HMS model. To simulate runoff for a sub-basin, base flow, loss, and transform properties must be defined. This is done by first selecting the Sub-basins hydrologic unit type from the HMS Properties dialog, and then turning on the display of the different properties and methods that you want to define. The properties are then edited in the properties table.

Unlike HEC-1, meteorological data are not defined as a sub-basin property, but rather as part of the meteorological parameters of the HMS simulation.

Related Topics

- Editing HMS Properties
- Base Flow
- Loss Rates
- Transform

HEC-HMS Transform

Clark

The parameters for the Clark method are as follows:

- Time of concentration in hours for the unit hydrograph. Several different equations exist for determining the time of concentration. The list of basin geometric attributes computed automatically when basins have been delineated from a terrain model can be useful in many of these equations. These attributes can be viewed from within the *Unit Hydrograph Method* dialog by choosing the **View Basin Geometrical Attributes** button. Time of concentration can be computed from one of several equations using these attributes, or by using a time computation coverage. These options are accessed from the Compute Parameters – Basin Data] and Compute Parameters – Map Data buttons respectively.
- Clark storage coefficient in hours.

A synthetic time area curve is used in the HMS Clark method, but you can use the HEC-1 model if you want to derive a time area curve that is more specific to your sub-basin.

Kinematic Wave

Distributed outflow from a basin may be obtained by utilizing combinations of three conceptual elements: overland flow planes, collector channels, and a main channel. These elements can be defined if the kinematic wave option is specified.

The first and second kinematic wave records can be used to distinguish between different properties such as pervious/impervious (grass/pavement). For each record, the following parameters can be supplied.

- L – Overland flow length.
- S – Representative slope.
- N – Manning's roughness coefficient.
- A – Percentage of sub basins area that this record represents (The total of the two records must sum to 100).

In addition to the kinematic wave records, collector channels and a main channel must be defined. The main channel must be defined, whereas the two collector channels are optional. The following parameters are used for each channel:

- L – Channel length.
- S – Channel slope.
- N – Manning's roughness coefficient for the channel.
- CA – Contributing area to the channel.
- SHAPE – The characteristic shape of the channel.
- WD – Channel bottom width or diameter.
- Z – Side slopes if the channel type requires it.

Snyder

Parameters for the Snyder unit hydrograph are as follows:

- Lag time in hours – Several different equations have been published to determine the lag time of a basin. Many of them use some of the geometric attributes computed automatically when a terrain model is present. These attributes can be viewed by choosing the **View Basin Geometrical Attributes** button. Lag time can be computed from one of several equations using these attributes, or by using a time computation coverage. These options are accessed from the **Compute Parameters – Basin Data** and **Compute Parameters – Map Data** buttons respectively (see Computing Travel Times).
- Peaking coefficient.

A synthetic time area curve is used in the HMS Snyder method, but you can use the HEC-1 model if you want to derive a time area curve that is more specific to your sub-basin.

SCS

Parameters for generating a unit hydrograph using the SCS dimensionless method include:

- SCS lag time in hours – Several different equations have been published to determine the lag time of a basin. Many of these use some of the geometric attributes computed automatically when a TIN is present. Lag time can be computed from one of several equations using these attributes, or by using a time computation coverage. These options are accessed from the Compute Parameters – Basin Data] and Compute Parameters – Map Data buttons respectively.

User Specified S-Graph

The user specified S-Graph method allows you to enter the exact specification of the empirical relationship between on unit of excess rainfall and the resulting direct runoff. The relationship is defined using a dimensionless s-graph in the time series editor.

User Specified Unit Hydrograph

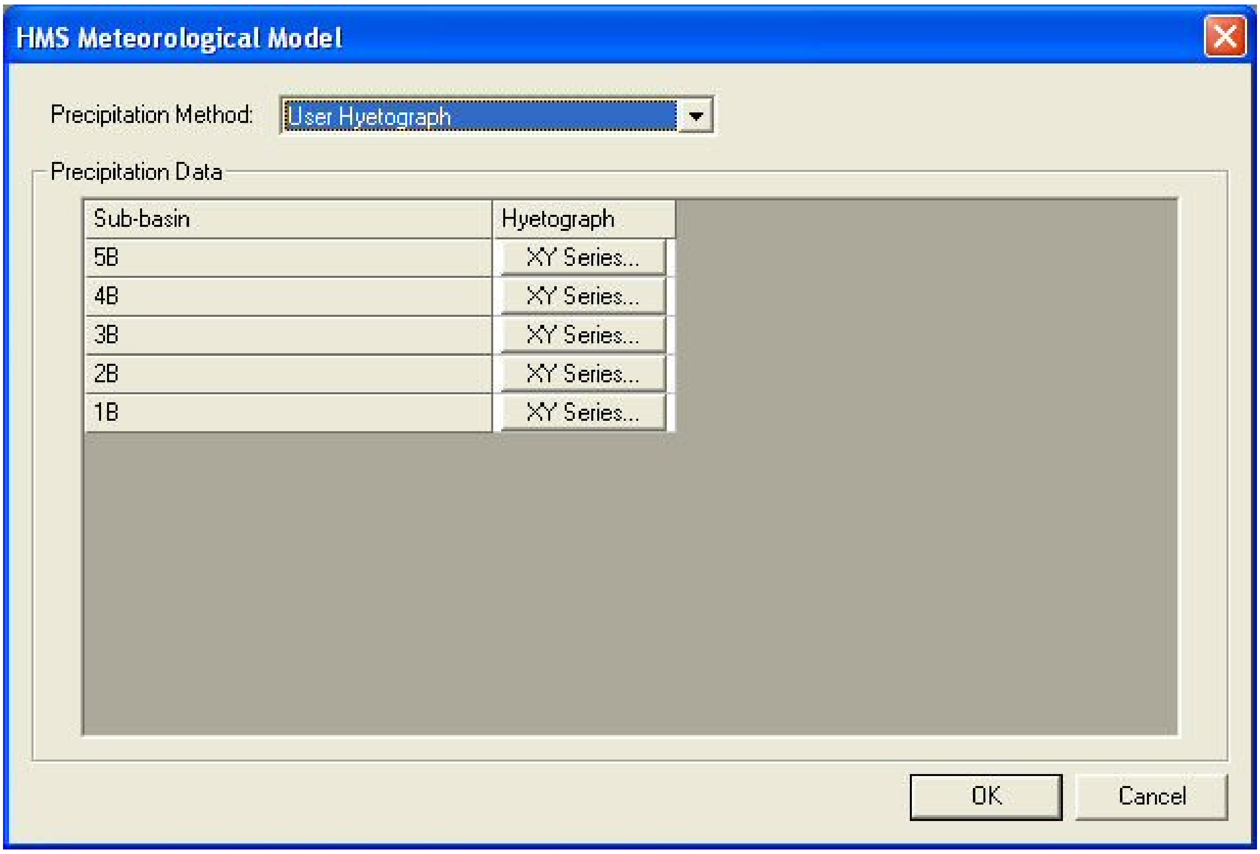
The user specified unit hydrograph option allows you to define the exact unit hydrograph for the basin in the time series editor.

Related Topics

- Editing HMS Properties
- Sub-Basins
- Base Flow
- Loss-Rates

HEC-HMS User Hyetograph

The user hyetograph method can be used if a hyetograph is known for each sub-basin of the watershed you are modeling. You can get this hyetograph from recorded data or from any other method. You can enter the same or different hyetographs for each basin, but each basin can only have a single hyetograph. Define the hyetograph by clicking on the **XY Series** button in the *HMS Meteorological Model* dialog shown below, and define the hyetograph in the XY series editor:



Related Topics:

- Defining the Meteorological Model

6.5. Hydrologic Engineering Center-River Analysis System (HEC-RAS)

HEC-RAS

HEC-RAS is a one-dimensional model for computing water surface profiles for steady state or gradually varied flow. HEC-RAS supports networks of channels and is capable of modeling subcritical, supercritical, and mixed flow regime profiles. HEC-RAS is able to model obstructions in the flow path.

The WMS interface to HEC-RAS includes tools for setting up the river networks and cross-sections as well as post-processing capabilities. WMS reads and writes the HEC-GeoRAS GIS import format.

The first step to using HEC-RAS with WMS is to setup the river schematic. For information on the tools and processes to setup a river schematic, see the 1D River Hydraulic Module. Specific attributes for a HEC-RAS simulation include:

Assigning Materials

Roughness values for HECRAS are tied to materials in WMS which must be assigned as line properties inside the cross-section database. To assign roughness values to the materials, choose **HecRas | Material Properties**. Also, the property type that is being used for the material values must be specified in the model control dialog. This dialog is accessed by selecting **HecRas | Model Control** from the menu.

Running HEC-RAS

Before loading the data into HEC-RAS, the simulation needs to be saved in HEC-RAS GIS export format. This is done by using **File | Save As** and changing the type to HEC-RAS GIS export format. The file can then be loaded into HEC-RAS by choosing **File | Import From GIS Format** inside the geometry editor. See HEC-RAS documentation [1] for information about setting up boundary conditions and running the model.

Once the solution has been generated the data may be used in WMS for post-processing. The data must first be exported to GIS format from within HEC-RAS (Under *File Menu* choose *export GIS format*). This file can be opened inside of WMS. Once this solution data has been loaded into WMS, plots can be generated from the solution. For information on how to use the post-processing tools that are found in WMS, see the 1D hydraulic Module document.

HEC-RAS Start

WMS allows model linkages between HEC-1, HEC-RAS, and flood plain delineation for the purpose of performing stochastic simulation. These simulations can be developed from either of the three modeling menus. You can read more about the stochastic simulation in the Stochastic Modeling article.

HEC-RAS Load Project

While the WMS interface does not allow you to develop a complete HEC-RAS model, you can launch the HEC-RAS compute engine, or the GUI, by having a project handle in WMS. The **Load Project** command allows you to define the current working HEC-RAS project and will prompt you for the filename of an existing HEC-RAS project.

HEC-RAS Run Simulation

The **Run Simulation** command allows you to start a HEC-RAS project from within WMS. WMS does not provide a complete GUI to HEC-RAS and so the HEC-RAS interface itself must be used to finish model development. However, once the model has been completed it can be run from within WMS (this is primarily to facilitate flood plain delineation and stochastic simulations) by specifying the project name.

HEC-RAS Delete Simulation

The **Delete Simulation** command deletes the link WMS maintains to the current HEC-RAS simulation project file.

HEC-RAS Plot Solution

The **Plot Solution** command allows you to send a solution to the HEC-RAS GUI for plotting.

HEC-RAS Read Solution

The **Read Solution** command reads the water surface elevation for each cross section after an HEC-RAS model run. A scatter point set is created with a scatter point at the intersection of each cross section and the centerline. The **Interpolate Results** command in the *River Tools* menu of the Map module can be used to create a more dense set of scatter points that are more suitable for flood plain delineation.

Related Topics

- 1D Hydraulic Modeling
- Export GIS File
- Import GIS File
- Managing Cross Sections
- Bridges/Culverts
- Area Property Coverage
- Extracting Cross Sections
- Recompute All Stations

References

[1] http://www.hec.usace.army.mil/software/hec-ras/

HEC-RAS Bridges/Culverts

You can either add bridges directly to your HEC-RAS model or you can setup a bridge as a cross section in WMS. In either case, when defining a bridge or a culvert, it is recommended that you add 2 cross sections upstream of the bridge or culvert and 2 cross sections downstream in WMS. You can also define a 5th cross section in WMS that represents a bridge itself or culvert itself, and it's a good idea to do this. You can either put this bridge/culvert cross section in WMS as a cross section or just define it in HEC-RAS.

When creating a cross section in WMS that represents a bridge or a culvert, it does not matter where the cross section is placed as long as it is somewhere between the two upstream and two downstream cross sections defined for the bridge or culvert. After creating the cross section representing the bridge or culvert, you must select the option in the cross section attributes dialog that makes it a bridge/culvert.

The floodplain delineation will work whether you enter your culverts or bridges in HEC-RAS or as cross sections in WMS, but you must define your 2 upstream and 2 downstream cross sections in WMS for WMS to read the water surface elevations at these locations from HEC-RAS.

You will have troubles reading the water surface elevations back into WMS if you add a cross section in HEC-RAS that is not in the WMS model.

Related Topics

- 1D Hydraulic Modeling
- Managing Cross Sections
- Extracting Cross Sections
- HEC-RAS Overview
- Recompute All Stations

HEC-RAS Model Control

When materials are mapped to cross sections a material ID is stored for the line segment properties. In order to properly link the material ID with a roughness when exporting the GIS file for HEC-RAS an Area Property coverage must be linked to a cross section database. The *Model Control* dialog allows you to make this specification. If no definition has been provided at the time you try to export your GIS file, you will be asked to make the association before proceeding with the export.

Related Topics

- Export GIS File
- Area Property Coverage

HEC-RAS Stochastic Modeling

WMS allows model linkages between HEC-1, HEC-RAS, and flood plain delineation for the purpose of performing stochastic simulation. These simulations can be developed from either of the three modeling menus. You can read more about the stochastic simulation in the Stochastic Modeling article.

Related Topics

- HEC-1 Modeling
- Flood Plain Delineation
- Developing Stochastic Simulations

HEC-RAS Unsteady Modeling

This topic explains how to create and run an HEC-RAS unsteady model. The first step for creating an unsteady model is to setup and export an HEC-RAS hydraulic model in the usual way. After you exported your HEC-RAS model, perform the following steps to set up the unsteady model and post-process the solution inside of WMS:

1. Go to HEC-RAS and set up a steady state simulation that runs to completion.
2. After your steady state simulation is set up, you need to define the unsteady model. One of the main requirements of an unsteady model is to provide a hydrograph as a boundary condition for the unsteady model. You can obtain this hydrograph from one of the hydrologic models in WMS, from measured data, or from any other method. Enter this hydrograph as a boundary condition in HEC-RAS, eliminating any 0 values from the hydrograph.
3. Run the unsteady model, continuing to fix any errors until the model runs to completion. Save your HEC-RAS project and close HEC-RAS.
4. Go back to WMS and (from the river module) read the solution for the HEC-RAS project (*.prj) file. All the time steps from the unsteady model will be read into WMS. To view the results of the unsteady simulation, you will now need to delineate the floodplain at each of these time steps. This can be done in the same way as delineating a floodplain from a steady state simulation.
5. Interpolate the water surface elevations in the 1D Hydraulic Centerline and the 1D Hydraulic Cross Section coverages. This will interpolate the water surface elevations at all time steps.
6. Delineate the floodplain. The floodplain will be delineated for all time steps.
7. Select the flood depth dataset from the floodplain delineation. Set the contour options as desired and run a film loop on this dataset.

Related Topic

- Hydraulic Modeling
- Export GIS File
- Read Solution
- Overview of Flood Plain Delineation
- Interpolate Results
- Delineate Flood Plain
- Setting up Film Loops

Recompute All Stations

The **Recompute All Stations** command updates river station information for the 1D hydraulic centerline and the 1D hydraulic cross section coverages. If you have made *any* changes to centerline arcs in the 1D hydraulic centerline coverage that changes the cross section stations in the 1D hydraulic cross section coverage, you should use this command. For example, if you have some cross sections with negative station values, using this command should fix the cross section stations. To update the cross section stations, first use this command in the 1D hydraulic centerline coverage and then use this command in the 1D hydraulic cross section coverage. This will insure your cross section stations are set to the correct values. This command does the following in the 1D hydraulic centerline and 1D hydraulic cross section coverages:

1D Hydraulic Centerline Coverage

This command recomputes the length and stationing along any centerline arcs in a 1D hydraulic centerline coverage.

1D Hydraulic Cross Section Coverage

This command recomputes the cross section stations for a 1D hydraulic cross section coverage.

Related Topics

- 1D Hydraulic Modeling
- Export GIS File
- Import GIS File
- Managing Cross Sections
- Area Property Coverage
- Extracting Cross Sections

6.6. Hydrological Simulation Program—FORTRAN (HSPF)

HSPF

Introduction

Hydrological Simulation Program – FORTRAN (HSPF) is an analytical tool designed to allow simulation of hydrology and water quality in natural and man-made systems. HSPF is used to apply mathematical models to simulate the movement of water, sediment and other constituents through watersheds. This analysis helps predict possible environmental problems in the watershed. With the growing need to care for and monitor the effects of man on the environment, it became apparent that a method for rapid analysis of those effects was needed.

The WMS interface creates a way for the user to input a multitude of parameters into the input file and then to run HSPF. The input parameters are linked to all the tools within WMS including automated basin delineation and parameter definition using a graphical user interface.

Using HSPF With WMS

WMS does not currently have any post-processing capabilities for the HSPF model. When WMS is installed, there are 4 programs installed under the BASINS folder in the *Windows Start* menu: **GenScn**, **WDMUtil**, **WinHSPF**, and **WinHSPFLt**. **GenScn** is used for generating various types of scenarios and does some post-processing. You might find some value in looking at the different capabilities for scenario generation and post-processing in **GenScn**. **WDMUtil** should be used for post-processing after running an HSPF simulation. Read the WDM file from the HSPF simulation into **WDMUtil** and you can view the "External Targets" (output) from the HSPF simulation. **WinHSPF** and **WinHSPFLt** can be used to run HSPF. These are both windows-based programs with a graphical user interface. **WinHSPF** has more tools for analyzing any errors from the HSPF run and for visualizing and editing your model, but you can also run HSPF using the **WinHSPFLt** program. The DOS version of HSPF is no longer distributed with WMS. As the last section of the HSPF tutorial says, you need to close WMS after saving your HSPF .uci file before running the file in WinHSPF or WinHSPFLt because WMS keeps the .wdm file open. HSPF cannot run the model unless the .wdm is closed in all other programs. The versions of WinHSPF, WinHSPFLt, WDMUtil, and GenScn are the same as the software distributed by EPA's BASINS program.

Related Topics:

- Getting Started
- Automated Land Segmentation
- Reading Existing Files
- Saving HSPF UCI Files
- Running HSPF

HSPF Automatic Segmentation of Land Segments

The core functionality of WMS allows the automated delineation of watershed and sub-basins. Additional tools have been developed as part of the HSPF interface to allow automated breakdown of sub-basins to land-use based segments. These segments can then be modeled as PERLND or IMPLND segments in HSPF. Once the HSPF simulation has been initialized, these tools can be accessed.

The division of sub-basins to segments is performed as a GIS based combination of two data layers: a watershed data layer which contains basin boundary polygons, and a land use layer which contains land-use type polygons or gridded data. The two layers, or coverages, are compared to determine the land-use types which fall into each basin boundary. Then a segment is created for each land-use type in the basin. The area, type of segment (IMPLND or PERLND), and other parameters can be determined in this operation.

Automated segment generation in WMS is driven by the *Compute GS Attributes* dialog. Prior to invoking this dialog:

- A watershed must be delineated and basin data computed.
- A land use coverage or grid must be read into WMS.
- HSPF data must be initialized in the HSPF Global Options dialog.

The dialog is opened by selecting the **Compute GIS Attributes** option from the *Calculators* menu in the Hydrologic Modeling module.

To generate the HSPF segments:

- Select **HSPF Segments** from the *Computation Type* group.
- Make sure the units of the watershed and land use data are set correctly by selecting the **Units** button and reviewing the settings.
- Ensure that the land use coverage or grid is specified correctly.
- Import a table that relates land use to HSPF parameters. This table is a simple text file that lists land-use ID, description, and pervious attribute as shown below.

    11, “RESIDENTIAL”, “pervious”

    12, “COMMERCIAL AND SERVICES”, “impervious”

    13, “INDUSTRIAL”, “pervious”

    16, “MXD URBAN OR BUILT-UP”, “impervious”

    17, “OTHER URBAN OR BUILT-UP”, “pervious”

    21, “CROPLAND AND PASTURE”, “pervious”

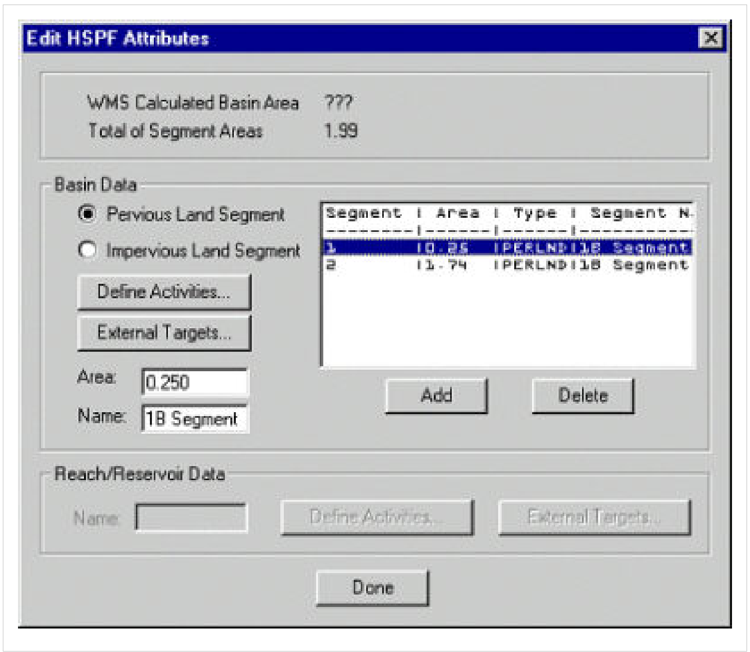
    32, “SHRUB & BRUSH RANGELAND”, “pervious”
- Click **OK**

This operation will create a segment for each unique land-use type in each basin, calculate the area, and assign the appropriate type (pervious or impervious) and name. These segments can then be modeled separately with HSPF.

Related Topics

- Overview of HSPF
- The HSPF Interface
- Basin Delineation from DEMs
- Basin Delineation from TINs
- Basin Delineation from Feature Objects
- Entering HSPF Parameters

HSPF Edit Parameters



Attributes or parameters for all land and reach segments of the topologic tree are viewed and/or edited using the *Edit HSPF Attributes* dialog. This dialog is accessed by selecting the **Edit HSPF Attributes** command from the *HSPF* menu or by double-clicking on a basin or an outlet when HSPF is the active model.

If a basin or multiple basins are selected on the topologic tree, the *Basin Data* portion of this dialog will be active and a list of all segments in the selected basins will be displayed in the text window. Selecting a segment from the widow will active the fields and buttons to edit/view the associated parameters. If an outlet is selected on the topologic tree, the *Reach/Reservoir Data* section of the dialog will be active. The parameters associated with the selected reach/reservoir may be viewed/edited using the active buttons.

Once the *Edit HSPF Attributes* dialog is open, it will remain open until the user selects the **Done** button. This enables the user to switch between basins or outlets quickly and define the necessary parameters without closing and opening the dialog for each hydrograph station.

Basin Data

The portion of the *Edit HSPF Attributes* dialog box that handles basin data is mainly a starting point for entering the rest of the basin segment’s HSPF data. The text in the window lists the land segments within the selected basin. When a segment is selected within the text window, it becomes possible to edit the attributes of that segment. The area section, above the basin section, shows the sum of the areas of the individual basin segments as well as the area of the basin as calculated by WMS if a terrain model was used to calculate basin data. A radio group is located in the basin area of the dialog box to allow the user to specify whether the segment is pervious or impervious. The **Define Activities** button brings up dialog which allow the user to activate modules (i.e. PWAT, PQUAL, IWAT, etc.) for the selected segment and define the required parameters. The **External Targets** button allows the user to define the output for the selected segment as required by the EXT TARGETS block of HSPF.

Below the text window listing the basin segments, buttons may be found to add or delete segments within the basin. The new basin segments default to pervious land segments with no specified area. This a way to segment a basin manually if no land use data exist to perform automatic segmentation.

Reach/Reservoir Data

The portion of the *Edit HSPF Attributes* dialog box that handles the input data for the RCHRES module of HSPF is located at the bottom of the *Edit HSPF Attributes* dialog box. Within that small section of the dialog box, the name of the RCHRES segment may be edited. The **Define Activities** button located next to the edit field accesses a dialog used to activate the desired modules of RCHRES. The **External Targets** button allows the user to define the output for the selected reach/reservoir as required by the EXT TARGETS block of HSPF.

Related Topics

- WMS:HSPF External Targets
- Mass Links
- WDM File Interaction

HSPF External Sources

Depending upon the options selected, HSPF requires input of a few or several time series. These time series are referred to as External Sources and should be located in a WDM file, which is specified in the HSPF Global Options dialog box. Not only must they exist in that file, but the time series must also be assigned to the proper segments according to the EXT SOURCES block of HSPF (see Section 4.6.2 of the HSPF manual[1]). The *Assign External Sources* dialog in WMS has been developed to allow the user to set up the required external sources for an HSPF run.

A button labeled **External Sources** is present in each of the HSPF module dialogs in WMS. Upon clicking this button, the *Assign External Sources* dialog box is activated. In this dialog, the user may select time series from a WDM file and assign them to the selected segment.

To assign a time series, simply select it from the list on the right side of the dialog, select the target member name appropriate for that series, select the other options desired from the boxes on the left side of the dialog, then click the **Assign** button to add that source to the list that appears in the window at the bottom of the dialog. To delete a source, select it from the list in the bottom window and click the **Delete** button. Also included in this dialog is an **Apply sources to segments** button. This button work in the same way the as the Apply parameters to segments button; the sources defined in the bottom window are copied to the chosen segments.

The sources specified for each segment of the HSPF model are written to the EXT SOURCES block of the UCI file when HSPF is run from WMS. For more information on the definition of external sources or the options available in this dialog, see Section 4.6.2 – EXT SOURCES block of the HSPF Manual.

Related Topics

- WDM File Interaction
- External Targets

References

[1] http://water.usgs.gov/software/HSPF/

HSPF External Targets

The output from HSPF is also in the form of time series data. These time series are referred to as External Targets and will be written to a WDM file, which is specified in the *HSPF Global Options* dialog box, each time HSPF is run. Thus, the user must specify what output is desired from HSPF and where to write the output. The *Assign External Targets* dialog in WMS has been developed to allow the user to set up the desired external targets for an HSPF run. For more information on the options for external targets, see Section 4.6.5 – EXT-TARGETS block of the HSPF manual.[1]

The *Assign External Targets* dialog may be opened from the *Edit HSPF Attributes* dialog by clicking on the **External Targets** button in the *Basin Data* box or the *Reach/Reservoir Data* box. The targets set up will be assigned to the selected segment.

To assign a target, the following options must be set: the group to which the source time series belong must be selected from the Group box; the source member name must be selected from the *Member Name* and subscript boxes; the units, access, aggregation, and transformation codes must be set; the multiplication factor should be entered; finally, the dataset to which to write the output data must be selected from the WDM file if the *Use existing dataset* option is chosen, or a dataset number, name, and tstype must be entered if *Create new dataset* is chosen. When the preceding options are set, clicking on the **Assign** button will add an external target to the list in the bottom window of the dialog. To delete a target, select it form the list in the bottom window and click the **Delete** button.

This dialog also contains an **Apply targets to segments** button. This button works that same as the **Apply parameters to segments** button explained above; the targets defined will be copied to the selected segments.

Related Topics

- WDM File Interaction
- External Sources

HSPF Getting Started

In order to begin an HSPF simulation in WMS, the HSPF data must be initialized for the watershed model. This is a simple procedure that involves the click of a button within the *HSPF Global Options* dialog box. Once the data is initialized, the user may access the various dialog boxes within the HSPF interface.

The following sections will describe the input dialogs involved in the interface and will explain the expected input parameters. It is recommended that the user consult the HSPF documentation to receive a complete description of how HSPF uses the parameters.

Related Topics

- Overview of HSPF
- Automatic Segmentation of HSPF Land Segments
- Global Options
- Entering HSPF Parameters
- HSPF Modules
- Basin Data
- Mass Links

HSPF Global Options

The *HSPF Global Options* dialog box is used to initialize the HSPF data for every hydrologic unit of the hydrologic tree and to set up parameters for the overall simulation. This dialog is opened by selecting the **Global Options** command from the *HSPF* menu.

The data for HSPF must be initialized before the user can access it; many features within the HSPF interface will be dimmed and remain dimmed until the user initializes the HSPF data. By clicking on the *Initialize HSPF Data* button within the *Global Options* dialog box, the user initializes all HSPF data within the watershed. The *Delete HSPF Data* button will remove all HSPF data from memory. This may be useful if the user wants to start over with HSPF input, or wants to use another model within WMS.

Once the HSPF data are initialized, several parameters are required in the *HSPF Global Options* dialog. Parameters required in the GLOBAL block of HSPF are input in this dialog. These include title, output levels, units flag, run flag, starting and ending times, and time step. Consult Section 4.1 of the HSPF manual[1] for details on these parameters.

Also in this dialog are fields to define the input and output filenames of the HSPF run. Note that currently WMS only supports one WDM file and one output file per simulation. If more that one WDM file contains input for the simulation, these files should be merged for use with WMS. The filename fields complete the data necessary for the FILES block of HSPF. Each file name should have the appropriate extension attached in these fields. These file names may be set to the same prefix by entering the prefix in the designated field and clicking on the **Update Prefix** button.

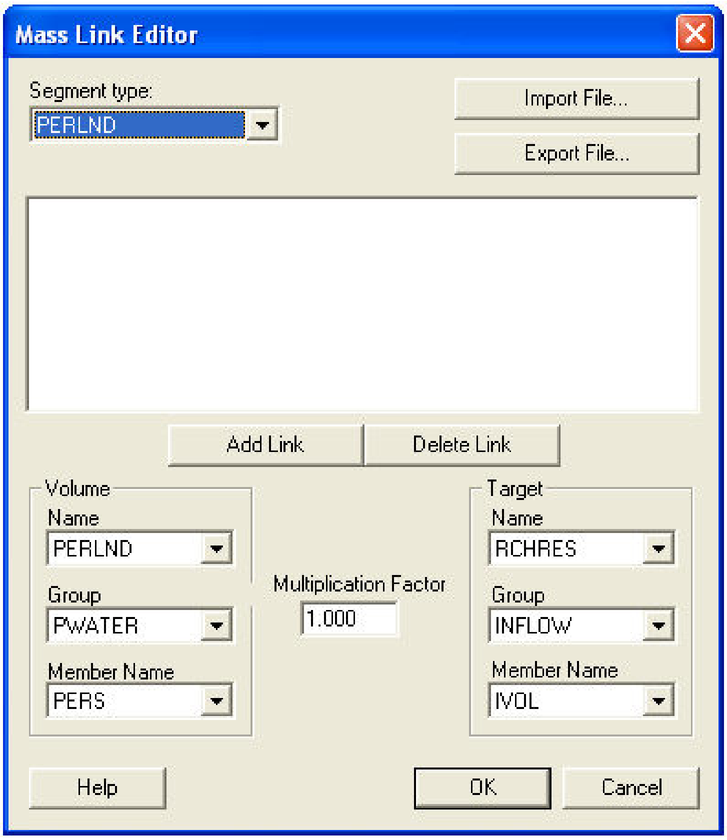
Related Topics

- Overview of HSPF
- The HSPF Interface
- Entering HSPF Parameters
- Basin Data
- Mass Links
- WDM File Interaction

HSPF Mass Links

The *Mass Link Editor* is a dialog box that is used to create links between two hydrologic units of the topologic tree and set up the MASS-LINK block of an HSPF run. Information the MASS-LINK block may be found in Section 4.6.4.2 of the HSPF manual[1]. In WMS, a list of mass links may be created for each segment type: PERLND, IMPLND, and RCHRES. The mass links for each segment type will be applied to all segments of that type in the model.

The *Mass Link Editor* dialog allows the user to add or delete mass links for each type of segment. To open the dialog, select **Mass Link Editor** from the *HSPF* menu in the Hydrologic Modeling module. To add, just select the segment type from the box in the upper left of the dialog, select the appropriate options in the various combo boxes, input a value as the conversion factor, then hit the **Add Link** button. To delete, select the line in the text window to be deleted and click the **Delete Link** button.



If a number of mass links are to be used for many simulations, it would be wise to save the mass links to a file that would be read in any time they are needed. The **Export** button allows the user to specify a filename and then the mass links are saved as an UCI-formatted ASCII text file. The **Import** button allows the user to read in a previously saved mass link file.

Related Topics

- Overview of HSPF
- Basin Data
- Reach/Reservoir Data

HSPF Modules

Input for the modules in HSPF is handled in WMS by a dialog box created for each module. These dialog boxes share many common features and function in the same basic way. A brief description of the functionality of these dialogs will simplify the input of parameters to HSPF.

First, many parameters in HSPF may be input as a single value or monthly values depending upon a flag that can be set in the module input. Such variables appear in the module dialogs with an edit field and a **Monthly Values** button. Either the edit field or the button will be highlighted to indicate what input is expected. The expected input can be changed by finding the appropriate check-box or drop-down box in the flag section of the dialog and setting it on/off. Similarly, some data tables may be required or omitted depending upon flags set in the module input. The check-boxes in the flags section of each dialog control the accessibility to fields and button for input to such tables. The correlation between flags and required input is covered in detail in the HSPF manual in the sections covering module input.

Next, in each dialog you will find a button entitled **External Sources** This button leads to a dialog where the input time series for the selected land or reach/reservoir segment may be selected and assigned from a WDM file according to the EXT SOURCES block format (see Section 4.6.2 of the HSPF manual). In a like manner, you will find an **External Targets** button that leads to a dialog where output time series to be generated by HSPF can be assigned to WDM datasets (Section 4.6.5 – EXT-TARGETS block of the HSPF manual[1]). These dialogs will be discussed in detail later in this document.

Next, each dialog contains a button entitled **Apply Parameters to Segments** This button accesses the *Apply Parameters to Segments* dialog. The purpose of this dialog is to allow the user to select one or several segments to which module input parameters will be copied. This can greatly reduce the time and effort necessary to develop the input for an HSPF run. The layout and use of the *Apply Parameters to Segments* dialog is fairly simple. The window on the left side displays the names of all segments of a given type in the model. For example, if the dialog were brought up from the *IWATER Parameters* dialog all of the impervious segments in the current model would be shown. The window on the right side of the *Apply Parameters to Segments* dialog lists the selected segments to which parameters will be copied. Segments may be moved to and from the two windows by selecting the segment, then clicking the move arrows located between the windows. By moving segments to the *Selected Segments* window and clicking on **OK**, a list of those segments is stored so that when **OK** is clicked in the calling dialog (such as IWATER Parameters) parameters are copied from the selected segment to all segments in the list. Clicking **Cancel** in the calling dialog will delete the list.

Finally, the default parameter values for each module will appear in the input fields for all data tables in the module input dialogs and help strings associated with each field give a reasonable range of values.

The items above are common to all module input dialogs in WMS. Further discussion of each dialog follows in the next section, but the general topics covered should be sufficient to allow you to use any of the dialogs.

Modules

Impervious Land Segment

Clicking the **Define Activities** button in the Edit HSPF Attributes dialog when an impervious segment is selected will open the *Impervious Land Activity* dialog. To activate a module, click the toggle box in the *Impervious Land Activity* dialog box. This will activate the button that leads to the dialog box for that module. This dialog allows activation of the following modules:

ATEMP

To access the *ATEMP Parameters* dialog box, press the **ATEMP** button in the *Pervious Land Activity* dialog box. The *ATEMP Parameters* dialog box lets the user enter the parameters for the ATEMP input table: ATEMP-DAT. Note that the dialog for ATEMP input for PERLND and IMPLND segments is the same. For information on the definition and function of specific variables in this module, see Section 4.4(1).2 – ATEMP Input of the HSPF Manual.[1]

SNOW

To access the *SNOW Parameters* dialog box, press the **SNOW** button in the *Pervious Land Activity* dialog box. The *SNOW Parameters* dialog box lets the user enter the parameters for the following SNOW input tables: ICE-FLAG, SNOW-PARM1, SNOW-PARM2, SNOW-INIT1, and SNOW-INIT2. Note that the dialog for SNOW input for PERLND and IMPLND segments is the same. For information on the definition and function of specific variables in this module, see Section 4.4(1).3 – SNOW Input of the HSPF Manual.[1]

IWATER|IWATER

To access the *IWATER Parameters* dialog box, press the **IWATER** button in the *Impervious Land Activity* dialog box. The *IWATER Parameters* dialog box lets the user enter the main parameters for the following IWATER input tables: IWAT-PARM1, IWAT-PARM2, IWAT-PARM3, and IWAT-STATE1. The following monthly value tables may also be input from this dialog if required: MON-RETN, MON-MANNING. Some parameters require a single value for the entire simulation and others are allowed to vary monthly, depending on the flags set in the IWAT-PARM1 table. For information on the definition and function of specific variables in this module, see Section 4.4(2).4 – IWATER Input of the HSPF Manual.[1]

SOLIDS

To access the *SOLIDS Parameters* dialog box, press the **SOLIDS** button in the *Impervious Land Activity* dialog box. The *SOLIDS Parameters* dialog box lets the user enter the parameters for the following SOLIDS input tables: SLD-PARM1, SLD-PARM2, and SLD-STOR. The following monthly value tables may also be input from this dialog if required: MON-SACCUM, MON-REMOV. For information on the definition and function of specific variables in this module, see Section 4.4(2).5 – SOLIDS Input of the HSPF Manual.[1]

IQUAL

To access the *IQUAL Parameters* dialog box, press the **include "global.sbh"** button in the *Impervious Land Activity* dialog box. This dialog box allows the user to specify up to 10 water quality constituents or pollutants in the outflows from an impervious land segment. The *IQUAL Parameters* dialog box lets the user enter the parameters for the following IQUAL input tables: NQUALS, IQL-AD-FLAGS, QUAL-PROPS, QUAL-INPUT. The following monthly value tables may also be input from this dialog if required: MON-POTFW, MON-ACCUM, and MON-SQOLIM. Some parameters require a single value for the entire simulation and others are allowed to vary monthly, depending on the flags set in the QUAL-PROPS table. For more information on these tables, see Section 4.4(2).7 – IQUAL input of the HSPF manual.[1]

The key to using this dialog is to note that the NQUAL field at the top of the box designates the total number of constituents to be modeled, while the *Editing Constituent Number* drop-down box indicates the constituent that is available for editing in the fields below.

Pervious Land Segment

Clicking the **Define Activities** button in the *Edit HSPF Attributes* dialog when a pervious segment is selected will open the *Pervious Land Activity* dialog. To activate a module, click the toggle box in the *Pervious Land Activity* dialog box. This will activate the button that leads to the input dialog box for that module. This dialog allows activation of the following modules:

ATEMP

To access the *ATEMP Parameters* dialog box, press the **ATEMP** button in the *Pervious Land Activity* dialog box. The *ATEMP Parameters* dialog box lets the user enter the parameters for the ATEMP input table: ATEMP-DAT. Note that the dialog for ATEMP input for PERLND and IMPLND segments is the same. For information on the definition and function of specific variables in this module, see Section 4.4(1).2 – ATEMP Input of the HSPF Manual.[1]

SNOW

To access the *SNOW Parameters* dialog box, press the **SNOW** button in the *Pervious Land Activity* dialog box. The *SNOW Parameters* dialog box lets the user enter the parameters for the following SNOW input tables: ICE-FLAG, SNOW-PARM1, SNOW-PARM2, SNOW-INIT1, and SNOW-INIT2. Note that the dialog for SNOW input for PERLND and IMPLND segments is the same. For information on the definition and function of specific variables in this module, see Section 4.4(1).3 – SNOW Input of the HSPF Manual.[1]

PWATER

To access the *PWATER Parameters* dialog box, press the **PWATER** button in the *Pervious Land Activity* dialog box. The *PWATER Parameters* dialog box lets the user enter the main parameters for the PWATER input tables: PWAT-PARM1, PWAT-PARM2, PWAT-PARM3, PWAT-PARM4 and PWAT-STATE1. The following monthly value tables may also be input from this dialog if required: MON-INTERCEP, MON-UZSN, MON-MANNING, MON-INTERFLW, MON-IRC, and MON-LZETPARM. Some parameters require a single value for the entire simulation and others are allowed to vary monthly, depending on the flags set in the PWAT-PARM1 table. For information on the definition and function of specific variables in this module, see Section 4.4(1).4 – PWATER Input of the HSPF Manual.[1]

SEDMNT

To access the *SEDMNT Parameters* dialog box, press the **SED** button in the *Pervious Land Activity* dialog box. The *SEDMNT Parameters* dialog box lets the user enter the parameters for the SEDMNT input tables: SED-PARM1, SED-PARM2, SED-PARM3, and SED-STOR. The following monthly value tables may also be input from this dialog if required: MON-COVER, MON-NVSI. Some parameters require a single value for the entire simulation and others are allowed to vary monthly, depending on the flags set in the SED-PARM1 table. For information on the definition and function of specific variables in this module, see Section 4.4(1).5 – SEDMNT Input of the HSPF Manual.[1]

PQUAL

To access the PQUAL Parameters dialog box, press the **PQUAL** button in the *Pervious Land Activity* dialog box. This dialog box allows the user to specify up to 10 water quality constituents or pollutants in the outflows from a pervious land segment. The *PQUAL Parameters* dialog box lets the user enter the parameters for the PQUAL input tables: NQUALS, PQL-AD-FLAGS, QUAL-PROPS, QUAL-INPUT. The following monthly value tables may also be input from this dialog if required: MON-POTFW, MON-POTFS, MON-ACCUM, MON-SQOLIM, MON-IFLW-CONC, and MON-GRND-CONC. Some parameters require a single value for the entire simulation and others are allowed to vary monthly, depending on the flags set in the QUAL-PROPS table. For more information on these tables, see Section 4.4(1).8 – PQUAL input of the HSPF manual.[1]

The key to using this dialog is to note that the NQUAL field at the top of the box designates the total number of constituents to be modeled, while the *Editing Constituent Number* drop-down box indicates the constituent that is available for editing in the fields below.

MSTLAY

To access the *MSTLAY Parameters* dialog box, press the '$include "global.sbh"button in the *Pervious Land Activity* dialog box. The MSTLAY Parameters dialog box lets the user enter the parameters for the following MSTLAY input tables: MST-PARM, MST-TOPSTOR, MST-TOPFLX, MST-SUBSTOR, and MST-SUBFLX. Note that fields for input to tables VUZFG, UZSN-LZSN, and MON-UZSN are also included in this dialog. These tables will be active and require input if MSTLAY is being modeled without PWATER active. For information on the definition and function of specific variables in this module, see Section 4.4(1).9 – MSTLAY Input of the HSPF Manual.[1]

PEST

To access the *PEST Parameters* dialog box, press the **PEST** button in the *Pervious Land Activity* dialog box. This dialog box allows the user to specify up to 3 pesticides on a pervious land segment. The PEST Parameters dialog box lets the user enter the parameters for the PEST input tables: PEST-FLAGS, PEST-AD-FLAGS, SOIL-DATA, PEST-ID, PEST-THETA, PEST-FIRSTPM, PEST-CMAX, PEST-SVALPM, PEST-NONSVPM, PEST-DEGRAD, PEST-STOR1, PEST-STOR2. The required combination of these tables varies depending on the PEST-FLAG table values. For more information on these tables, see Section 4.4(1).10 – PEST input of the HSPF manual.[1]

The key to using this dialog is to note that the NPST field at the top of the box designates the total number of pesticides to be modeled, while the Now Editing Pesticide drop-down box indicates the constituent that is available for editing in the fields below.

NITR

To access the *NITR Parameters* dialog box, press the **NITR** button in the *Pervious Land Activity* dialog box. The *NITR Parameters* dialog box lets the user enter the parameters for the NITR input tables: NIT-FLAGS, NIT-AD-FLAGS. In addition, buttons which lead to dialogs for input to the following tables are found in this dialog: NIT-FSTGEN, NIT-FSTGEN, NIT-ORGPM, NIT-AMVOLAT, NIT-CMAX, NIT-SVALPM, NIT-UPTAKE, MON-NITUPT, SOIL-DATA2, CROP-DATES, NIT-YIELD, MON-NUPT-FR1, MON-NUPT-FR2. The required combination of these tables varies depending on the NIT-FLAG table values entered. In addition, a button that leads to the *SOIL-DATA table* dialog is active in this dialog if the PEST module is inactive. For information on the definition and function of specific variables in this module, see Section 4.4(1).11 – NITR Input of the HSPF Manual.[1]

PHOS

To access the *PHOS Parameters* dialog box, press the **PHOS** button in the *Pervious Land Activity* dialog box. The *PHOS Parameters* dialog box lets the user enter the parameters for the PHOS input tables: PHOS-FLAGS, PHOS-AD-FLAGS, and PHOS-FSTGEN. In addition, buttons which lead to dialogs for input to the following tables are found in this dialog: PHOS-FSTPM, PHOS-CMAX, PHOS-SVALPM, PHOS-UPTAKE, MON-PHOSUPT, PHOS-YIELD, MON-PUPT-FR1, MON-PUPT-FR2, PHOS-STOR1, PHOS-STOR2. The required combination of these tables varies depending on the PHOS-FLAG table values entered. In addition, a button that leads to the SOIL-DATA, SOIL-DATA2, and CROP-DATES tables dialog is active in this dialog if the PEST module and NITR module are inactive. For information on the definition and function of specific variables in this module, see Section 4.4(1).12 – PHOS Input of the HSPF Manual.[1]

TRACER

To access the *TRACER Parameters* dialog box, press the **TRACER** button in the *Pervious Land Activity* dialog box, explained above. The *TRACER Parameters* dialog box lets the user enter the parameters for the TRACER input tables: TRAC-ID, TRAC-AD-FLAGS, TRAC-TOPSTOR, and TRAC-SUBSTOR. For information on the definition and function of specific variables in this module, see Section 4.4(1).13 – TRACER Input of the HSPF Manual. [1]

Reach Reservoir Segment

The *Reach/Reservoir Activity* dialog allows activation of the following modules: HYDR, ADCALC, CONS, HTRCH, SEDTRN, RQUAL. To activate a module, click the toggle box in the *Reach/Reservoir Activity* dialog box. This will activate the button that leads to the dialog box for that module.

HYDR

To access the HYDR Parameters dialog box, press the **HYDR** button in the *Reach/Reservoir Activity* dialog box. The *HYDR Parameters* dialog box lets the user enter the parameters for the following HYDR input tables: HYDR-PARM1, HYDR-PARM2, and HYDR-INIT. The following monthly value tables may also be input from this dialog if required: MON-CONVF. Other tables used to set up “categories” to be simulated are not supported in WMS. Some parameters require a single value for the entire simulation and others are allowed to vary monthly, depending on the flags set in the HYDR-PARM1 table. For information on the definition and function of specific variables in this module, see Section 4.4(3).2 – HYDR Input of the HSPF Manual.[1]

In addition to the fields and buttons to edit HYDR input variables, a few other buttons exist in this dialog. The **Print Info** button leads to a dialog where the PRINT-INFO block for RCHRES segments may be set up (see Section 4.4(3).1.2 of the HSPF Manual). The **Define FTABLE** button leads to a dialog where the FTABLE for the selected reach/reservoir may be input (see Section 4.5 – FTABLES Input of the HSPF Manual). This functionality will also be discussed later.

ADCALC

To access the *ADCALC Parameters* dialog box, press the **ADCALC** button in the *Reach/Reservoir Land Activity* dialog box. The *ADCALC Parameters* dialog box lets the user enter the parameters for the ADCALC input table: ADCALC-DATA. For information on the definition and function of specific variables in this module, see Section 4.4(3).3 – SOLIDS Input of the HSPF Manual.[1]

CONS

To access the *CONS Parameters* dialog box, press the **CONS** button in the *Reach/Reservoir Activity* dialog box. This dialog box allows the user to specify up to 10 water quality constituents or pollutants in the flows from a reach/reservoir segment. The CONS Parameters dialog box lets the user enter the parameters for the following CONS input tables: NCONS, CONS-AD-FLAGS, CONS-DATA. For more information on these tables, see Section 4.4(2).7 – IQUAL input of the HSPF manual.[1]

The key to using this dialog is to note that the NCONS field at the top of the box designates the total number of constituents to be modeled, while the *Currently Editing Constituent* drop-down box indicates the constituent that is available for editing in the fields below.

HTRCH

To access the *HTRCH Parameters* dialog box, press the **HTRCH** button in the *Reach/Reservoir Activity* dialog box. The *HTRCH Parameters* dialog box lets the user enter the parameters for the following HTRCH input tables: HT-BED-FLAGS, HEAT-PARM1, HT-BED-DELH, HT-BED-DELTT, and HEAT-INIT. The following monthly value tables may also be input from this dialog if required: MON-HT-TGRND. Some parameters require a single value for the entire simulation and others are allowed to vary monthly, depending on the flags set in the HT-BED-FLAGS table. For more information on these tables, see Section 4.4(3).5 – HTRCH input of the HSPF manual.[1]

SEDTRN

To access the *SEDTRN Parameters* dialog box, press the **SEDTRN** button in the *Reach/Reservoir Activity* dialog box. The *SEDTRN Parameters* dialog box lets the user enter the parameters for the following SEDTRN input tables: SANDFG, SED-GENPARM, SAND-PM, SILT-CLAY-PM, SSED-INIT, BED-INIT. The SED-HYDPARM table may also be active if the HYDR module is inactive. The combination of required tables will vary, depending on the flags set in the SAND-FG table. The required tables will be listed in the drop-down list to indicate input that must be completed. Simply choose a table from the list and the input fields for that table will appear. For information on the definition and function of specific variables in this module, see Section 4.4(3).6 – SEDTRN Input of the HSPF Manual.[1]

RQUAL

This module is used to simulate the behavior of constituents involved in biochemical transformations. Unlike the other modules, this has multiple sections that are each as involved as any other module. Those sections are OXRX, NUTRX, PLANK and PHCARB, which model primary dissolved oxygen and biochemical oxygen demand balances, inorganic nitrogen and phosphorus balances, plankton populations and associated reactions, and pH and inorganic carbon species, respectively. The constituents that may be modeled using RQUAL are dissolved oxygen, biochemical oxygen demand, ammonia, nitrite, nitrate, orthophosphorus, phytoplankton, benthic algae, zooplankton, dead refractory organic nitrogen, dead refractory organic phosphorus, dead refractory organic carbon, pH and carbon dioxide.

To access the *RQUAL Parameters* dialog box, press the **RQUAL** button in the *Reach/Reservoir Activity* dialog box. The *RQUAL Parameters* dialog box lets the user enter the parameters for the following RQUAL input tables: BENTH-FLAG and SCOUR-PARMS. The four checkboxes and buttons located in the *RQUAL* dialog box lead to the four sections of the RQUAL module: OXRX, NUTRX, PLANK and PHCARB. Input for those sections will be discussed in detail below. For information on the definition and function of specific variables in this module, see Section 4.4(3).8 – RQUAL Input of the HSPF Manual.[1]

OXRX

This section is used to simulate the primary processes that determine the dissolved oxygen concentration in a reach or mixed reservoir. To access the *OXRX Parameters* dialog box, press the **OXRX** button in the *RQUAL Parameters* dialog box, explained above. The *OXRX Parameters* dialog box lets the user enter the parameters for the following OXRX input tables: OX-FLAGS, OX-GENPARM, ELEV, OX-BENPARM, OX-CFOREA, OX-TSIVOGLOU, OX-LEN-DELTH, OX-TCGINV, OX-REAPARM, and OX-INIT. A table may be edited by selecting it in the Table Type drop-down box; the required fields will then appear in the dialog.

The segment parameters for these tables are listed in the dialog box and the help strings include a range of values that are reasonable. The combination of required tables will vary, depending on the flags set in the OX-FLAGS table. For information on the definition and function of specific variables in this module, see Section 4.4(3).8.1 – OXRX Input of the HSPF Manual.[1]

NUTRX

This section of the RQUAL module simulates the primary processes which determine the balance of inorganic nitrogen and phosphorus in natural waters. This section is used to simulate the primary processes that determine the dissolved oxygen concentration in a reach or mixed reservoir. To access the *NUTRX Parameters* dialog box, press the **NUTRX** button in the *RQUAL Parameters* dialog box. The *NUTRX Parameters* dialog box lets the user enter the parameters for the following NUTRX input tables: NUT-FLAGS, NUT-AD-FLAGS, CONV-VALI, NUT-BENPARM, NUT-NITDENIT, NUT-NH3VOLAT, MON-PHVAL, NUT-BENCONC, NUT-ADSPARM, NUT-ADSINIT, NUT-DINIT. A table may be edited by selecting it in the *Table Type* drop-down box; the required fields will then appear in the dialog.

The segment parameters for these tables are listed in the dialog box and the help strings include a range of values that are reasonable. The combination of required tables will vary, depending on the flags set in the NUT-FLAGS table. For information on the definition and function of specific variables in this module, see Section 4.4(3).8.2 – NUTRX Input of the HSPF Manual.[1]

PLANK

This section of the RQUAL module simulates phytoplankton, zooplankton and/or benthic algae. To access the *PLANK Parameters* dialog box, press the **PLANK** button in the *RQUAL Parameters* dialog box. The *PLANK Parameters* dialog box lets the user enter the parameters for the following PLANK input tables: PLNK-FLAGS, PLK-AD-FLAGS, SUF-EXPOSED, PLNK-PARM1, PLNK-PARM2, PLNK-PARM3, PHYTO-PARM, ZOO-PARM1, ZOO-PARM2, BENAL-PARM, PLNK-INIT. A table may be edited by selecting it in the *Table Type* drop-down box; the required fields will then appear in the dialog.

The segment parameters for these tables are listed in the dialog box and the help strings include a range of values that are reasonable. The combination of required tables will vary, depending on the flags set in the PLNK-FLAGS table. For information on the definition and function of specific variables in this module, see Section 4.4(3).8.3 – PLANK Input of the HSPF Manual.[1]

PHCARB

This section of the RQUAL module simulates pH, carbon dioxide, total inorganic carbon and alkalinity in the RCHRES. To access the *PHCARB Parameters* dialog box, press the **PHCARB** button in the *RQUAL Parameters* dialog box. The *PHCARB Parameters* dialog box lets the user enter the parameters for the following PHCARB input tables: PH-PARM1, PH-PARM2, and PH-INIT. A table may be edited by selecting it in the Table Type drop-down box; the required fields will then appear in the dialog. For information on the definition and function of specific variables in this module, see Section 4.4(3).8.4 – PHCARB Input of the HSPF Manual.[1]

Related Topics

- HSPF Edit Parameters

HSPF Reading Existing Files

While WMS is capable of reading HSPF User's Control Input (*.uci) files created manually or in other programs, some limitations exist. Some of these limitations are permanent in WMS, other are currently under development and will be resolved.

- WMS will only allow input from one WDM file and output to one OUT file. If more WDM or OUT files are included in the FILES block of the *.uci file, they will be ignored.
- WMS currently reads the SCHEMATIC and MASS LINK blocks to determine the structure and connectivity of the watershed. The NETWORK block is not yet supported.
- If a particular land segment contributes to more than one reach/reservoir segment, that land segment will be copied to a new segment for each reach/reservoir. This can result in many new segments if the *.uci file contains several of these cases. Each new segment created will have the exact same attributes as the original, and will represent the area of the original segment which contributes to each reach/reservoir.
- WMS currently does not read the COPY or GENER block from the *.uci file.

Many of these limitations will be eliminated as development of the HSPF interface continues.

Related Topics

- Overview of HSPF
- Reading and Writing HSPF Files

HSPF Run Simulation

Once the data for HSPF input has been defined, the HSPF model may be launched from WMS. The **Run Simulation** item in the **HSPF** menu will invoke the Run HSPF dialog.

The Run HSPF dialog will allow the user to specify the .uci file, which is necessary to run HSPF. All other files required to run HSPF and to view the output were defined in the HSPF Global Options dialog box.

Once the input file has been defined and you select OK, WinHSPFLt will be executed. A separate window will appear and information about the HSPF simulation will be reported.

If HSPF is not executed successfully when issuing this command, be sure that the path to WinHSPFLt is correct. If HSPF does not run to a successful completion you can view the ASCII output file using the **View File** command in the **File** menu to find the error messages.

As the last section of the HSPF tutorial says, if you have viewed or edited the WDM file data, you need to close WMS after saving your HSPF .uci file before running the file in WinHSPF or WinHSPFLt because WMS keeps the .wdm file open. HSPF cannot run the model unless the .wdm is closed in all other programs.

Related Topics:

- Overview of HSPF
- Reading and Writing HSPF Files
- Reading Existing Files

HSPF Saving and Reading HSPF UCI Files

While working on the definition of HSPF parameter, or when finished, you will want to save the data in an HSPF input (*.uci) file. The **Save HSPF UCI File** command in the *HSPF* menu saves all HSPF input data to an HSPF input file (*.uci). The format of this file is consistent with that defined in the HSPF users manual. This file may be named anything the user chooses, but generally should have the uci extension. Note that the HSPF input file does not save any geographic data; though the geographic data is not necessarily needed, it may be saved in a WMS superfile using the **Save As** command in the *File* menu.

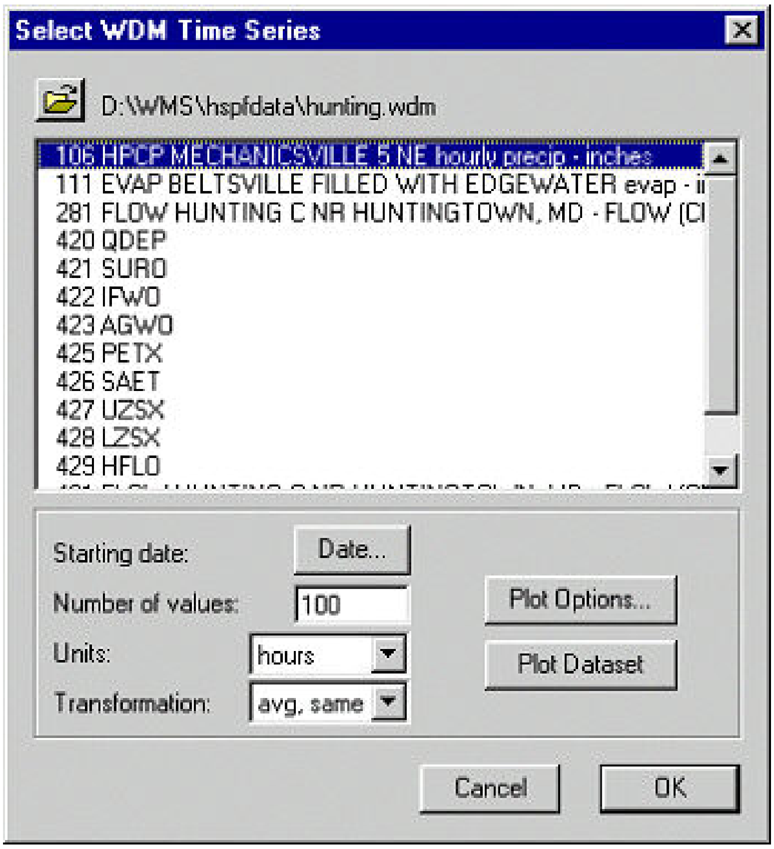
Once an HSPF file has been saved, it may be read in later using the **Open HSPF UCI File** command. Opening this file will build a topological tree and assign the HSPF parameters to the tree. Then the parameters may be edited as described above.

Related Topics

- Overview of HSPF
- Reading Existing Files
- Running HSPF

HSPF Viewing WDM Files

The output data from HSPF are stored in a WDM file. AS in that file are the time series used for input to HSPF. In order to view the se datasets, open the *WDM Datasets* dialog box by going to the **WDM Datasets** command in the *HSPF* menu. The user should make sure the proper WDM file is listed at the top of the dialog, if not, the correct file may be chosen using the file browser. Once a file is chosen, the datasets present in the file will be listed by dataset number and name in the main window of this dialog.



A dataset may be retrieved and plotted by doing the following:

1. Select the dataset number/name from the main window.
2. Note the starting date of the dataset; data will be retrieved from this time forward.
3. Enter the number of values to be retrieved from the dataset.
4. Select the time units of the data.
5. Select the transformation code used when retrieving the data.
6. Click on the **Plot Dataset** button.

These steps will open the *Hydrograph Window* of WMS and plot the specified data as a linear plot. Some of the plot format may be changed by clicking on the **Plot Options** button and setting the options as desired.

Related Topics

- WDM File Interaction
- External Sources
- External Targets

HSPF WDM File Interaction

HSPF requires several time series datasets as input for hydrologic modeling. It also reports results of simulation as time series datasets. These data are stored in a watershed data management (WDM) file. These files are binary, direct-access data files which can hold several thousand datasets. Thus, management of the interaction with these WDM files is a large part of HSPF modeling. Datasets must be selected from the WDM file as input to a simulation and datasets must be specified as targets for simulation results. The EXT SOURCES and EXT TARGETS blocks of the HSPF input file control these interactions. Tools have been developed in WMS to aid in reviewing WDM files and setting up these input blocks for HSPF.

Related Topics

- Overview of HSPF
- External Sources
- External Targets
- Viewing WDM Files

6.7. MODified RATional Method (MODRAT)

MODRAT

Introduction

MODRAT is a modified rational method computer program developed by the Los Angeles County Department of Public Works (LACDPW) to compute runoff rates under a variety of conditions common to the area of Los Angeles, California. The successor to F0601 and/or MORA, MODRAT contains all the features of the original F0601 as well as updated capabilities for watershed modeling in the Los Angeles area. MODRAT may be used to find flow rates for any watershed with any combination of existing or proposed channels and drains. Further, the watershed may be undeveloped, partially developed, or completely developed. The model will compute runoff rates for any frequency design storm (storm patterns developed by LACDPW), as well as any other storm which can be represented by a rainfall mass curve. Given any combination of the above variables, MODRAT will compute hydrographs for each subarea and mainline or lateral collection point in the watershed.

The MODRAT model has been updated and maintained by the WMS development team under the direction of the LACDPW. The current MODRAT model is the result of several updates in methodology approved by the LACDPW. Current implementation of the MODRAT model interface and parameter computations done by WMS are approved by the LACDPW for hydrologic studies.

Modifications to the Rational Method in MODRAT

As a method of urban hydrology, the rational method falls short in several ways. First, the method does not produce a hydrograph, only a single flow rate. Second, the rational method does not account for changing (time dependent) conditions such as soil condition or rainfall intensity. Finally, results are not very accurate for large areas. Due to these problems, MODRAT contains the following modifications:

- Rainfall intensity, i, is a variable dependent on rainfall frequency, storm time, and time of concentration. The variation of i is represented by a temporal distribution curve (rainfall mass curve).
- C, the runoff coefficient, varies with soil type, rainfall intensity, and imperviousness.
- The time variation of C and i allow the flow, Q, to vary with time, thus producing a hydrograph. The area under the hydrograph represents the total volume of flow from a watershed, a variable which the rational method does not provide.
- Hydrographs may be computed for a number of subareas, for each lateral to the main channel, and for each collection point on the main channel. These hydrographs are routed and combined as computation progresses downstream.

The above modifications to the rational method allowed for the computation of storm hydrographs for any size watershed. With such improvements, the modified rational method (MODRAT) has been adopted by LACDPW as the preferred method of hydrologic analysis.

Related Topics

- MODRAT WMS Interface
- Editing MODRAT Parameters
- Saving and Reading MODRAT Files
- Running MODRAT

MODRAT WMS Interface

The objective of the interface developed in WMS for the MODRAT model is to provide graphical representation of MODRAT data, as well as automate the definition of many of the required parameters. Many of the basic input parameters are computed as part of WMS' basic functionality. Additionally, specialty functions have been added to WMS to enable the usage of MODRAT directly with WMS. Several custom dialogs and menu options have been developed in WMS to facilitate the definition of MODRAT input data. Furthermore, MODRAT can be launched directly from WMS through menu options. These menus can be found in the Hydrologic Modeling Module under the MODRAT heading.

Related Topics

- Overview of MODRAT
- Editing MODRAT Parameters

MODRAT Conveyance Type

Each reach in an MODRAT model must be assigned to be one seven conveyance types. Certain parameters must also be defined to describe the geometry of the conveyance. Boxes in which to enter these parameters will appear when a conveyance type is selected. The conveyance types are:

- Variable – if this type is selected, MODRAT will begin flood routing in a street section, change from street to pipe when the flow depth reaches the curb height, from pipe to rectangular channel when a pipe diameter of 8 feet is exceeded. Routing will continue in the rectangular channel until a maximum depth of 13 feet is reached.
- Mountain channel – no additional parameters need to be defined for this conveyance type.
- Natural valley channel – no additional parameters need to be defined for this conveyance type.
- Street channel – the street width (feet) and the curb height (inches) must be selected when using this option.
- Circular pipe – the diameter of the pipe (feet) of proposed or existing drains may be entered; otherwise, leave this at zero and MODRAT will compute the needed diameter of this conveyance type.
- Rectangular channel – the roughness of the channel sidewalls and either the base width (feet) –OR– the depth of the channel (feet) are needed for this conveyance type.
- Trapezoidal channel – channel side slope, maximum peak velocity (ft/s), side wall roughness, and either the base width –OR– depth of the channel are needed for MODRAT routing computations.

Related Topics

- Editing MODRAT Parameters
- Length
- Slope
- Manning's N
- Relief Drains
- Reservoirs

MODRAT Course Overview

Overview of the MODRAT Training Course in Los Angeles, California

A screen cast training presentation of MODRAT can be found in the following links.

Segment 1: MODRAT Course Overview
MODRAT Course Overview [1]
Segment 2: General WMS Demonstration
DEM Delineation Demo [2]
Segment 3: Basic MODRAT Modeling
Basic MODRAT Modeling [3]

References

[1] http://wms.aquaveo.com/MODRATCourseOverview.htm
[2] http://wms.aquaveo.com/DEMDelineation.htm
[3] http://wms.aquaveo.com/modrat.htm

MODRAT Creating Burned Simulations

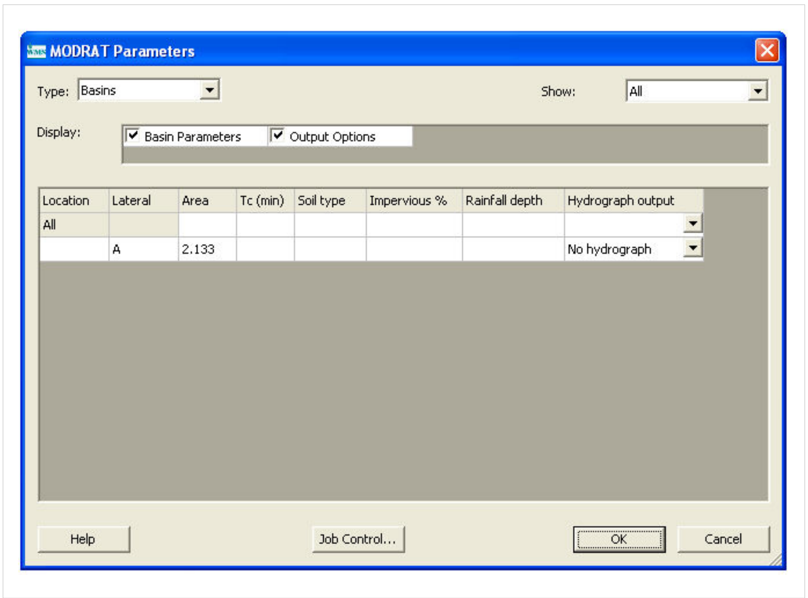
Burned soil simulations can be created using existing normal soil simulations. For the burned soils increment enter the difference between the existing soil numbers and burned soil numbers in the soil file (lasoilx.dat). WMS increments the soil number so that it will be a burned soil for each sub-basin with a percent impervious value less than the percent impervious limit specified in this dialog. Save the *.lac file to use the burned soil numbers in your simulation. Be sure to save the *.lac file with a different name before creating a burned simulation if you would like to keep the normal soil simulation.

Related Topics

- Editing MODRAT Parameters
- Running MODRAT
- Saving and Reading MODRAT Files

MODRAT Edit Parameters

Edit MODRAT Parameters Dialog



Attributes or parameters for all MODRAT hydrograph stations are defined and/or edited using the *Edit MODRAT Parameters* dialog, shown below. This dialog is accessed by selecting the **Edit MODRAT Parameters** command from the *MODRAT* menu or by double-clicking on a basin, reach, or relief drain when MODRAT is the active model in WMS.

If a subarea, reach, or relief drain is selected before issuing the **Edit MODRAT Parameters** command, then the data for that object is loaded for editing. When a hydrograph station is selected (subarea/reach/relief drain), only the items which correspond to the selected hydrograph station are active, all others are inactive. For example, when a subarea is selected, all items under the Basin Data heading are active and all items under the Reach Data heading are inactive.

Once the *Edit MODRAT Parameters* dialog appears, it will remain open until the user selects the **Done** button. This enables the user to switch between basins, reaches, or relief drains quickly and define the necessary parameters without closing and opening the dialog for each hydrograph unit.

Location

The number entered in the *Location* field of the *MODRAT Parameters* dialog indicates the place in computational order the subarea or reach occupies. Computations in MODRAT begin at location 1 and continue sequentially. The location assignment is generally done automatically in WMS; for more information on numbering your watershed model, see the MODRAT Numbering the Tree section.

Lateral

The letter entered in the Lateral field of the *MODRAT Parameters* dialog indicates the line to which the subarea or reach is connected. The main line in any MODRAT model is line A; laterals may be labeled B through Z. Reaches with two letters indicate a confluence point of two or more reaches. The lateral assignment is generally done automatically in WMS; for more information on lettering your watershed model, see the MODRAT Numbering the Tree section.

Area

Enter the tributary area, in acres, of the subarea. If the WMS graphical modeling tools have been used to define the watershed from digital data, the area of each subarea will be automatically computed and will appear in this text field. If not, you must manually enter the area of each subarea.

Time of Concentration/Tc (min)

Enter the time of concentration of each subarea, in minutes. Normally, this number is between 5 and 30 minutes. LACDPW has developed several regression equations to compute the time of concentration of a subarea, which are embedded in WMS. The Compute Regression Tc button will take you to the Time Computation dialog where you can compute Tc using the LACDPW regression equations.

Soil Type

Determine the soil type most representative of each subarea, between 2-172 for normal soil conditions and 202-372 for burned soil conditions. Soil type maps from LACDPW are used to find which soil is predominant in the subarea. This procedure has been automated in WMS; a digital soil map may be imported into WMS and overlaid on the watershed to be modeled. WMS will then calculate which soil covers the majority of the area of a subarea. This function is part of “mapping MODRAT attributes”; a full description can be found in the Map MODRAT Attributes section. If the watershed being modeled has not been graphically defined in WMS, enter the appropriate soil number based on visual inspection.

Impervious Area

Define the percent of the subarea that is impervious to infiltration. The percent impervious generally depends on the land use in the basin; as with soil type, land use maps have been developed to allow definition of this parameter through graphical analysis. WMS can determine the percent impervious by mapping MODRAT attributes, or the user may enter this number independently.

Rainfall Depth

Enter the depth of rainfall, in inches, on the subarea. The depth of rainfall on any subarea in the Los Angeles area can be determined from isohyetal maps developed by LACDPW (maps for the 2, 5, 10, 25, 50, 100, and 500-year are available). These isohyetal maps are available in ARC/INFO® Grid format for import into WMS. WMS can determine the rainfall depth in any subarea from these rainfall grids through the Map Attributes command in the

MODRAT menu.

Along with defining a rainfall depth, a temporal distribution series must also be defined. The **Define Series** button allows you to enter the XY Series Editor in WMS. This editor allows you to import or manually enter a temporal distribution for the rainfall in the subarea. The temporal series typically used for LA County can be imported from the file LACDPWStorm1500min.xys, found in the /MODRAT/Rain/ folder of your WMS installation.

Related Topics

- Reach Data
- Mapping Parameters from GIS Data Files
- Job Control Parameters
- Renumbering the Tree
- I/O
- Computing Tc

MODRAT Input/Output Hydrographs

Input Hydrographs

An inflow hydrograph may be specified at any location (subarea or reach) of an MODRAT watershed model. To indicate where the input hydrograph enters the model, simply select the location and click on the check box entitled *Input hydrograph*. This will direct MODRAT to import a hydrograph from a file at this point.

This file is typically a result from a previous MODRAT run, but could be manually created provided that the correct format is used. One option for creating the file is to toggle the box next to the **Define Input Hydrograph...** button on and click the **Define Input Hydrograph...** button to enter input hydrograph data at the selected location. WMS will write the file in the correct format based on the options specified in the Job Control.

The MODRAT model may have several input hydrographs for different locations; however, the hydrographs in the input file must be ordered from upstream to downstream for MODRAT to access the data correctly. The format of the hydrograph data in this file is standard MODRAT hydrograph file format.

Output Hydrographs

Three options are available for hydrograph output at each location in an MODRAT watershed. These options are:

- No hydrograph – This option instructs MODRAT to not write out any hydrograph data for the given location.
- WMS plot file (*.sol) – This option instructs MODRAT to write a hydrograph plot file which can be loaded into WMS and viewed as a flow vs. time plot in the *Hydrograph* Window. It also instructs MODRAT to write the hydrograph to the summary file (named *.sum) in standard MODRAT hydrograph format.
- Hydrograph (*.hyf) file and WMS plot file (*.sol) – This option writes the hydrograph to a MODRAT hydrograph file (named *.hyf), to a summary file (*.sum), and to a WMS plot file (*.sol).

Printing Confluence Details

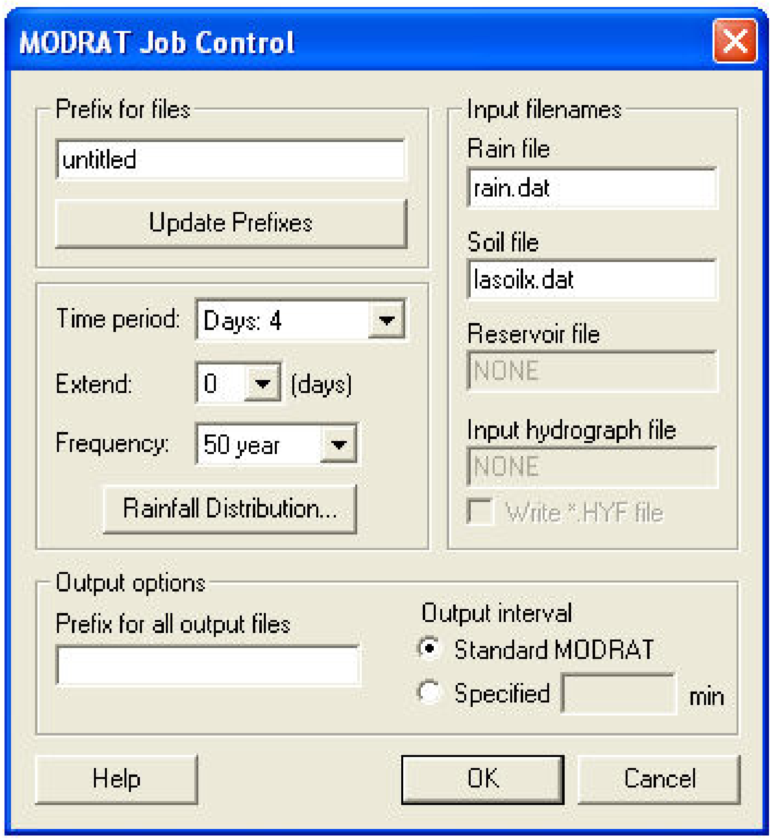
This option is available whenever a confluence point is selected. Checking this box will instruct MODRAT to give a detailed print out (in the standard MODRAT output file) of the hydrograph at this point.

Related Topics

- Editing MODRAT Parameters
- Job Control Parameters
- Running MODRAT
- Saving and Reading MODRAT Files

MODRAT Job Control Parameters

The *Job Control* dialog of MODRAT is used to set several options which affect the duration MODRAT will model and the data it will use for input. This dialog is pictured below.



Prefix for Files

Enter the prefix you want for MODRAT files (input and output), then click the **Update Prefixes** button to assign this prefix to the Reservoir file, Input hydrograph file, and output files listed in the *Job Control* dialog.

Input File Names

Rain File

Enter the name of the file that will contain the rainfall mass curves to be used by MODRAT. This file will be created by WMS—you can name it anything you think appropriate.

Soil File

The file containing soil type information for MODRAT should be entered in this field. This is generally the standard file created by the LACDPW named LASOILX.DAT.

A user-defined soil type file may be created and the name entered in this field if any soil number not included in the standard file of 2-172 or 202-372 is used. This file must be sequentially numbered and be in the standard MODRAT soil file format. The easiest way to create this file is to copy the standard L.A. County Soil file (LASOILX.DAT), then add additional soil numbers and data to the end of it, beginning with number 373.

Reservoir File

Enter the name of the file that will contain the reservoir storage-capacity and elevation-discharge curves to be used by MODRAT. If you have defined the reservoir with the *Detention Basin Calculator* in WMS, this file will be created by WMS – you can name it anything you think appropriate. If you have manually created this file in the format required by MODRAT, you can enter the name of the file here.

Input Hydrograph File

If an input hydrograph has been specified at any location in your watershed, the name of the file containing the appropriate data must be entered here. The file must contain all input hydrographs for the watershed if more than one is specified; the order of the hydrographs in the file must be from upstream to downstream. In other words, if an input hydrograph is specified at 1A and 5B, the data for the hydrograph at 1A must come first in the input file, followed by the data for 5B. The format of the hydrograph data file is identical to the hydrograph output file created by MODRAT (the *.hyf file).

Toggle the *Write *.HYF* check box on to have WMS write the *.hyf file using input hydrograph data that you entered in the Edit MODRAT Parameters dialog.

Rainfall Distribution

Time Period

MODRAT can be run to calculate runoff for 4 different time periods:

- Days 1, 2, 3, 4 of a four day storm
- Days 2, 3, 4 of a four day storm
- Days 3, 4 of a four day storm
- Day 4 of a four day storm

Frequency

The frequency chosen in the *Job Control* dialog does not affect the MODRAT computations. It is, however, included for reference in output from MODRAT.

Output Options

Prefix for All Output Files

Enter the prefix you want for all output files MODRAT will create when it runs (*.out, *.sum, *.out, etc.). This may be any name you deem appropriate.

Reservoir Output Interval

Choose whether to use the standard MODRAT interval or specify a time interval (min) for MODRAT to output reservoir routing results.

Related Topics

- Editing MODRAT Parameters
- I/O
- Reservoirs
- Saving and Reading MODRAT Files
- Editing MODRAT Parameters

MODRAT Length

The length, in feet, of each reach in a watershed is needed to perform routing computations in MODRAT. When the watershed has been graphically delineated in WMS, the length of each reach is automatically calculated and assigned. Otherwise, you must calculate and enter the length of each reach manually.

Related Topics

- Editing MODRAT Parameters
- Slope
- Conveyance Type
- Manning's N
- Relief Drains
- Reservoirs

MODRAT Manning's Roughness Coefficient, n

Enter the Manning’s roughness coefficient, n. The default value for MODRAT is 0.014. Any other roughness coefficient value deemed appropriate may be entered.

Related Topics

- Editing MODRAT Parameters
- Length
- Slope
- Conveyance Type
- Relief Drains
- Reservoirs

MODRAT Numbering the Tree

For MODRAT to properly compute the flow rates in a watershed, basins and reaches must be numbered and lettered properly, from most upstream to downstream. WMS will automatically number and letter the watershed components once a topological tree has been created. Choosing **Number Tree** from the *MODRAT* menu will initiate the process to number and letter the watershed model.

The numbering/lettering will start at the most upstream point of the watershed and continue automatically until a confluence point with sub-basins attached is encountered. At this point, WMS cannot determine which lateral the sub-basin drains into. Thus, you will be prompted to select the sub-basins corresponding to each lateral that confluences at the point in question. The confluence point in question will be highlighted and WMS zooms to a window which encompasses the confluence point and all drainage basins and outlet points attached to the confluence point. You must select the basins corresponding to each lateral as you are prompted to do so in the dialog. Note that WMS will automatically skip numbers/letters at certain confluences to allow for multiple line confluence nodes.

Helpful tips when numbering:

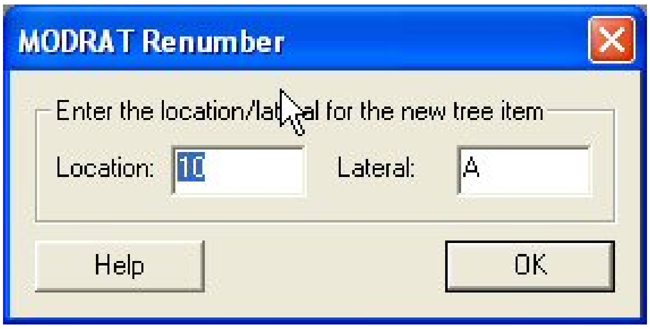
- WMS will select the upstream end of the longest branch and start the numbering there by default. However, you can select the upstream basin of any branch, then select the **Number Tree** command to designate the main line (A).
- Use the mouse wheel to zoom in/out on the selected confluence point
- You do not have to select a sub-basin for all laterals, just select **Done** without selecting any basins

If the renumbering results are not what you want, you can choose the **Number Tree** command again and try something different. If it is not possible to achieve the numbering/lettering you want with the automatic process, you will have to edit the Location and Lateral of each watershed component in the Edit MODRAT Parameters dialog.

Inserting Tree Spacing

WMS will update all location numbers by multiplying them by the scaling factor in order to create node spacing. This is useful for creating space in the model for inserting tree nodes in the future. Automatic tree renumbering may be a more effective method of obtaining similar results.

Renumbering the Tree



Adding Tree Items

After the tree has been initially numbered, WMS displays the default location and lateral every time you add a new basin, reach, or diversion. Change the location and lateral if the default values are not correct and select **OK** to add the tree item and automatically renumber the tree. If you add a basin at a confluence point the default location number will be updated based on the lateral that you enter.

Deleting Tree Items

The tree is renumbered when a basin or diversion is deleted. If a reach is deleted you will need to number the entire tree again if you want to update the location numbers and laterals for each tree node.

Renumbering automatically occurs if you edit the *Location* or *Lateral* in the *Edit MODRAT Parameters* dialog.

Related Topics

- Editing MODRAT Parameters

MODRAT Relief Drains

When the reach where a relief drain exits the line is selected, the items under the *Relief Drain* heading will be active. The first item defines how the hydrograph at that point will be split between the main line and the relief drain. Four options are available:

- Computed percentages – This option requires the user to input the peak flow, in cfs, to remain in the main drain (usually the peak capacity of the drain). The hydrograph is split based on a percentage computed as the ratio of specified flow rate to the hydrograph peak flow rate. The hydrograph remaining in the drain has peak flow entered by the user, with residual flow transferred to the relief drain.
- Specified percentage – This option requires that the user input the percentage of total flow to remain in the main drain. The hydrograph will be split strictly on a percentage basis, with the specified percentage remaining in the drain and the remainder transferred to the relief drain.
- Specified Drain Capacity – This option requires that the user specify the existing drain capacity (cfs). The hydrograph will be separated such that all flow up to the peak flow will remain in the drain, with excess flow transferred the relief drain.
- Specified Relief Capacity – This option requires that the user specify the drain capacity, in cfs. The hydrograph is split such that flow up to the specified capacity goes to the relief drain, with all flow above the drain remaining in the existing drain.

Related Topics

- Editing MODRAT Parameters
- Length
- Slope
- Conveyance Type
- Manning's N
- Reservoirs

MODRAT Reservoirs

Checking the Reservoir routing box instructs MODRAT to perform Modified-Puls Level Pool routing (reservoir routing) at the confluence point selected. This routing operations occurs prior to the channel routing specified. This requires input of a storage-capacity curve and an elevation-discharge curve for the reservoir to be modeled. When the box is checked, the **Define Reservoir** button will become active. Clicking the button will open the Detention Basin Calculator; this calculator provides several options for defining the curves needed for routing.

Multiple elevation discharge relationships can be defined in the *Detention Basin* calculator. WMS will use each elevation discharge relationship to interpolate discharges for all elevations defined in the storage capacity relationship and use those to compute a composite elevation discharge curve for routing.

Reservoir routing will generate a reservoir routing output file from the MODRAT model. A full report of input curves and the inflow/outflow/storage table is included in this file.

Related Topics

- Editing MODRAT Parameters
- Length
- Slope
- Conveyance Type
- Manning's N
- Relief Drains
- Detention Basin Calculator

MODRAT Run Simulation

Once the data for MODRAT input have been defined, the MODRAT model may be launched directly from WMS.

The *Run MODRAT* dialog provides two options for running a MODRAT model:

1. Create a new MODRAT input file from the data in WMS and run it. To do this, make sure the *Save file before run* checkbox is checked, then click the **Input File** button, browse to the folder where you want to save the MODRAT input and output files and enter a new filename for the MODRAT input file. Note that the option to specify the prefix for output files from MODRAT is active with this option. This provides a reminder of the name specified in the *Job Control* dialog, and allows you to change the name if desired. WMS will save a *.lac file (standard MODRAT input), and *.dat for rainfall input, and reservoir or input hydrograph files if necessary.
2. Run MODRAT with an existing input file. To do this, make sure the *Save file before run* is unchecked, then click the **Input File** button and choose the file you wish to input to MODRAT. Click **OK** in the *Run MODRAT* dialog to launch the model run. WMS will not save any files to disk—it will just launch the model. The MODRAT model will run and write the standard output files.

Once an input file has been specified and you select OK, MODRAT will be executed. A separate window will appear and information about the MODRAT simulation will be reported. If MODRAT is not executed successfully when issuing this command, be sure that the path to the MODRAT.EXE file is correct. If MODRAT does not run to a successful completion you can view the ASCII output file using the **View File** command in the *File* menu.

Related Topics

- Overview of MODRAT
- Saving and Reading MODRAT Files

MODRAT Slope

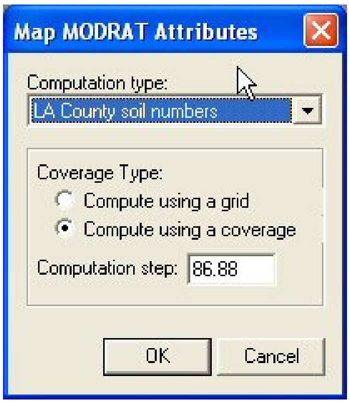
The slope of each reach is needed to perform routing in MODRAT. If the watershed to be modeled has been graphically delineated in WMS, this value will be calculated and assigned to each reach. If not, enter this value manually.

Related Topics

- Editing MODRAT Parameters
- Length
- Conveyance Type
- Manning's N
- Relief Drains
- Reservoirs

Using GIS Data to Map Parameters

Part of the MODRAT interface in WMS is an automated mapping procedure to define rainfall depth, soil type, percent impervious, and DPA zones from digital data layers. The procedure requires a digital map of the parameter, such as rainfall depth, to be imported into WMS. Digital maps of rainfall depth (isohyetal maps), soil types, land use (percent impervious), and DPA zones are all available from the LACDPW. Once the map is in WMS, functions have been created which will overlay the parameter map on your watershed model and assign the appropriate value to each sub-basin. These functions can be accessed by choosing **Map Attributes** from the *MODRAT* menu; the following dialog controls the mapping procedure:



Computation Type

Three parameters may be mapped from this dialog:

LA County soil numbers – this option will determine the predominate soil type in each sub-basin of the watershed.

To compute the soil number for each sub-basin of your MODRAT model, you must read in a soil type data layer. Soil type data is generally stored in a GIS, the best way to read it into WMS is though a Shapefile. To import soil type data for use with MODRAT, create a Soil Type Coverage, then import the polygon file and ensure that the *Soil Number* field (often called CLASS_NO by the LACDPW) is assigned to LA County Soil type field of WMS. Once the soil type data is present in WMS, you can compute and assign soil numbers using the **Map Attributes** command from the *MODRAT* menu.

LA County land use – this option will determine the predominate land use type in each sub-basin. The percent impervious is then computed from the land use type.

To compute the Percent Impervious for each sub-basin of your MODRAT model, you must read in a land use data layer. Land use data is generally stored in a GIS, the best way to read it into WMS is though a Shapefile. To import land use data for use with MODRAT, create a Land Use Coverage, then import the polygon file and ensure that the *Percent Impervious* field (often called IMPERV_ by the LACDPW) is assigned to LA County Soil land use field of WMS. Once the land use data is present in WMS, you can compute and assign % impervious using the **Map Attributes** command from the *MODRAT* menu.

LA County DPA zones – this option is currently not in use. The computation will determine which DPA zones exist in a sub-basin, and the area corresponding to these zones.

LA County rainfall depth is mapped using the GIS calculator.

To compute the rainfall depth on the sub-basins of your MODRAT model, you must read in a rainfall depth data layer. The isohyetal maps for LA County have been digitized and stored in ARC/INFO® Grid format by the LACDPW; these files can be found in the /MODRAT/Rain/ folder installed with WMS. Import this data by choosing **Open** from the *File* menu, then choose *Rainfall Depth Grid (*.*)* as the file type.

Alternatively, if you have other rainfall data in vector format (ArcView® Shapefiles, DXF, etc.) you can import these into WMS. Make sure that you create a Rainfall Coverage (see the Coverages dialog) and map the appropriate database fields when importing this data.

Once the rainfall depth data is present in WMS, you can compute and assign depths using the **Compute GIS Attributes** command from the *Calculators* menu.

Coverage Type

MODRAT parameters may be mapped from two types of data: gridded (raster) data or coverage (vector) data. These options will be inactivated or activated in accordance with the type of data which exists in WMS.

Computation Step

Mapping attributes in WMS is a grid-based operation; when a parameter is mapped, each basin is broken into an imaginary grid, then each grid cell is assigned the appropriate parameter value. Either an area-weighted average of the parameter value or the predominate value can then be determined.

The computation step value defines the size of each imaginary grid cell. A larger value creates a larger cell size and thus a lower resolution grid. For example, a computation step of 100 will create cells 100 X 100 whereas a step of 50 will create smaller cells (50 X 50). Computation step may need to be adjusted depending on the size of the sub-basins in the watershed. WMS sets the default value in the Computation Step box to a number that will divide the smallest sub-basin in the watershed into a grid 25 cells X 25 cells in resolution. Generally, the default is adequate. If greater accuracy is required, a lower computation step should be used. Note however, a lower computation step will require more computational time.

When gridded attribute data are used (a rainfall depth grid, for example), the computation step will be the same as the grid cell size in the attribute data set and cannot be changed.

Related Topics

- Editing MODRAT Parameters

Saving and Reading MODRAT Files

While working on the definition of MODRAT parameters, or when finished, you will want to save the data. There are 2 options for saving a MODRAT model:

1. Save a WMS project file – this will save any data in WMS (DEMs, soil layers, watershed delineation, etc.) as well as the MODRAT data assigned to each basin, outlet, or diversion of the watershed model. This is the option that is most useful as you are working to build a model and want to return to edit it later. To read this file back into WMS at a later time, just use the **Open** command from the *File* menu.
2. Save only the MODRAT standard input files – the **Save MODRAT File** command in the *MODRAT* menu saves all MODRAT input data to standard MODRAT input files. The format of these file is consistent with that defined in the MODRAT users manual. This file may be named anything you choose, typically with the .lac suffix. The rain input file will be saved with the *.dat suffix. If there are any reservoirs or input hydrographs defined, these input files will be saved as well. The files saved can be reviewed in the MODRAT *Job Control* window. Note that the MODRAT input file does not save any geographic data (maps, DEM, tree item locations, etc.)

Once these MODRAT files have been saved, they may be read in later using the **Read MODRAT** command. Reading this file will build a topological tree and assign the MODRAT parameters to the tree.

Related Topics

- Overview of MODRAT
- Running MODRAT

6.8. National Streamflow Statistics Program (NSS)

NSS

WMS includes an interface to the National Streamflow Statistics Program (NSS). The NSS program is a compilation of all the current statewide and metropolitan area regression equations. The NSS interface in WMS version 8.1 and later uses the same database as the Windows version of the NSS program (released in 2006) which supersedes all previous versions of the NFF program such as the windows version of NFF (released in 2003) and the 1993 derivative used in previous versions of WMS.

You may need to install the latest version of the NSS program from the following web site to get the program to work:

http://water.usgs.gov/software/NSS/

The regression equations are a result of years of effort by the United States Geological Survey (USGS) to develop regional regression equations for estimating flood magnitude and frequency of ungaged watersheds. The USGS, in cooperation with the Federal Highway Administration and the Federal Emergency Management Agency compiled all the regression equations into a single database file. This database file is the basis of the NFF program, which can be used to guide the user through the input required to compute peak flows for different frequencies using the database of state by state regression equations.

The NSS interface in WMS provides a windows based, graphical user interface to the same database of regression equations. The entire program is run from a single dialog. Further, if a digital terrain model is available for the study area, all of the geometric parameters required for the regression equations are automatically supplied as the individual equations are specified. These parameters include area, slope, elevation, distances, and others. The GIS overlay command can be used to compute other variables such as forest cover, lake cover, etc.

The NSS equations are useful for estimating a peak flood discharge and typical flood hydrograph for a given recurrence interval of an unregulated rural or urban watershed. These techniques should be useful to engineers and hydrologists for planning and design purposes.

WMS uses the DLL's that are part of the USGS Windows based version of the NSS program. The USGS program does not include any kind of GIS component, but if you wish to run the NSS outside of WMS you can still use WMS to determine the necessary input. Complete documentation on the USGS NSS program, including specific information for each state can be found on line at the following USGS website:

http://water.usgs.gov/software/NSS/

The descriptions for each state are found at the bottom of the above web page.

Related Topics

- Defining an NSS Simulation
- Saving and Restoring a Simulation
- NSS Model Troubleshooting

NSS Assigning Regression Parameters

As regions are selected in the Regions overlapped by watershed window the relevant parameters for the regional equations are added to the spreadsheet. Parameters that can be computed by WMS are defaulted to their computed values at the time the regression equation is selected for use. You may override defaulted values if you wish, and later return to the computed values by clicking on the **Restore Computed Geometric Values** button.

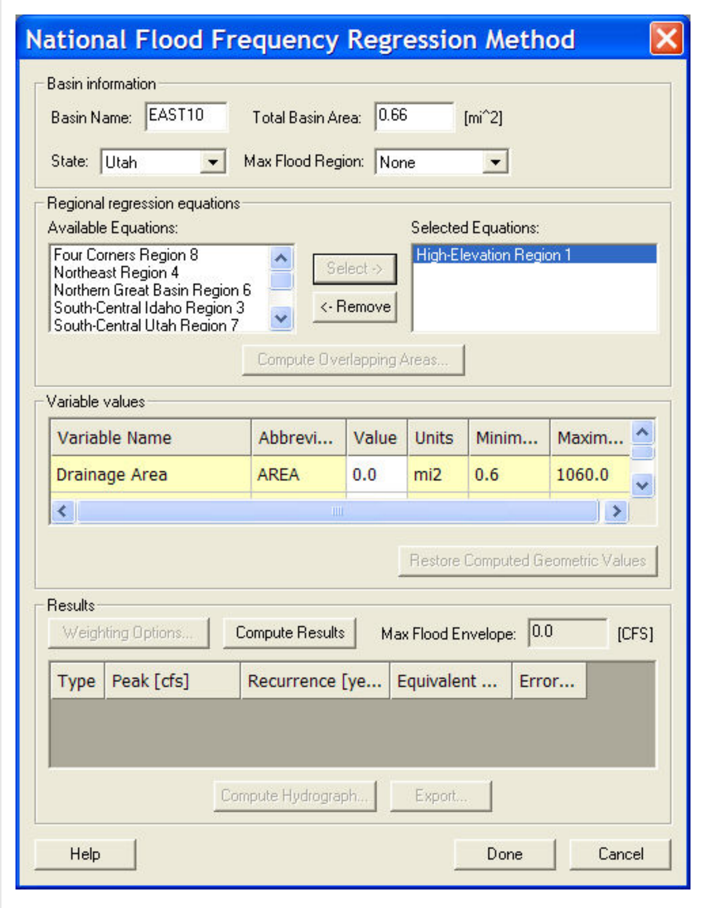
Parameters not computed by WMS when computing basin data can sometimes be computed using the generic *GIS Overlay calculator*. This option (under the **Calculators** menu) can be used for example to determine the percentage of lake or forest cover from your drainage coverage and a land use layer.

The NSS program includes in its database the range of parameter values for the watersheds used in developing the regression equations. The applicable range, along with the expected units are displayed on the same row as the parameter edit field.

Related Topics

- Defining an NSS Simulation
- Overview of NSS
- GIS Overlay

NSS Computing Peak Discharges



Setting up the analysis consists of the following steps, all done within the single dialog shown below:

1. Delineate your watershed and compute your basin area and other geometric parameters. You can enter the value manually if you have not delineated from a DEM, drainage coverage, or TIN.
2. Select the state your watershed is in and assign the Maximum Flood Region.
3. Select the region equation that the watershed is in and while highlighted from the *Available Equations* window choose the **Select** button so that it will be listed in the *Selected Equations* window. If the watershed overlaps more than one region then select the other regions as well.
4. Variable values derived from a DEM or Drainage coverage will automatically be placed in the appropriate edit fields. Other variables not computed will need to be defined.
5. Select the **Compute Results** button to estimate the peak flows for the 2, 5, 10, 25, 50, 100 and 500-year recurrence intervals.

A hydrograph can be estimated by selecting one of the peak flow rows from the Results table and choosing the **Compute Hydrograph** button.

Related Topics

- Overview of NSS
- Saving/Restoring a Simulation
- Hydrographs
- GIS Overlay

NSS Hydrographs

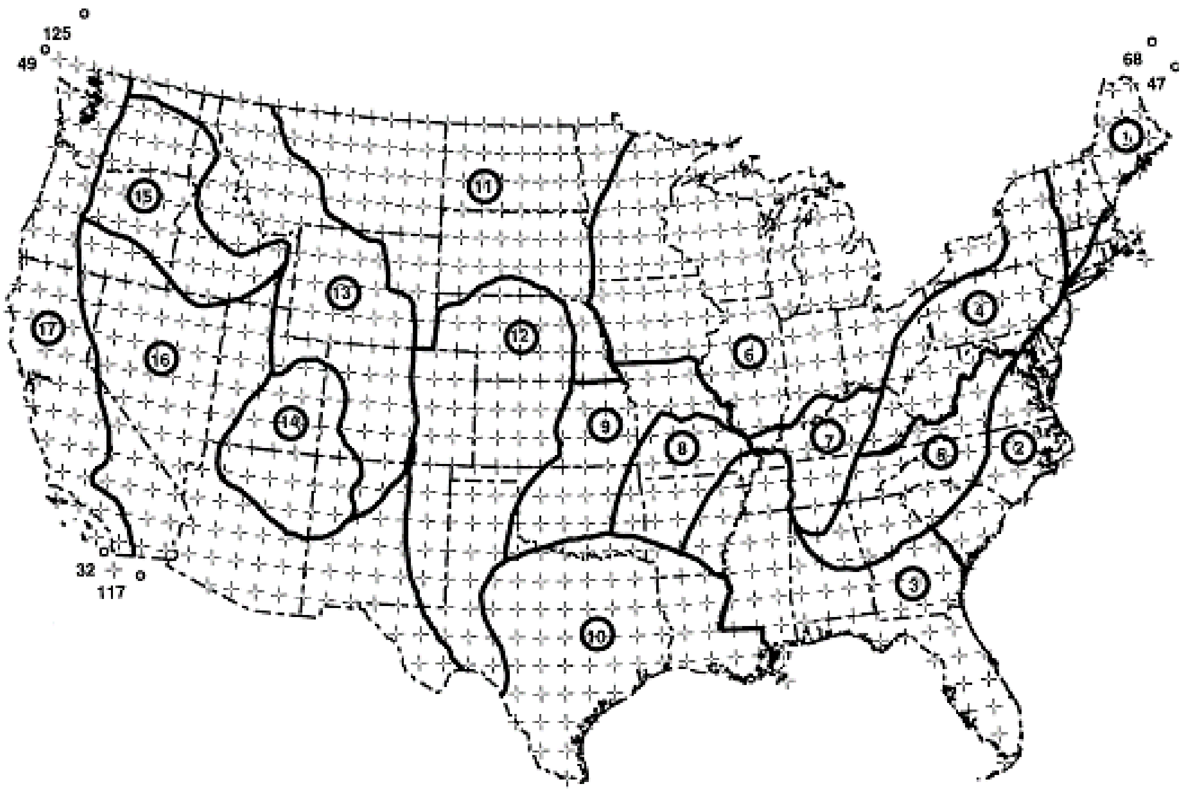
A hydrograph can be estimated using the dimensionless hydrograph derived by the NSS program once peak flows have been computed. When computing a hydrograph a lag time must be defined. The lag time can be computed using either of the available methods in WMS, or by entering it manually.

Related Topics

- Defining an NSS Simulation
- Overview of NSS
- Computing Lag Time

NSS Maximum Flood Region

For comparison and evaluation, the NSS program compares each extrapolated 500-year flood peak discharge with the maximum flood-envelope curves given by Crippen and Bue (1977) and Crippen (1982). The map below shows the different flood regions, and should be used to determine the appropriate region for your watershed. A default value has been determined for each region, but this value may be overwritten.



Related Topics

- Overview of NSS
- Defining an NSS Simulation

NSS Model Troubleshooting

If you cannot get NSS to run from within WMS, try one of the following:

1. You may not have write access to the C:\Program Files directory on your computer. To bypass this, perform the following steps:
   1. Navigate to the C:\Program Files\WMS82 directory
   2. Copy the **wmsnss.exe** file and the **NSSv4.mdb** file from this directory and place them in a directory to which you have write access, such as *My Documents*
   3. Launch WMS and Select Edit | Preferences
   4. Scroll down to the NSS heading and click the **Browse...** button.
   5. Navigate to the wmsnss.exe file in the directory to which you have write privileges and click **Open**

      You should now be able to run NSS.

NSS Overlapping Regions

Each state is divided up into hydrologically similar regions with separate regression equations. Once the state is selected, the available regions appear in the text window below the name.

Selecting Overlapping Regions

You may select the region(s) the study watershed overlaps by clicking on the region name in the Regional regression equations available text window and then **Select →** button. The region named will be "moved" to the *Selected Equations* text window, and any relevant parameters will be added to the values spreadsheet. If a mistake is made the regional equation can be "moved" back again using the **← Remove** button after selecting the *Selected Equations* window.

The USGS Report has a description and applicable maps for each state. These maps can be used to identify which regions are overlapped by your study watershed. You can also download one of these maps, register the image, and create an NSS coverage. With an NSS coverage, and providing consistent coordinate systems are used, the overlapping regions can be computed automatically. The NSS tutorial demonstrates how to do this for a state.

The website for the USGS report and individual state reports/maps can be found at:

http://water.usgs.gov/software/NSS/

Related Topics

- Overview of NSS
- Defining an NSS Simulation
- NSS Region Coverage

NSS State

The NSS regression equations are separated by state. The **State** button is used to specify the state that your watershed is located in. Once the state is specified, the available regions will appear in the regional equations text window. Besides the 50 US states, an equation for Puerto Rico and a custom defined equation can be selected from the available choices.

Related Topics

- Overview of NSS
- Defining an NSS Simulation
- State and National Urban Equations

NSS State and National Urban Equations

The regional equations for each state correspond to analysis for rural watersheds. However each state includes the national urban equation in its list of regions. The national urban equation contains additional parameter values and must be used in conjunction with the appropriate regional rural equation. In addition, some states include regional urban equations which are used separately from the rural equations.

The list of available regression equations changes according to which equations have already been selected. For example if a regional urban equation is selected all other equations disappear, since it must be used by itself. Further, if a regional rural equation is selected any regional urban equations are removed since they cannot be combined.

Related Topics

- Overview of NSS
- Defining an NSS Simulation
- State

NSS Total Basin Area

The total basin area includes the area of all regions overlapped by the watershed. If a terrain model is used to compute areas, this value will be defaulted to the area computed for the selected basin.

Related Topics

Overview of NSS

Defining an NSS Simulation

Saving and Restoring a Simulation

The **Save Simulation** command in the *NSS* menu can be used to save topologic tree structures with any state, regions and parameters which have been defined. The **Read Simulation** command will restore the tree and parameters so that you may continue with a particular model at a later time.

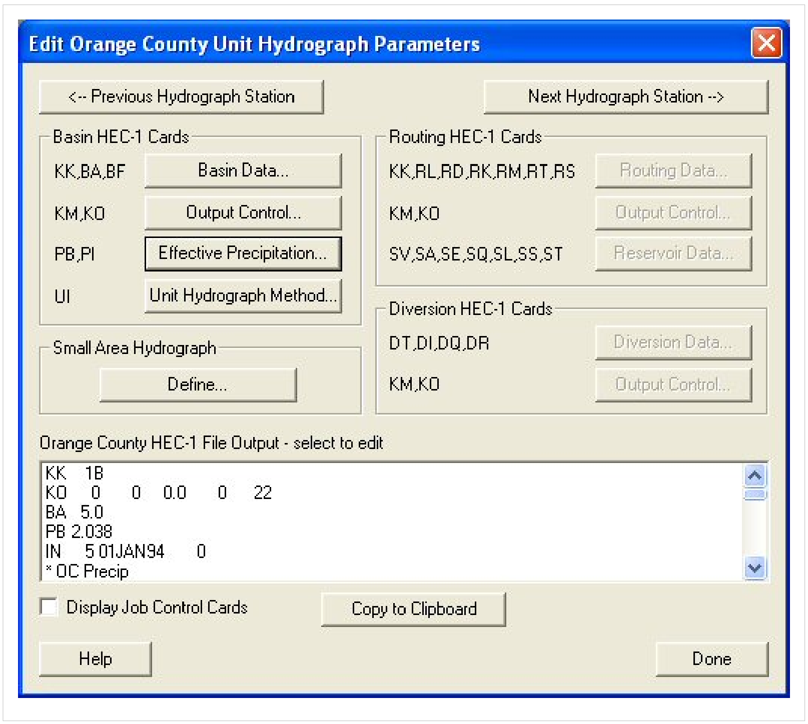
Related Topics

- Overview of NSS

6.9. Orange County Unit Hydrograph

Orange County Unit Hydrograph

Unit hydrograph analysis is used for watersheds greater than 1 square mile (640 acres) in Orange County, California. Losses are accounted for by calculating an effective precipitation. Runoff hydrographs are computed using a unit hydrograph, which is developed using an S-graph. You can use a small area hydrograph instead of a full blown unit hydrograph analysis for watersheds with a Tc < 25 min. WMS creates the input files required for HEC-1 to compute hydrographs for the Orange County unit hydrograph analysis. Much of the interface for the Orange County unit hydrograph analysis is similar to the HEC-1 interface within WMS. Some of the processes specific to Orange County are described below:



- Basin data
- Effective precipitation – losses are included in the precipitation rather than defining a loss method
- Unit Hydrograph Method – unit hydrographs are developed using Orange County S-graphs and lag time derived from the time of concentration computed in an Orange County Rational Method analysis
- Routing
  1. Stream/channel routing – a convex routing method is included in HEC-1
  2. Flow-through detention – define using reservoirs or storage routing in HEC-1
  3. Flow-by detention – define using diversions in HEC-1

Related Topics

- Orage County Hydrology
- Editing HEC-1 Parameters
- Job Control
- Design Storms
- Small Area Hydrographs
- Using GIS Data
- Running a HEC-1 Analysis
- Saving a Simulation
- Post Processing

OC Unit Hydro Design Storms

Define the point precipitation values that WMS will use to compute the design storm by selecting **Define Storm** in the *OC Hydrograph* menu. There are three types of storms: single-day event, multi-day event with flow-through detention, and multi-day event with flow-by detention. Choose a frequency and storm duration (for multi-day events). The antecedent moisture condition (AMC-I, AMC-II, AMC-III) that should be used to compute losses is displayed.

If the design storm event is changed, WMS automatically updates the effective precipitation for all sub-areas in which effective precipitation was previously defined.

Related Topics

- Effective Precipitation

OC Unit Hydro Effective Precipitation

Effective precipitation is the rainfall that actually contributes to runoff. Compute effective precipitation for a basin by first selecting the basin and then choosing the **Effective Precipitation** button from the Edit Orange County Unit Hydrograph Parameters dialog. Effective precipitation is developed by:

1. Weighting point precipitation values for mountainous and non-mountainous terrain – There are different point precipitation values for mountainous (above 2000 ft) and non-mountainous terrain. Enter a decimal value that describes the percentage of mountainous terrain in the sub-area to automatically calculate weighted point precipitation values. Click on the Compute Mountainous button if a DEM or TIN exists to have WMS calculate the amount of mountainous terrain in the sub-area.
2. Applying depth-area reduction factors to get adjusted point precipitation – Point precipitation values are reduced as sub-area size increases. Enter an area to determine the depth-area reduction factors and view the adjusted point precipitation values.
3. Computing unit rainfall according to the time interval specified in the Job Control dialog
4. Subtracting unit losses from the unit rainfall – Two loss rates are used as a basis for computing unit losses in Orange County. There is a low loss rate (Ybar), which is derived from calculating SCS curve numbers, and a maximum loss rate (Fm) that is based on soil type. Click on the **Compute Losses...** button to computes losses using GIS data.

See the Orange County Hydrology Manual for more detailed descriptions of these steps.

The precipitation and losses data that are displayed for multi-day events correspond to the day selected in the combo box. Toggle on the *Edit precipitation parameters* check box to edit the point precipitation, depth-area reduction factors, and adjusted precipitation.

Click the **Next** button to view the effective rainfall in a graphical and tabular format.

Select the **Done** button to write the effective precipitation to the PI card for analysis using HEC-1.

Related Topics

- Design Storms
- Using GIS Data

OC Unit Hydro Job Control

The *Job Control* dialog is used to define general information about the HEC-1 model. Selecting the **Job Control** command from the *HEC-1* or *OC Hydrograph* menu accesses this dialog.

Computational Time Interval

The computational time interval entered here will be used to computed the effective precipitation and unit hydrograph.

Expected Value Analysis

Toggle this checkbox on to do an expected value analysis. Effective precipitation values and losses will be automatically updated for each sub-area if they have already been computed.

Related Topics

- Orange County Unit Hydrograph Parameters

OC Unit Hydro Post Processing

By default WMS will read in the hydrographs computed by HEC-1 and display small icons representing the hydrographs next to their corresponding tree nodes. It is possible to view a larger plot of the hydrograph by selecting a hydrograph icon and choosing the **Open Hydrograph Plot** in the *Display* menu or just by double-clicking the icon. Right-click on the hydrograph plot for options that allow you to format, print, and export the plot and its data.

Use the commands on the *Hydrographs* menu to open or delete HEC-1 results (*.sol file) as well as to view and export the data.

View the HEC-1 output file by using the **Edit File** command in the *File* menu to open the *.out file. This file provides a detailed report of the HEC-1 calculations.

Related Topics

- Displaying Hydrographs

OC Unit Hydro Saving a Simulation

Save only the HEC-1 input files using the **Save Simulation** command in the *OC Rational* menu. This will only save the hydrologic tree and the HEC-1 input parameters at each of the tree nodes. Terrain data, feature data, and GIS data will not be saved unless you save a WMS project file using the **Save** command in the *File* menu.

Related Topics

- Saving HEC-1 Files
- WMS Project Files

OC Unit Hydro Small Area Hydrographs

Small area hydrographs can be used in lieu of developing and applying a unit hydrograph when the time of concentration is less than 25 minutes. A flow rate for each computational time interval is computed using the Orange County rational method equation. Effective precipitation is calculated by subtracting unit losses (the smaller of the two loss values, Fm or Ybar) from the unit rainfall (Orange County IDF curves). The Rational unit hydrograph is applied to each flow rate and the runoff hydrograph results from arranging each of the unit hydrographs according to the design storm pattern used in Orange County, with the peak flow unit hydrograph beginning at hour 16.

- Click the **Update Frequency** button to select the return period and IDF curves
- Computational time interval must be equal to or greater than the time of concentration
- Use the **Computes Losses** button to compute losses using GIS data
- Select **Next** to view a tabular output of the calculations
- Exit all dialogs and select the hydrograph icon to view the runoff hydrograph

Related Topics

- Orange County Rational Method Equation
- Rational Hydrograph

OC Unit Hydro Unit Hydrograph Method

A lag time and S-graph are required to compute a unit hydrograph. Compute a unit hydrograph for a basin by first selecting the basin and then choosing the **Unit Hydrograph Method...** button from the Edit Orange County Unit Hydrograph Parameters dialog.

Lag Time

Basin lag time (hrs) is usually based on the time of concentration (T_c) computed using the rational method analysis. Lag time is automatically calculated to be $0.8 * T_c$when you enter a value for T_c, but you can always enter any value for lag time. Entering a $T_c$value is not required because it is only used to compute lag time.

S-graphs

There are four standard S-graphs that are used in Orange County: valley developed, valley undeveloped, foothill, and mountain. You must toggle the check box on next to each S-graph that you would like to use. A composite S-graph is computed using the decimal weights assigned to each of the standard S-graphs.

The **Plot Unit Hydrograph** button computes the unit hydrograph using the lag time and composite S-graph and displays the results in both a graphical and tabular format.

Clicking on **OK** will write the unit hydrograph data to the UI card for analysis using HEC-1.

Related Topics

- Derived Unit Hydrograph
- Tc from Rational Method

OC Unit Hydro Using GIS Data

GIS data is especially useful in WMS for delineating watersheds and computing Orange County loss rates.

Delineation

Sub-area delineation can be automated when digitial terrain data (DEMs or TINs) exists within WMS. GIS data may be used to identify geometric features such as sub-area boundaries, streams, and concentration points.

Computing Orange County Losses

Soil type and land use data are required in order to compute Orange County losses (Fm and Ybar). Compute losses by choosing the **Compute GIS Attributes** command on the *Calculators* menu or by clicking on the **Compute Losses** button in the Orange County Precipitation Wizard. Losses are computed by overlaying either triangles on a TIN or basin polygons (from a drainage coverage) divided into small squares with the GIS data to calculate an Fm and Ybar value for each square or triangle. Composite loss values for each sub-area can be calculated using the triangles or squares located within the sub-area. Follow these steps to compute losses:

1. Import shapefiles with soil type and land use data
2. Map GIS data to feature objects, if necessary
3. Select *Orange County Losses* for the computation type in the Compute GIS Attributes dialog
4. Choose the coverages, grids, or GIS layers to use for computations
5. Import a mapping table – this table needs to have percent impervious information for each land use in addition to SCS curve numbers
6. Select **OK** to compute losses for all drainage basins

Related Topics

- Creating Watershed Models
- Watershed Delineation with DEMs
- Watershed Delineation with TINs
- GIS Module
- Importing Shapefiles
- Compute GIS Attributes
- Land Use
- Soil Type

Orage County Hydrology

Orange County uses a rational method and unit hydrograph analysis that have been customized to meet their needs for doing hydrologic studies.

WMS provides an interface with a visual representation of all data used in a rational or unit hydrograph analysis that is easy to interact with. There are tools within WMS for using GIS data to automate basin delineation and loss rate computations. Many of the geometric properties derived from geographic features, including area, flow length and slope, can be computed by WMS. It is possible to set up one model and develop time of concentration, using the rational method, and then to run a unit hydrograph analysis. WMS performs all calculations for the rational method and uses HEC-1 to run the unit hydrograph analysis. Small area hydrographs can also be computed.

Related Topics

- Orange County Rational Method
- Orange County Unit Hydrograph Method
- Creating Watershed Models
- GIS Module

6.10. Orange County Rational Method

Orange County Rational Method

WMS includes an interface to the Orange County Rational Method which can be used for computing peak flows on small, mostly urban watersheds. The interface includes the capability to combine runoff from multiple basins.

Use the Orange County rational method to determine runoff rates for watersheds with an area less than or equal to 1 square mile (640 acres) or to estimate time of concentration (Tc) for an Orange County unit hydrograph analysis for any size watershed. Infiltration is accounted for in the losses rather than just using the standard runoff coefficient (based on percent impervious) in the rational equation. Complex reach routing is also factored into the determination of Tc.

Tc is determined at a concentration point by determining the combination of sub-area Tc and routing travel time that produce maximum flow. Given a time of concentration for the outlet, a rainfall intensity can be determined from a rainfall-intensity-duration curve and a peak flow computed.

All of the computations for peak flows and routing are done within WMS.

Related Topics

- Orage County Hydrology
- Orange County Rational Job Control
- Reach Routing
- Sub-Area Data
- Orange County Rational Equation
- Running a Orange County Simulation
- Unit Hydrograph Method

OC Rational Equation

Flows for the Orange County rational method are calculated from the following equation:

$$Q = 0.90(I - F_m)A$$

where:

- Q – peak flow $(ft^3/s\ or\ m^3/s)$
- I – rainfall intensity (in/hr, mm/hr)
- F_m – maximum loss rate, where $F_m = a_p * F_p$
- A – catchment area (acres)

This equation is very similar to the traditional Rational Method equation. The term I - F_m is often referred to as the effective precipitation.

Related Topics

- Losses

OC Losses

Watershed outflow in Orange County is a function of losses. Examples of losses include depression storage, vegetation interception and transpiration, minor amounts of evaporation, and infiltration. Many, if not all, of these losses are affected by the land cover or soil type.

Fm, or the maximum loss rate, is a measurement of the effect of these losses on the peak flow rate. This value corresponds to the soil group, cover complex, and imperviousness of the drainage subarea. $Fm$is calculated by the following equation: $Fm = ap * Fp$where,

- ap – pervious area fraction.
- Fp – pervious loss rate. This is a function of the soil group where different values are shown in the following table:

SOIL GROUP	A	B	C	D
Fp	.40	.30	.25	.20

Group A: Soils with low runoff potential, high infiltration rates.

Group B: Soils with moderate infiltration rates.

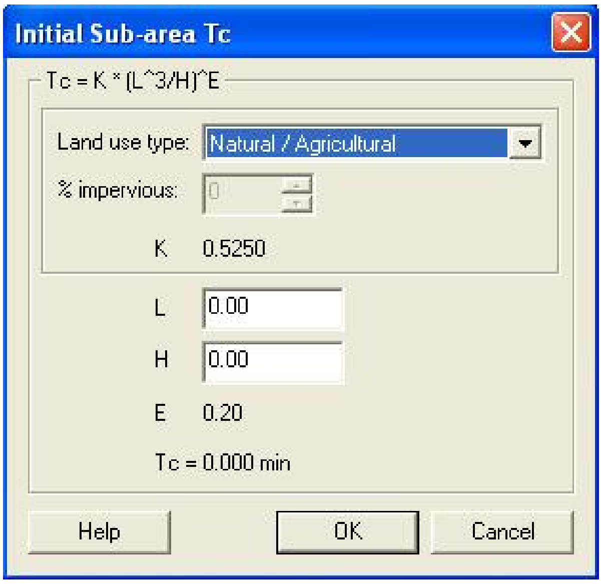
Group C: Soils with slow infiltration rates.

Group D: Soils with high runoff potential, very low infiltration rates.

Related Topics

- Orange County Equation

OC Rational Initial Sub-area Time of Concentration



The time of concentration for initial sub-areas is a critical component in performing a Orange County Rational Method analysis. The time of concentration is defined as the interval of time (minutes) required for the flow at a given point to become a maximum under a uniform rainfall intensity. Generally, the time of concentration is the interval of time from the beginning of rainfall for water from the hydraulically most remote portion of the drainage area to reach the point of concentration. For a Orange County analysis, the time of concentration is calculated using the following equation:

$$Tc = K(L^3/H)^E$$

where:

- K – coefficient depending on percent impervious (land use type)
- L – length of initial sub area flow path (ft)
- H – drop in elevation along the flow path (ft)
- E – Orange County constant = 0.20

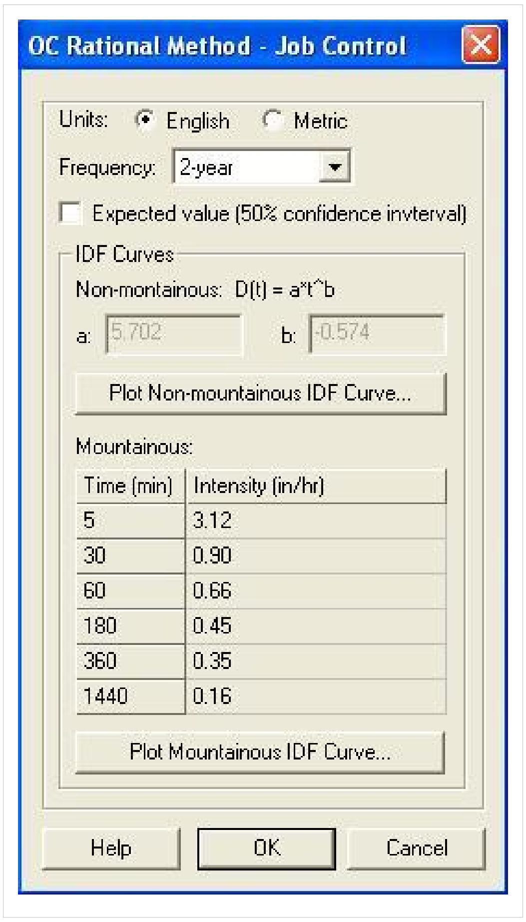
Related Topics

- Sub-area Data

OC Rational Job Control

The Job Control parameters include all of the universal data necessary to run an Orange County analysis that are not a part of a basin, reach, or reservoir. The following are the specific parameters defined as part of job control:

- **Units** – By default Orange County performs computations in English units, however metric unit calculations can be specified.
- **Frequency** – Select the return period of the design storm. Non-mountainous and mountainous IDF curve values are updated based on this selection. The user must enter IDF curve values for the 500-yr storm.
- **Expected Value** – Toggle this check box on to do an expected value analysis. IDF curves and losses will be automatically updated for each sub-area if they have already been computed.



IDF Curves

IDF (intensity-duration-frquency) curves in Orange County are calculated differently depending on the location of the analysis. This section is subdivided into non-mountainous (areas below 2000 feet) and mountainous (areas above 2000 feet) regions.

- **Non-mountainous** – The IDF curves are generated using a regression equation $I = atb$where:
  - I – rainfall intensity (in/hr, mm/hr)
  - t – duration (min)
  - a, b – regression equation coefficients
- **Mountainous** – The IDF curves are generated by interpolating between known intensities at specified time values.

Related Topics

- Sub-area Data
- Reach Routing
- Losses

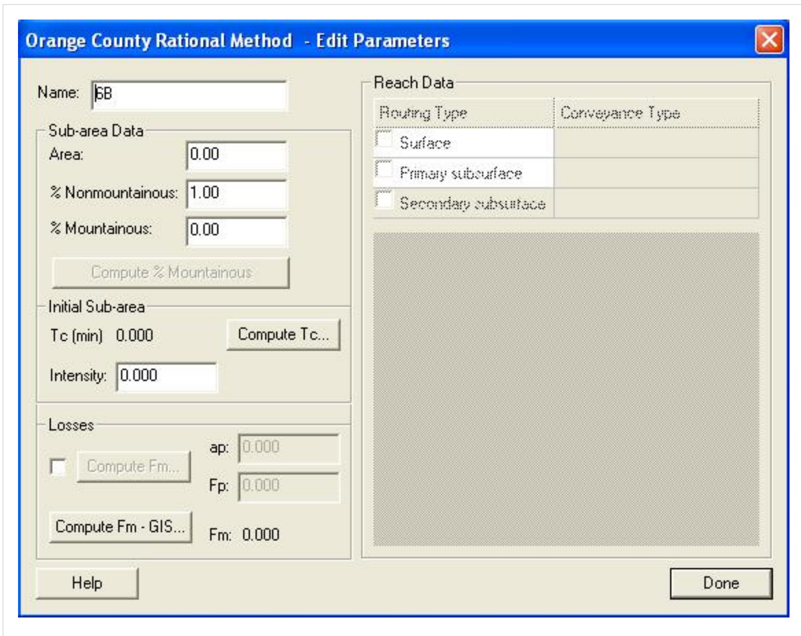
OC Rational Post Processing

View results of an Orange County rational method analysis at any time using the **Display Results** command in the *OC Rational* menu. The time of concentration (Tc) for any concentration point can be used as input for an Orange County unit hydrograph analysis.

Related Topics

- Reports
- Running a Simulation
- Unit Hydrograph Method

OC Rational Reach Routing



Reach routing data is assigned to the upstream node of each reach. Select a node and use the **Edit Parameters** command in the *OC Rational* menu to define data. You can select another node at any time while the *Orange County Rational Method – Edit Parameters* dialog is displayed and view its reach data.

Entering Parameters

Toggle on the checkboxes next to each routing type and select the conveyance type to use any combination of surface (streets and trapezoidal channels) and sub-surface (pipes and culverts) routing. Enter parameters for the highlighted routing type in the spreadsheet below. Values for reach length, up elevation, down elevation, and slope will be computed for you if you are using GIS data.

Routing Computations

Flow will be routed through subsurface conveyance types first and then any excess flow will be routed using the surface conveyance type, if defined. Overflow messages will be displayed in the output report.

Design/Analysis Mode

There is a design and analysis mode if you are using sub-surface circular pipes. Enter the actual pipe diameter for the analysis mode or enter 0.0 for the pipe diameter to use the design mode. The design mode will calculate the smallest standard pipe to carry the computed flow. View the design pipe size here after running a simulation.

Related Topics

- Sub-area Data
- Reports
- Running a Simulation
- Creating Watershed Models

OC Rational Reports

Input and output values of an Orange County rational method analysis are displayed in a spreadsheet after running a simulation. View this report at any time after running a simulation using the **Display Results** command in the *OC Rational* menu. Final results are displayed for each concentration point and, if necessary, the confluence table details are shown afterwards in yellow. It is easy export the data in the report using the Print button or by highlighting the desired data and using Ctrl-C and Ctrl-V. Time of concentration (Tc) values from this report may be used as input for an Orange County unit hydrograph analysis.

Related Topics

- Running a Simulation
- Saving a Simulation
- Post Processing
- Unit Hydrograph Method

OC Rational Run Simulation

Once all relevant data has been recorded and entered into a Orange County analysis, a rational method can be run. Choosing the **Run Simulation** command in the *OC Rational* menu will run a Orange County Rational analysis.

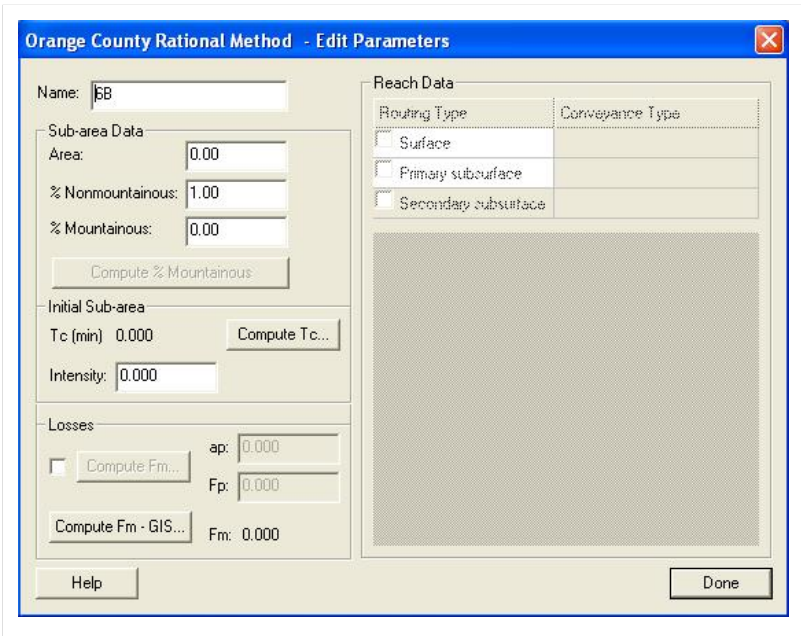
Related Topics

- Orange County Rational Method
- Tree Mapping

OC Rational Saving a Simulation

The **Save Simulation** command in the *OC Rational* menu can be used to save the topologic tree structure with any parameters which have been defined. The **Read Simulation** command will restore the tree and parameters so that you may continue with a particular model.

OC Rational Sub-area Data



When setting up a Orange County Rational Method simulation, data must be entered for each sub-area.

- **Area** – The sub-area area (acres or ha.) can be automatically computed when using a DEM or TIN
- ***%* Nonmountainous and *%* Mountainous** – The percentage of the sub-area that lies below and above 2000 ft, respectively.

Use the **Compute *%* Mountainous** button to calculate these values using a DEM or TIN.

Initial Sub-area

- T_c – The time of concentration (min)
- **Intensity** – The rainfall intensity (in/hr) is automatically computed based on the T_c

Losses

- a_p – Pervious area percentage
- F_p – Pervious area loss rate
- F_m – Maximum area loss rate

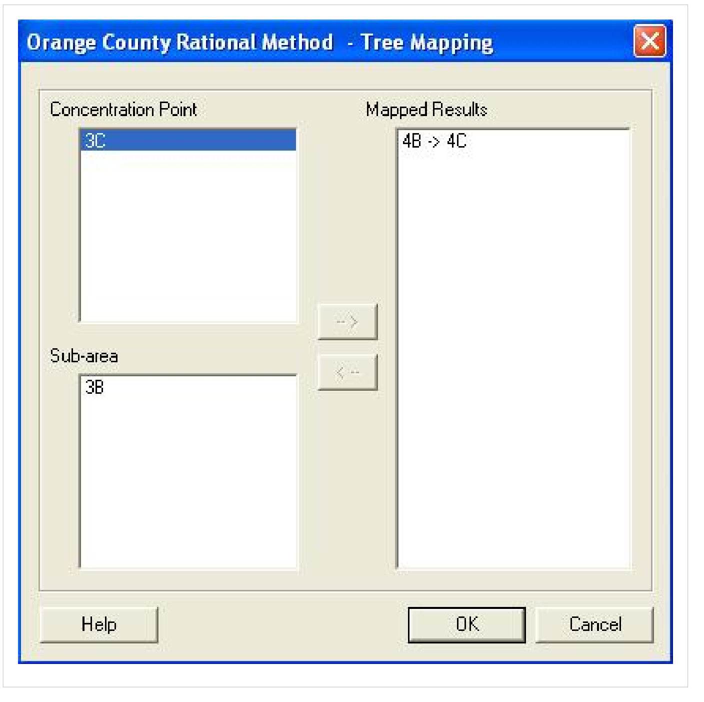
Calculate $F_m$by toggling on the *Compute* $F_m$checkbox, entering values for $a_p$and F_p, and using the **Compute** $F_m$button or click on the **Compute** F_m **– GIS** button to use GIS data.

Related Topics

- Losses
- Initial Sub-area Time of Concentration
- Orange County Rational Equation
- Orange County Rational Method
- Reach Routing
- Creating Watershed Models

OC Rational Tree Mapping

When building a Orange County Rational simulation, where GIS or map data is not available, the *Tree Mapping* dialog can be used to map concentration points to related downstream basins. The reason for this is the possibility of numerous downstream basins being present, thus allowing for the mapping of basins to the correct upstream terminus.



Concentration Point – Contains a list of concentration points that have not been mapped

Sub-area – Contains a list of all possible basins that can be mapped to the selected concentration point

Mapped Results – Contains a list of those basins that have been mapped to concentration points

Use the arrow keys (−− >and < −−) to map and unmap selections.

Related Topics

- Running a Simulation

6.11. Rational Method

Rational Method

WMS includes an interface to the rational method which can be used for computing peak flows on small urban and rural watersheds. The interface includes the capability to combine runoff from multiple basins. Two different methods for determining peak flows/hydrographs at downstream confluences are available.

Traditionally a time of concentration is determined at a downstream confluence by determining the longest combination of time of concentration and routing travel time. Given a time of concentration for the outlet, a rainfall intensity can be determined from a rainfall-intensity-duration curve and a peak flow computed. The hydrograph for the confluence is then determined in the same manner they are determined for sub-basins; by using the peak flow, time of concentration, and a dimensionless hydrograph.

Alternatively, hydrographs for the sub-basins can be computed and then routed (lagged) and combined by summing at the confluence points. When using this method detention basins may be defined at confluence points in order to determine the effect of storage on the computations.

All of the computations for peak flows, hydrographs, and routing are done within WMS.

Saving and Running a Rational Method Simulation

The **Save Simulation** command in the *Rational* menu can be used to save the topologic tree structure with any rational method parameters which have been defined. The **Read Simulation** command will restore the tree and parameters so that you may continue with a particular model.

Related Topics

- Rational Method Equation
- Entering Rational Method Parameters
- Computing Hydrographs with Rational Method

Rational Method Basin Data

If multiple basins are selected while the dialog is opened, the edit fields are grayed out (not dimmed) and follow the rules of multi-selection. For example if you wish to set the value of i (rainfall intensity) for all basins then you can select all basins, open the *Rational Method* dialog, click once on the edit field for i and specify the value. C (runoff coefficient) could be set in a similar fashion, while A(area) cannot be changed when multiple basins are selected.

All values used for computing peak flow with the rational method are stored with the basin attributes so that each time you select a basin the edit fields are filled with the values of that basin. This also makes it possible to use the areas computed from a feature object, DEM, or TIN using the **Compute Basin Data** command in the *TIN* or *DEM Drainage* module menus, or in the *Feature Objects Map* module menu.

Peak flow (Q) for each basin should update automatically, however the **Update Q** button may be used at any time to signify that you are finished entering the value for a given parameter (alternatively you can tab to the next field, or click in another edit field).

Peak flow values for all selected basins will be computed and displayed in the main text window of the *Rational Method* dialog.

The time of concentration, which is used to determine an appropriate rainfall intensity (i), can be determined in one of four ways:

1. The time of concentration can be computed “outside” of WMS and entered into the appropriate edit field.
2. The Compute T_c – Basin Data button can be chosen and one of the time of concentration (or lag time) equations specified (this option is only available when you have computed basin data from either a TIN or a DEM).
3. A series of time computation arcs may be used to define overland, sheet, and channel flow within a basin and then travel times for each arc are summed to compute the total travel time or time of concentration for the basin. The Compute T_c – Map Data button accesses the dialog that allows you to combine arcs within the currently selected basin to compute a time of concentration.
4. Finally, the kinematic wave equation can be used from within the *IDF curves* dialog.

Related Topics

- Outlet Data
- Rainfall Intensity
- Entering Rational Method Parameters

Rational Method Computing Hydrographs

The Rational Method equation is designed to compute peak flows. However, a hydrograph, based on the peak flow and basin (or outlet) time of concentration, can be computed using one of five different unit hydrographs. Furthermore, there are two different methods (traditional and route by summing) hydrographs can be computed at confluence points.

The *Rational Method Hydrographs* dialog is used to specify the dimensionless unit hydrograph method, and the way hydrographs at outlets are computed.

Related Topics

- Traditional vs Route by Summing
- Rational Hydrograph
- Modified Rational Hydrograph
- Dekalb Rational Hydrograph
- Universal Rational Hydrograph
- User Defined Rational Hydrograph

Rational Method Edit Parameters

Like the other models supported by WMS, the Rational Method can be defined for a watershed/catchment developed from feature objects, DEMs, or TINs, or built using the tools provided in the hydrologic modeling module under the *Tree* menu. Once the topologic tree has been constructed the *Rational Method* dialog can be accessed using the **Run Simulation** command from the *Rational* menu.

The *Rational Method* dialog allows you to enter all of the necessary values for computing a peak flow for a selected catchment area, or confluence point. The values for C, i, and Arepresent the values of the currently selected basin or outlet.

Related Topics:

- Basin Data
- Outlet Data
- Rainfall Intensity
- Rational Method Equation

Rational Method Equation

The rational method is used around the world for peak flow estimation of small rural drainage basins and is the most widely used method for urban drainage design. The rational method equation is given below:

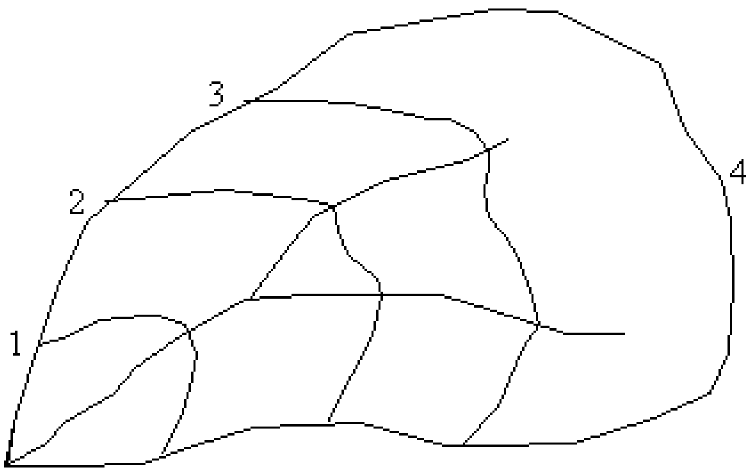
$$Q = kCiA$$

where:

- Q – peak flow (cfs or m^3/s).
- k – conversion factor equal to 1.008 (SI) or .00278 (metric).
- C – dimensionless runoff coefficient.
- i – rainfall intensity (in/hr, mm/hr).
- A – catchment area (acres, ha).

The rational method is generally considered to be an approximate model for computing the flood peak resulting from a given rainfall, with the runoff coefficient accounting for all differences between the rainfall intensity and the flood peak. Such differences result from infiltration, temporary storage, and other losses. A table of Cvalues recommended by the American Society of Civil Engineers and Water Pollution Control Federation is found in the article Runoff Coefficient Table.

The Rational equation was developed from a simplified analysis of runoff using isochrones, or lines of equal travel time with areas in acres between them as illustrated in the figure below. The method assumes no temporary storage in the basin, so the ratio between the peak runoff and the rainfall intensity is then the same as the ratio of the volumes of runoff and rainfall. If a constant rainfall intensity (in/hr) begins at time t=0 and has a duration of the time of concentration (t_c) for the basin, the hydrograph will reach an instantaneous peak at Ci. The $t_c$of the basin can be thought of as the time after rainfall excess begins to when all portions of the watershed are contributing to the peak flow at the outlet. If the duration is longer than t_c, the hydrograph will remain constant after reaching a value of $Ci$for a time period equal to the difference of the rainfall duration and t_c. In either case the time of rise and time of recession are equal to t_c.



With such a derivation, the effects of rainfall and basin size are accounted for explicitly, while most other physical characteristics of the watershed are accounted for indirectly by the time of concentration and runoff coefficient. This simple equation illustrates the critical nature of t_c. For durations less than t_c, the entire area is not contributing. For durations larger than $t_c$there is no increase in contributing area, and therefore no increase in peak flow.

Related Topics

- Important Limitations
- Runoff Coefficient Table
- Entering Rational Method Parameters
- Computing Hydrographs with Rational Method

Rational Method Outlet Data

Some of the information for outlet points is automatically determined from the contributing area upstream, while other parameters are entered separately. The area is determined by summing the area of all upstream sub-basins. The runoff coefficient is computed from the upstream basins using the area-weighted equation shown below.

$$C_{OUTLET} = \frac{\sum A^i C_{BASIN^i}}{A}$$

where:

C_{OUTLET}= The runoff coefficient for the outlet.

C_{BASIN^i}= The runoff coefficient for the i^{th} upstream basin.

A^i= The area of the i^{th} upstream basin.

A= The total upstream area at the outlet/confluence.

The time of concentration is determined by computing the longest combination of upstream time of concentration and channel travel time to the given outlet point. For example if a given outlet point had two contributing sub-basins the time of concentration for the outlet would be the longest time of concentration of the two upstream sub-basins. If there were other outlets upstream of the given outlet then the travel time would be added, and again the longest time or combination of times would be used as the time of concentration for the outlet.

The rainfall intensity value should be supplied separately for the outlet in the same way it is for a sub-basin. However, it is a function of the time of concentration and can be determined from an IDF curve relationship.

There are two ways routing of a hydrograph can be accounted for using the WMS implementation of the rational method. The first is to simply apply a time of travel between outlets. When hydrographs are computed (only with the summing method) at downstream outlets they are lagged by the travel time and added with other contributing basins. In addition, simple level-pool reservoir routing may be performed on an outlet hydrograph before it is routed downstream. Again, both of these options are available only when choosing the summing method of hydrograph generation rather than the traditional method where a peak flow and resulting hydrograph are determined from the time of concentration (and therefore rainfall intensity) at the outlet point.

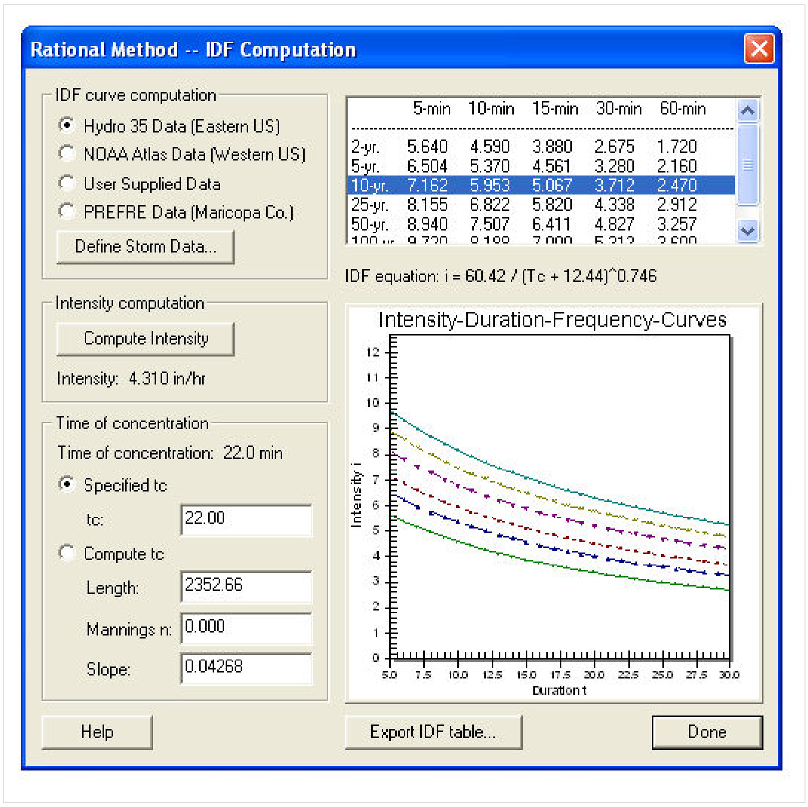
Related Topics

- Entering Rational Method Parameters
- Basin Data
- Rainfall Intensity

Rational Method Rainfall Intensity

Precipitation intensity-duration-frequency (IDF) information is necessary for the specific locality in which the Rational Method will be used. In general this is done using either HYDRO-35 or NOAA Atlas 2 data. Using the IDF Curves… button (for either basins or outlets), WMS can be used to develop curves from either of these two data sources, statistically derived data, or you can directly enter an i value if you typically compute it in another way or already know the design value you want to use.

The *IDF Computation* dialog (shown below) can be used to create a series of T-year IDF curves from HYDRO-35, NOAA Atlas 2, or user defined data.



The type of data that will be used to create the IDF curves is specified with the radio group options in the upper left portion of the dialog. The **Define Data** button can then be used to bring a dialog which allows you to enter the following values, depending on the data type specified:

- **HYDRO-35 –** The 2-yr 5, 15, and 60 minute rainfall values, and the 100-yr 5, 15, and 60 minute rainfall values. The HYDRO-35 maps for determining the six required rainfall values have been included below.
- **NOAA Atlas 2 –** The 2-yr 6 and 24 hour rainfall values, the 100-yr 6 and 24 hr rainfall values and the mean basin elevation. When entering rainfall values they are entered as depth values and not intensity values.
- **User Defined –** The recurrence interval, and the 5, 10, 15, 30 and 60 minute rainfall values. With this option only a single curve for the recurrence interval will be generated.

The methods WMS uses to compute the IDF curve equations are discussed in Appendix A of FHWA HEC-12 [1] and in FHWA HDS-2 [2].

Once the specified data has been entered the corresponding IDF-curves will be generated and plotted in the graphics window of the *IDF Computation* dialog. You can then specify the recurrence interval you want to use for analysis by selecting it in the text list window in the upper right portion of the dialog. After selecting the recurrence interval, the appropriate curve in the plot window will be displayed in red. Finally, an intensity value, i, is determined by specifying a time of concentration as outlined below and then clicking on the Intensity button. When clicking on the **Done** button for the *IDF Computation* dialog, the computed intensity value will automatically be updated in the edit field for i of the *Rational Method* dialog.

Computing the Time of Concentration

A time of concentration value needs to be entered in order to determine the intensity value to be used in the Rational Method equation. This value can either be entered manually or computed from the overland flow length, Manning's n, and slope. One equation used to compute the time of concentration from basin geometric parameters is the kinematic wave equation:

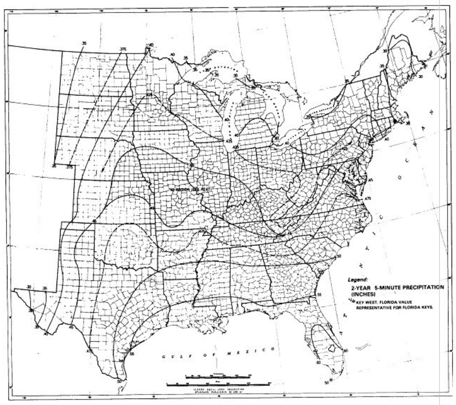
$$t_c = \frac{KL^{0.6}n^{0.6}}{i^{0.4}S^{0.3}}$$

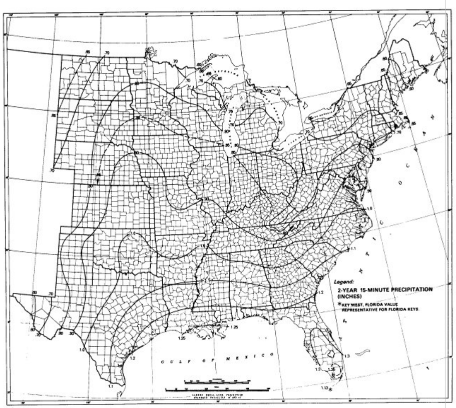
where:

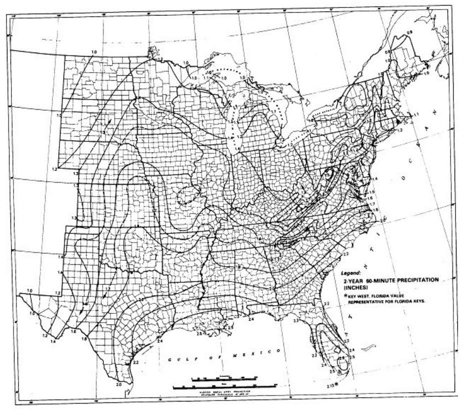
- tc= the time of concentration.
- L= overland flow length.
- n= Manning's roughness coefficient.
- i= rainfall intensity.
- S= average slope of the overland area.
- K= 0.93

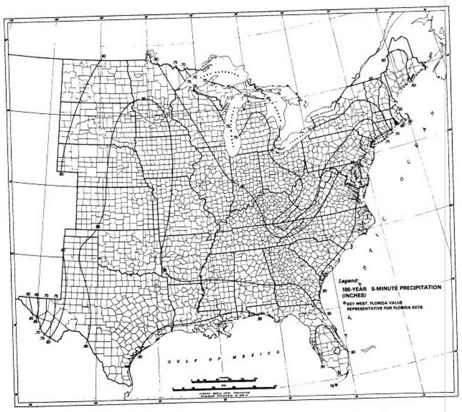
This method has been adopted by the FHWA for general use, but other equations can be used as defined elsewhere

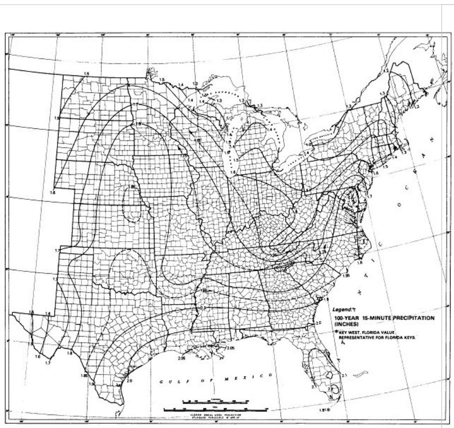
HYDRO-35 maps

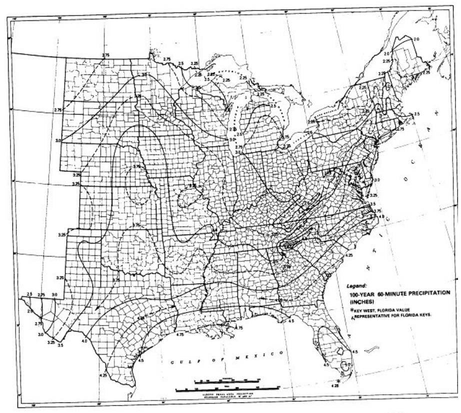

2 years 5 minutes


2 years 15 minutes


2 years 60 minutes


100 years 5 minutes


100 years 15 minutes


100 years 60 minutes

Related Topics

- Entering Rational Method Parameters
- Basin Data
- Outlet Data

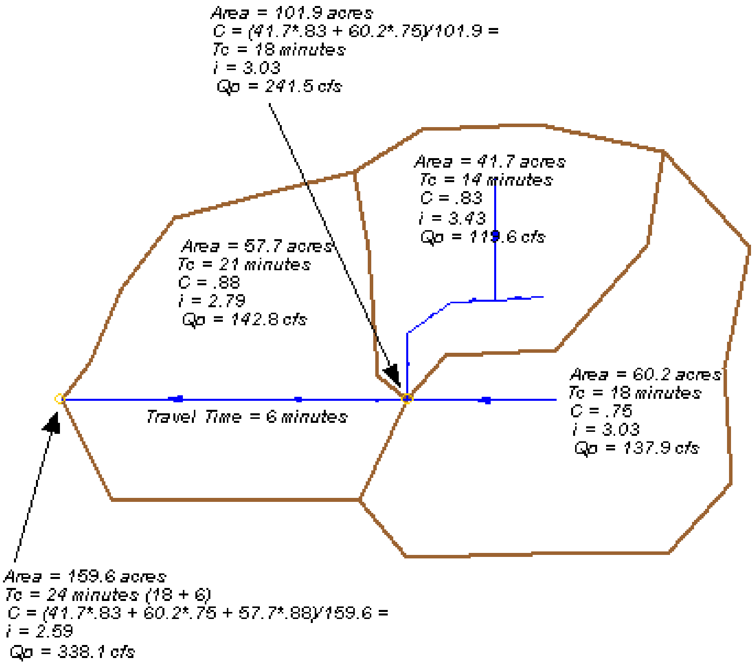
References

[1] http://www.fhwa.dot.gov/engineering/hydraulics/library_arc.cfm?pub_number=52&id=28
[2] http://www.fhwa.dot.gov/engineering/hydraulics/library_arc.cfm?pub_number=2&id=6

Rational Method Traditional vs Route by Summing

The main difference between these two methods is that the traditional method computes a peak flow at an outlet by determining composite C, t_c, and areas from upstream basins and tributaries. Using the composite t_ca rainfall intensity from an IDF curve is chosen to compute peak flow and then one of the hydrograph methods is used to develop a runoff hydrograph. The route by summing method, on the other hand, computes peak flows and hydrographs for basins in the traditional manner, but hydrographs at outlet points are determined by combining or summing the hydrographs from upstream basins and tributaries. The lag time determines the timing offset for hydrographs that "arrive" at an outlet through a tributary channel. With this method you also have the option of defining a detention basin, through which the outlet hydrograph may be passed. The detention basin is defined and calculated using the same techniques as outlined in the hydrologic calculators (see Detention Basin Calculator).

The differences can probably best be understood with the following example. The figure below shows a diagram of the basin containing two "upstream" basins that combine at a junction and are routed through a lower basin. The areas, runoff coefficients, times of concentration and lag times for each basin and for the outlets (using the traditional method) are as shown.



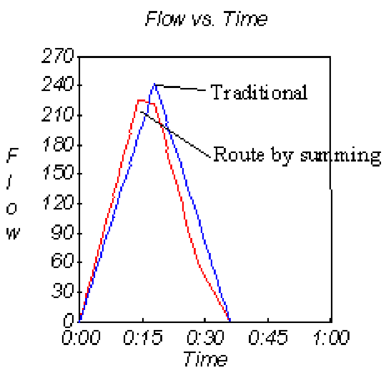
Sample Rational Method Model

Rainfall intensities are determined using the times of concentration and an IDF curve. As would be expected the higher the time of concentration, the smaller the intensity.

Peak values for the three sub-basins are identical for each of the two methods. However, for the outlet points the peak flows (and therefore hydrographs) are different.

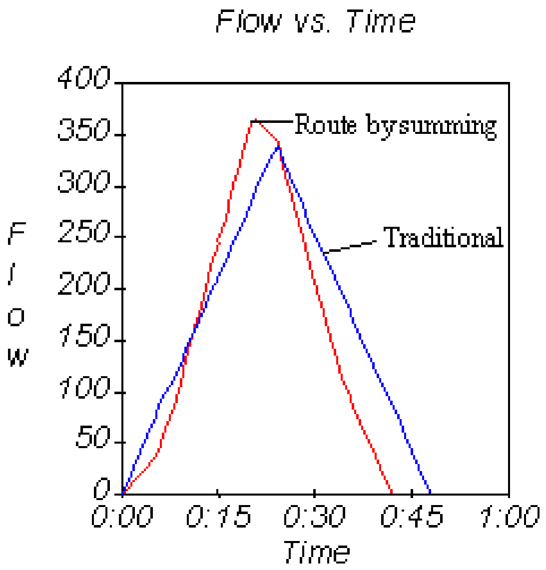
In the case of the traditional method the time of concentration for the "upstream" outlet (the location where the two upper basins join) is determined to be 18 minutes (the largest $t_c$of the two basins). From this t_ca rainfall intensity of 3.03 is determined from the IDF curve. The contributing area is 101.9 acres (the sum of the two upstream basins), which when using equation 13.1 results in a peak flow of 241.7 cfs.

For the route by summing method the two upstream hydrographs are summed and the resulting peak flow value is determined. In our example the standard rational method (triangular shaped hydrograph) option was chosen. Since the times of concentration are different (and therefore the times to peak for the hydrographs) the resulting peak flow is not the sum of the two basin peak flows, but as can be seen in below, is 225 cfs.



Rational Method Hydrograph Comparison Upper Outlet

For the lower outlet point a similar comparison can be made. The only difference is that the time of concentration is the longest time between the time of concentration of the lower basin (21 minutes), and the longest time of concentration of the two upstream basins plus the travel time through the tributary (18 + 6 = 24 minutes). In this case it is 24 minutes. Using the traditional method the rainfall intensity for a time of concentration equal to 24 minutes is 2.59, the total summed area is 159.6 acres and the resulting peak flow 338.1 cfs. The peak flow for the route by summing methods is 366 cfs, and the comparison of hydrographs using the standard triangular rational method dimensionless unit hydrograph is shown below.



Rational Method Hydrograph Comparison Lower Outlet

One final difference is that with the route by summing method the addition of reservoirs (detention basins) in the calculations is possible. However, with either method a resulting hydrograph could be used in the design of a detention basin as a separate operation.

Related Topics

- Computing Hydrographs with Rational Method
- Rational Method Equation

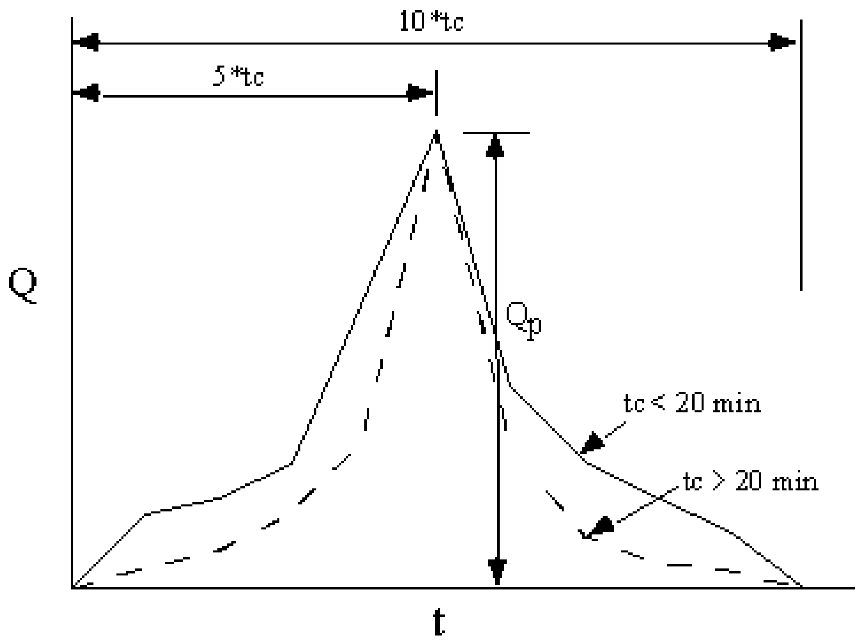
Dekalb Rational Hydrograph

The Dekalb Rational Hydrograph was developed by Dekalb County Georgia, and like the Universal Rational Hydrograph, ordinates are computed by scaling the peak discharge by an appropriate value. The time to peak occurs at $5 * t_c$, while the time base is $10 * t_c$. All coefficients occur at increments of t_c, and are different depending on whether $t_c$is less than 20 minutes or not. The following table lists the coefficients and a typical hydrograph is shown below.

Dimensionless Time and Hydrograph Ordinates		
t / t_c	Q / Q_p **for** t_c **< 20 min**	Q / Q_p **for** t_c **>= 20 min**
0	0.00	0.00
1	0.16	0.04
2	0.19	0.08
3	0.27	0.16
4	0.34	0.32
5	1.00	1.00
6	0.45	0.30
7	0.27	0.11
8	0.19	0.05
9	0.12	0.03
10	0.00	0.00

where:

- t_c= time of concentration.
- Q= Flow at time t, in cfs.
- Q_p= Peak flow.



Related Topics

- Computing Hydrographs with Rational Method
- Rational Method Equation

Important Limitations

Due to assumptions regarding homogeneity of rainfall and equilibrium conditions at the time of peak flow, the rational method should not be used on areas larger than about 1 square mile without subdividing the overall watershed into sub-basins including the effect of routing through any drainage channels.

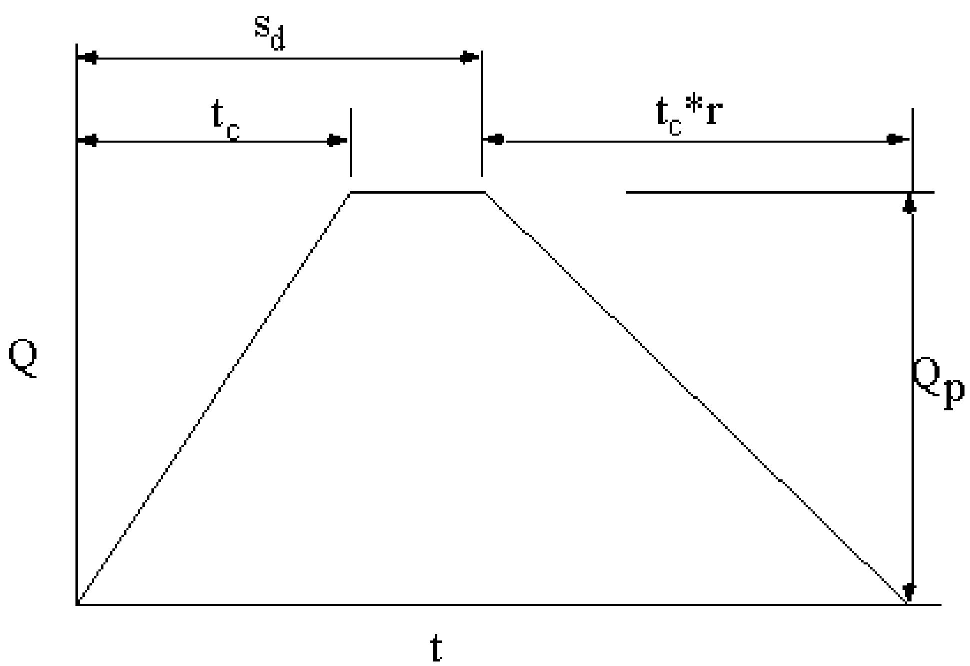
WMS includes two different methods for determining runoff from larger watersheds, subdivided into smaller sub-basins, including the ability to account for routing and lag through drainage channels and detention basins.

Related Topics

- Rational Method Overview
- Rational Method Equation

Modified Rational Hydrograph

The Modified Rational Hydrograph also assumes that the time to peak is equal to the t_c, but allows for the duration of the storm to be longer than t_c, resulting in a trapezoidal shaped hydrograph as shown below. A coefficient to modify the slope of the receding limb may also be applied with this method.



where:

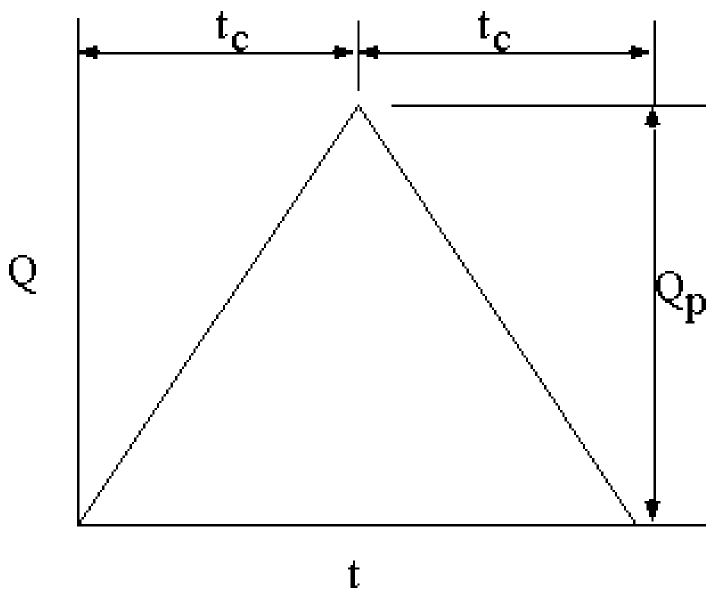
- t_c= time of concentration.
- Q= Flow at time t, in cfs.
- Q_p= Peak flow.
- r= falling limb coefficient
- s_d= storm duration

Related Topics

- Computing Hydrographs with Rational Method
- Rational Method Equation

Rational Hydrograph

The Rational Unit Hydrograph has a time to peak equal to the time of concentration. Both the rising and receding limbs of the hydrograph have a duration equal to the time of concentration, and therefore the shape of the hydrograph is an isosceles triangle with a time base of $2t_c$ as shown below. This method is chosen by specifying the Modified Hydrograph Method using 1.0 for the recession limb coefficient and t_c for the storm duration.



Related Topics

- Computing Hydrographs with Rational Method
- Rational Method Equation

Runoff Coefficient Table

Area Description	Runoff Coefficient C
Business	
Downtown	0.70-0.95
Neighborhood	0.50-0.70
Residential	
Single-Family	0.30-0.50
Multiunits, detached	0.40-0.60
Multiunits, attached	0.60-0.75
Residential (suburban)	0.25-0.40
Apartment	0.50-0.70
Industrial	
Light	0.50-0.80
Heavy	0.60-0.90
Parks, cemeteries	0.10-0.25
Playgrounds	0.20-0.35
Railroad yard	0.20-0.35
Unimproved	0.10-0.30
Character of surface	Runoff Coefficient C
Pavement	
Asphaltic and concrete	0.70-0.95
Brick	0.70-0.85
Roofs	0.75-0.95
Lawns, sandy soil	
Flat, 2 percent	0.05-0.10
Average, 2-7 percent	0.10-0.15
Steep, 7 percent	0.15-0.20
Lawns, heavy soil	
Flat, 2 percent	0.13-0.17
Average, 2-7 percent	0.18-0.22
Steep, 7 percent	0.25-0.35

Related Topics

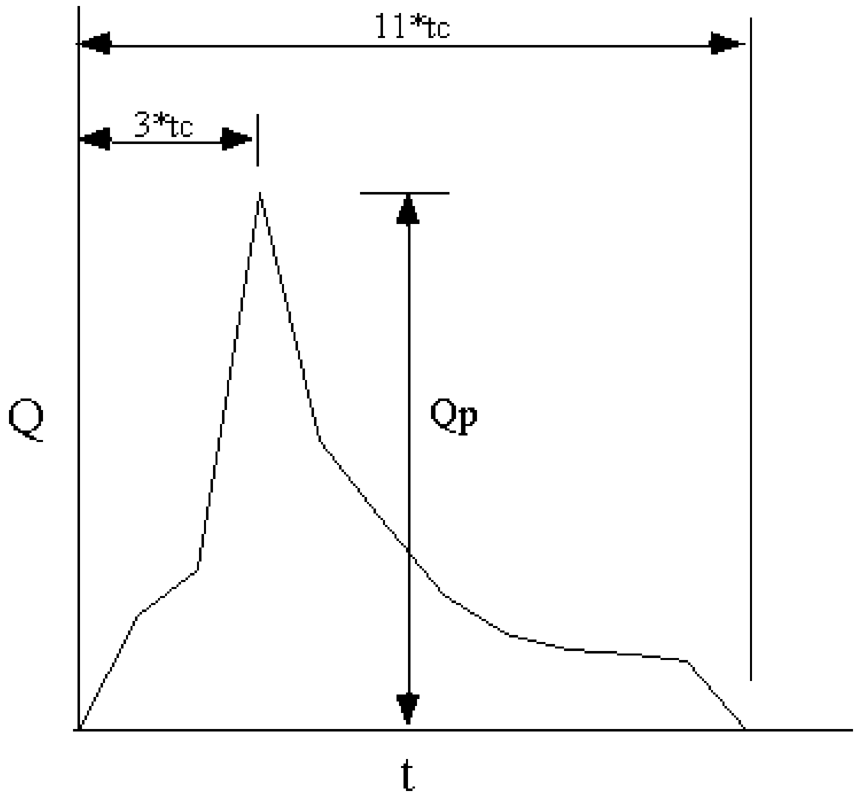
- Rational Method Equation
- Rational Method Overview
- Basin Data

Universal Rational Hydrograph

The Universal Rational Hydrograph uses a set of coefficients and the peak discharge to compute the hydrograph ordinates at different times. The coefficients are shown in the following table and the resulting hydrograph is displayed.

Dimensionless Time and Hydrograph Ordinates	
t/t_c	Q/Q_p
0	0.00
1	0.21
2	0.30
3	1.00
4	0.54
5	0.39
6	0.25
7	0.18
8	0.15
9	0.14
10	0.13
11	0.00

Notice that the peak occurs at $3 * t_c$, and the time base is equal to $11 * t_c$.



where:

- t_c= time of concentration.
- Q= Flow at time t, in cfs.
- Qp= Peak flow.

Related Topics:

- Computing Hydrographs with Rational Method
- Rational Method Equation

User Defined Rational Hydrograph

With the User Defined method of hydrograph generation, you specify the number of ordinates for a dimensionless unit hydrograph and then define the $t/\ t_c$and $Q/\ Q_p$values.

NOTE: All $t/\ t_c$values must be even integers.

User defined files may be exported or imported so that they do not need to be re-entered from one run of WMS to the next.

Related Topics

- Computing Hydrographs with Rational Method
- Rational Method Equation

6.12. Storm Drain

Storm Drain

The storm drain coverage is used for defining urban hydraulic models. The procedure for defining hydraulic models in this coverage is similar to defining other hydraulic models since streams, ditches, and sewer pipes are created from upstream to downstream. Currently, this coverage only supports links and nodes for the SWMM model.

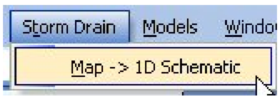
When you have created your conceptual model in the storm drain coverage, select the **Map→1D Schematic** menu item from the *Storm Drain* menu to generate a hydraulic schematic from the coverage. After you have done this, you can define your model-specific data on the hydraulic schematic in the river module.

In other words, here are the steps you would take to define a simple urban hydraulic model:

1. Create a new storm drain coverage.
2. Define any pipes, ditches, and streams as arcs going from upstream to downstream in the storm drain coverage.
3. Select the **Map to 1D Schematic** menu item
4. Go to the River Module
5. Define your model-specific data in the river module (currently, only SWMM model data is supported).

Storm Drain Menu

The *Storm Drain* menu is available in the map module when a Storm Drain coverage type is selected as the active coverage. This menu contains the **Map→1D Schematic** menu command. The menu is shown below:



Map to 1D Schematic

The **Map→1D Schematic** menu item creates a hydraulic schematic based on the configuration of the arcs and nodes in a Storm Drain or 1D Hydraulic Centerline coverage. If a schematic already exists for this coverage, the existing schematic is updated based on the coverage configuration.

Storm Drain Simulations

Run Simulation

You can build an FHWA Storm Drain simulation by creating *Storm Drain-FHWA* type coverage. The Storm Drain coverage is used for building HY-12 and SWMM models.

When you run an FHWA Storm Drain simulation (by invoking the **Run Simulation** of the *Storm Drain-FHWA* menu), an input file is saved and the storm drain program started using the defined input file. An output file name is also required and all of the output generated by the hydraulic analysis is saved to this file.

Plots of results can be created by reading the output file generated using the **Read Solution** command and then setting up an FHWA Storm Drain plot using the *Plot Wizard*.

Read Solution

If you have run the FHWA storm drain model generated by WMS and you have a solution file, plots of results can be created by reading the output file generated using the **Read Solution** command and then setting up an *FHWA Storm Drain* plot using the *Plot Wizard*.

Save Simulation

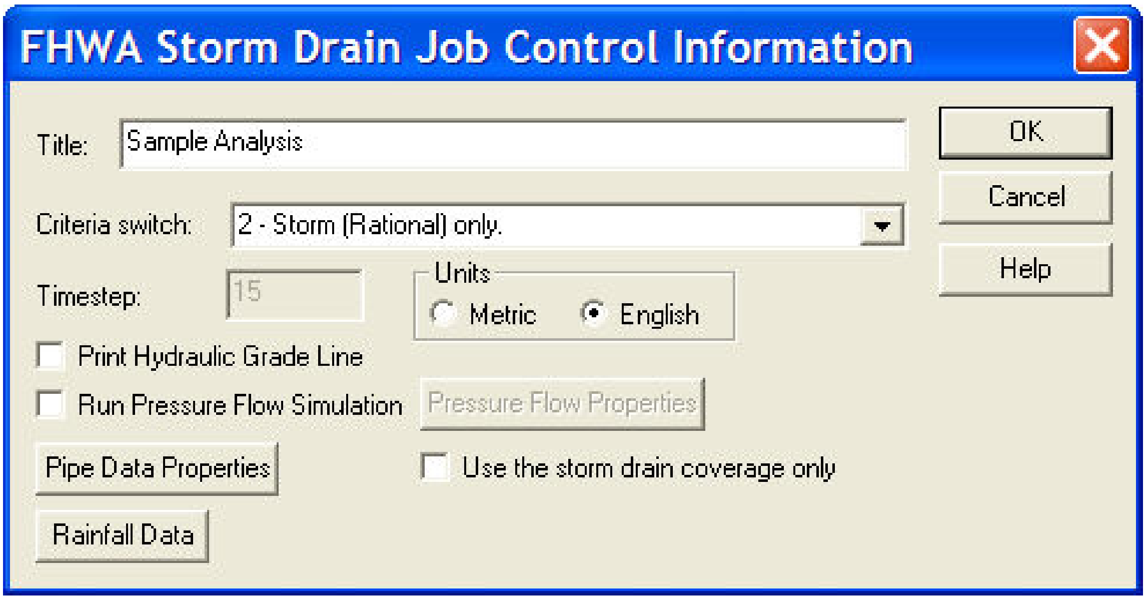
If you have defined arcs representing pipes and nodes in a Storm Drain-FHWA coverage, you can save the simulation using the **Save Simulation** command. A Storm Drain model will be saved in the correct format to run the FHWA storm drain model.

Related Topics

- Storm Drain Job Control
- Plot Wizard

Storm Drain Job Control

Global parameters for a storm drain analysis are defined in the *Job Control* dialog.



Title

The title is placed at the top of the input file to identify the model run.

Criteria Switch

Two different kinds of storm drain analysis can be performed. Option 2 is Storm only and uses the rational method (peak flow analysis) to generate the flows for the pipe network. Contributing areas, runoff coefficients, and times of concentration must be define for each storm inlet point. An intensity, duration, frequency curve must also be defined for the modeling area.

The second option is also Storm only (at this point sanitary is not an option) but uses hydrographs to define the inflow rather than the rational method analysis. In this case hydrographs can be computed from any of the models supported by WMS (i.e. HEC-1, rational, TR-55, etc.) and then mapped to the storm drain inlets.

Time Step

When running a hydrographic analysis a time step should be entered to indicate the timing for each step of the analysis.

Units

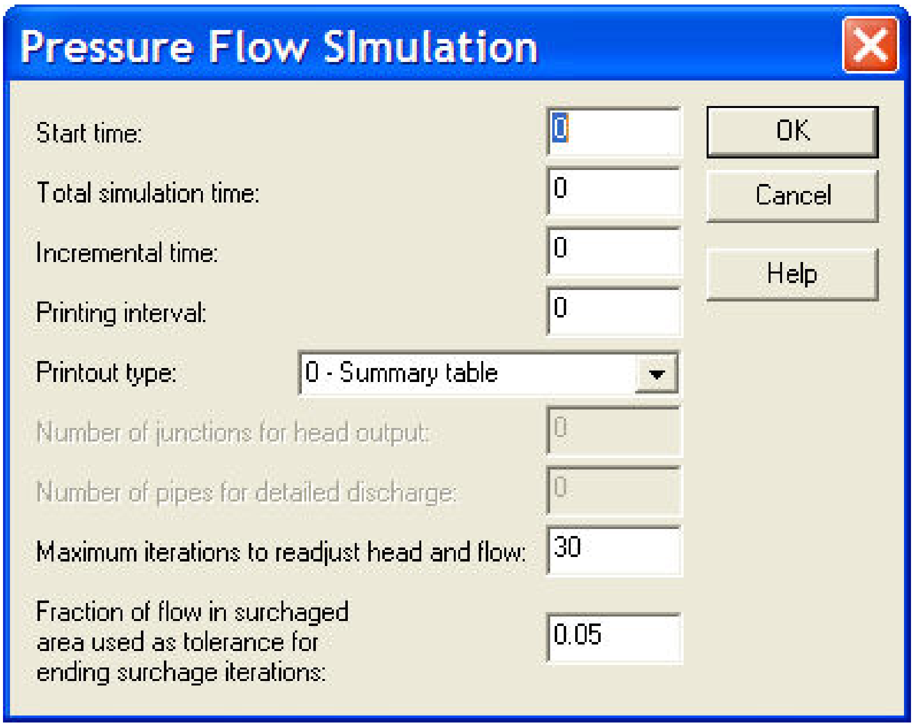
Storm drain can be run using either Metric or English units. Parameters entered should be consistent with the units specified.

Print Hydraulic Grade Line

Turn this option on to see a printout of the hydraulic grade line in the output file.

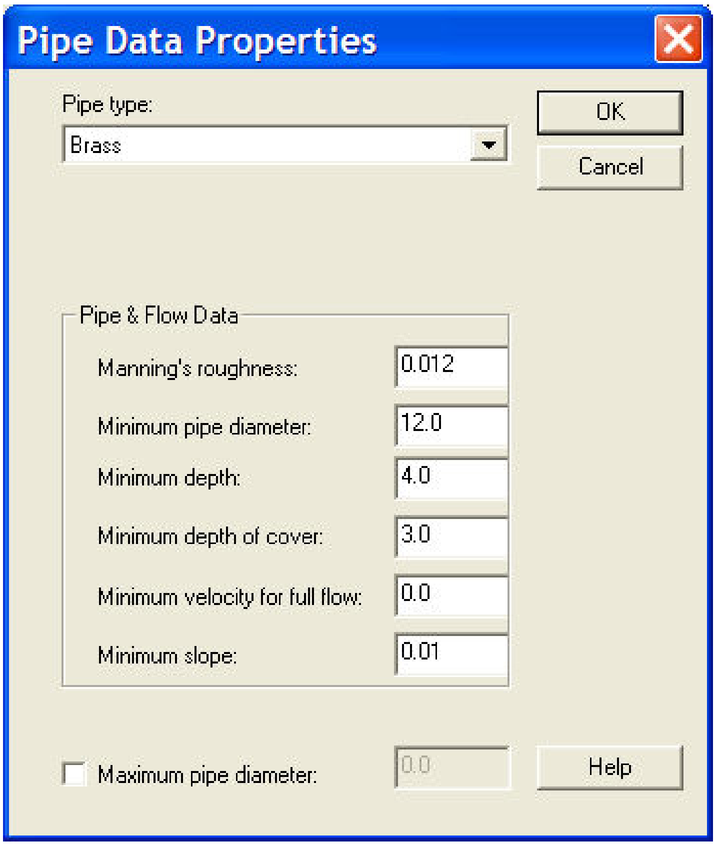
Run Pressure Flow Simulation

When running a pressure flow simulation the following properties must also be defined (see help strings for each edit field when defining for more information).



Pipe Data Properties

Default pipe properties are assigned in the *Job Control* dialog. If individual pipe segment properties are not defined as part of the attributes of the feature lines that make up the pipe network they will automatically inherit the properties defined here. The following dialog lists the pipe properties that must be defined.



Rainfall Data

When running a rational simulation an intensity duration frequency curve must be defined for the entire area (only a single curve can be used). The **Rainfall Data** button allows you to enter manually the values defining a curve, or use the WMS IDF Curve generation dialog to set up an IDF curve for different areas of the United Stated.

Use the storm drain coverage only

A storm drain analysis is comprised of two data layers. The pipe network and the surface drainage. WMS coverages can be used to develop the parameters for both of these layers (storm drain and drainage coverages), but for some analysis it may be preferable to analyze a pipe network without having to define an entire surface drainage coverage. In this case you can toggle this option on and the necessary surface drainage information can be defined from the nodes of the storm drain coverage rather than from a drainage coverage. These parameters include the area, tc, and runoff coefficients for the basins that connect to a given node in the pipe network.

Related Topics=

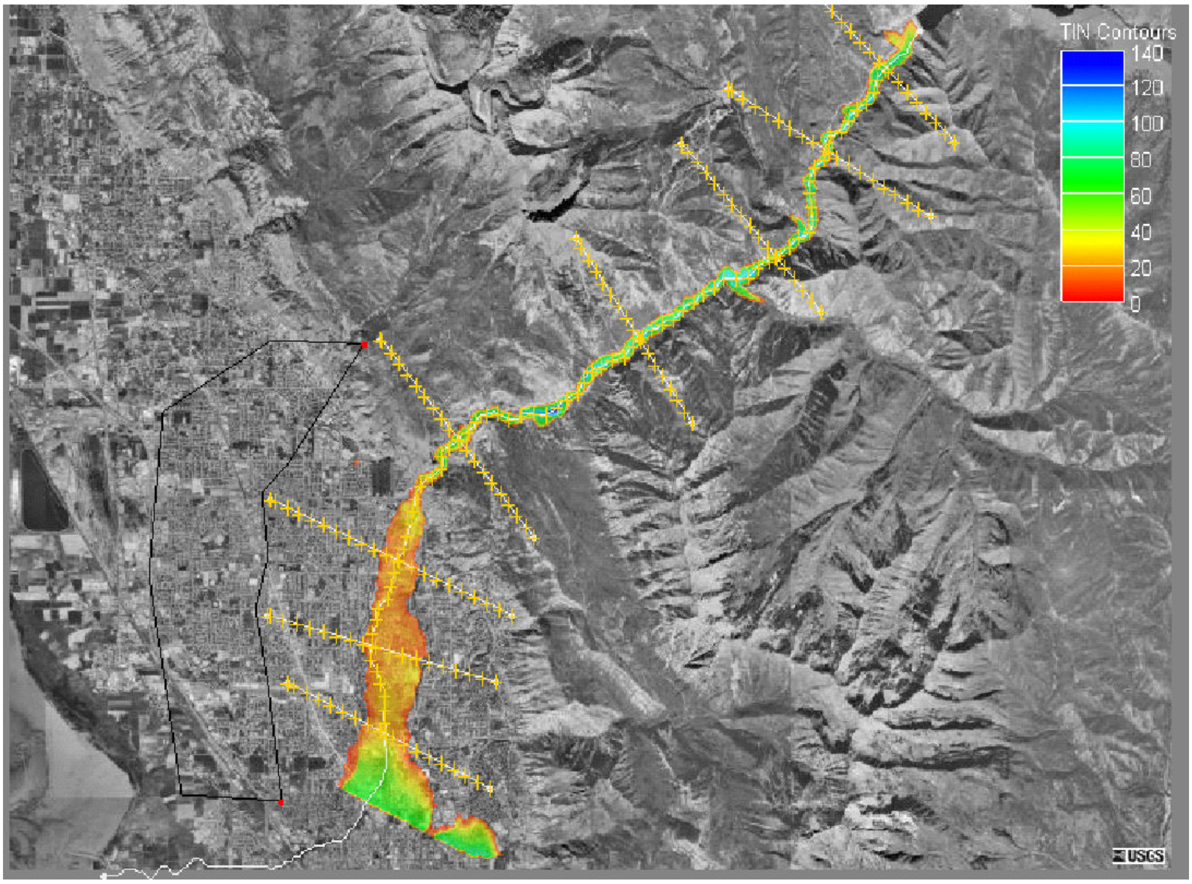
- Storm Drain Modeling

6.13. Simplified Dam-Break (SMPDBK)

SMPDBK

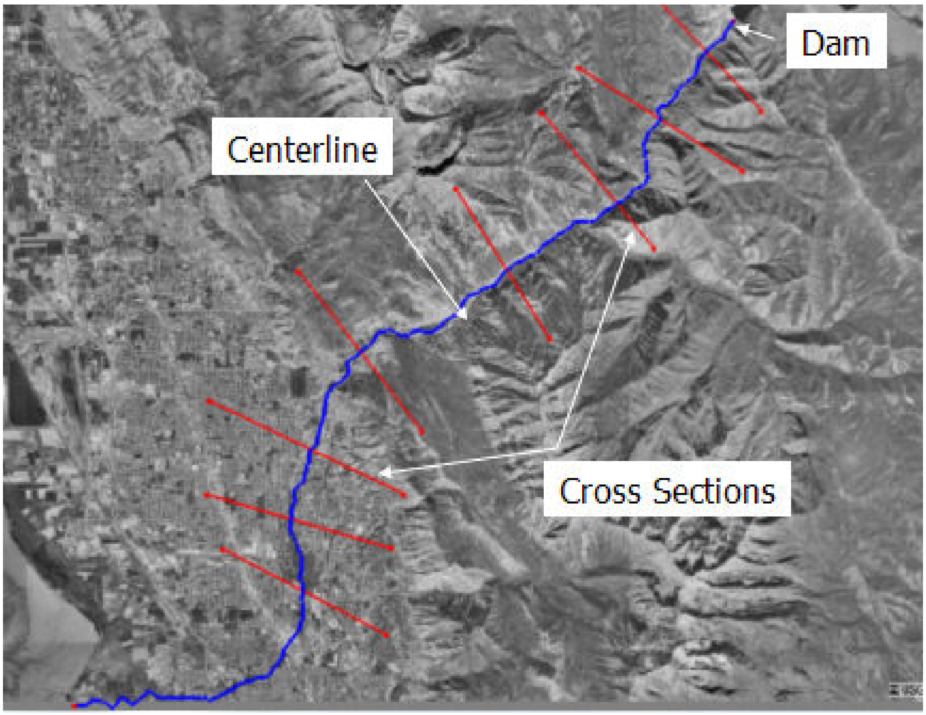
The **Simplified Dam-Break (SMPDBK)** was developed by the National Weather Service (NWS) for predicting downstream flooding produced by a dam failure. This program is still capable of producing the information necessary to estimate flooded areas resulting from dam-break floodwaters while substantially reducing the amount of time, data, and expertise required to run a simulation of the more sophisticated unsteady NWS DAMBRK, or now called FLDWAV. The SMPDBK method is useful for situations where reconnaissance level results are adequate, and when data and time available to prepare the simulation are sparse. Unlike the more sophisticated versions of DAMBRK and FLDWAV, the SMPDBK method does not account for backwater effects created by natural channel constrictions of those due to such obstacles as downstream dams or bridge embankments.

The input required for a SMPDBK model is a stream centerline, cross sections, and information regarding the storage and failure of the dam being modeled. WMS saves the model data to a properly formatted input file for SMPDBK and then launches the executable. The executable is the same version distributed by the NWS. When a model is successfully run, WMS will automatically read the results and create a water surface elevation data set that can be used for automated floodplain delineation as illustrated in the picture below.



Centerline

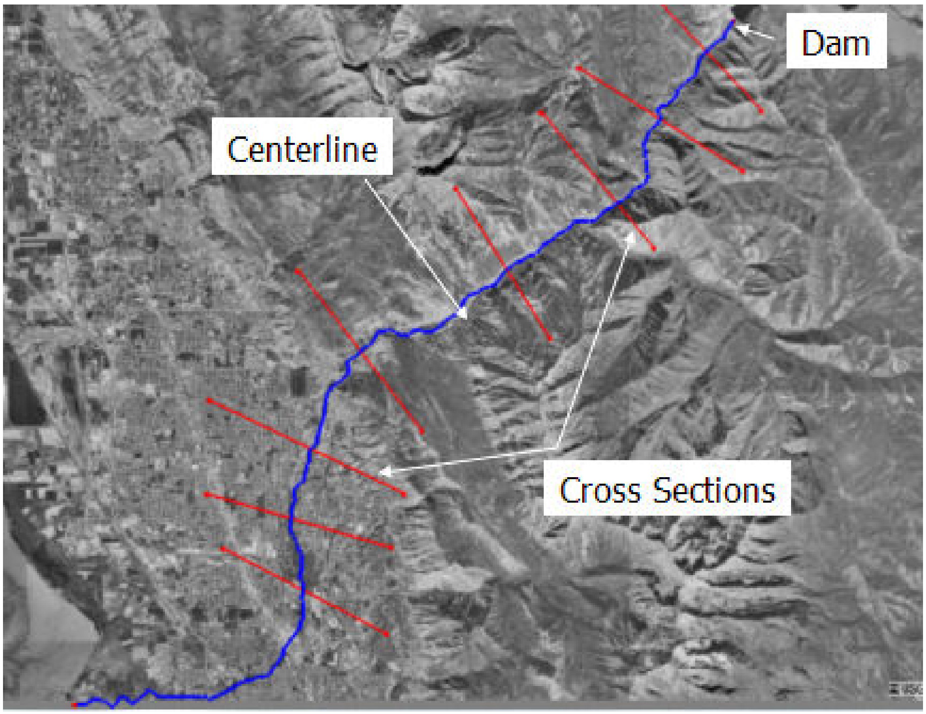
The centerline must be defined in a 1D-Hydraulic Centerline coverage and consists of a single feature arc that should start at the dam being modeled and end at or just after the last cross section. The centerline determines the extents of the simulation and helps to establish the stationing of cross sections that are defined along its length. You should create centerlines from the upstream (dam side) end to the downstream end in order to provide the proper direction to the model.



Cross Sections

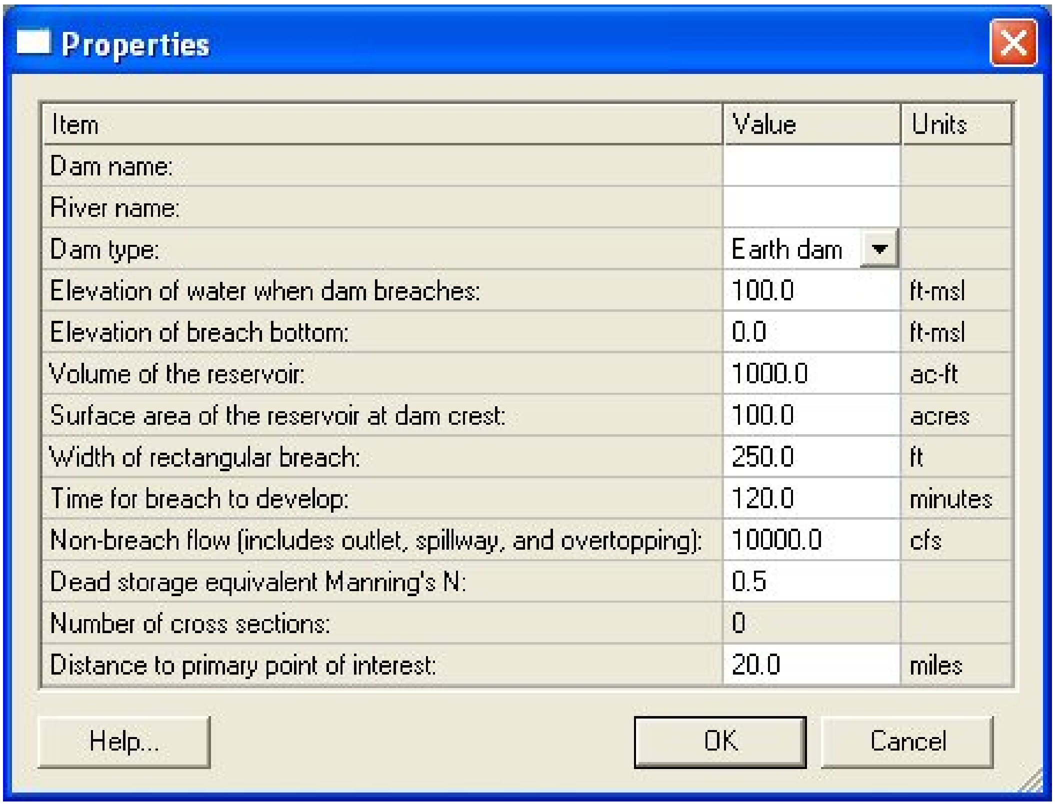
Cross sections for a simplified dam-break analysis must be defined in a 1D-Hydraulic Cross Section coverage. A minimum of two cross sections are required (upstream and downstream ends of the river to be modeled), but sufficient cross sections to define the floodplain should be included. If you are using a digital terrain model to extract cross sections then it is fairly simple to include additional cross sections without the requirement of a field survey.

SMPDBK uses a table of top-widths corresponding to incremental elevations to define the cross sections. WMS will compute 8 incremental depths automatically for each cross section since this is the maximum number of increments allowed by SMPDBK. The eight increments are equal and determined based on the low and high elevations of the cross section. Manning's roughness values can be determined for segments of the cross section based on an area property coverage.



SMPDBK Dam Properties

The *Dam Properties* dialog (from **Edit Parameters** of the *SMPDBK* menu) is used to set several options which describe the dam. This dialog is pictured below.



Data for this dialog can be collected by searching for the data on the Internet. Data could also be obtained from the National Inventory of Dams web site at http://geo.usace.army.mil/pgis/f?p=397:1:0 [1] (2004).

Running a Simulation

The SMPDBK executable is distributed with WMS and should be present in the WMS program directory (SMPDBK.EXE). When choosing the **Run Simulation** command you will be prompted for a file name that WMS will use to write the properly formatted input based on the centerline, cross section, roughness, and dam properties defined. After writing the input file WMS will launch the SMPDBK executable and pass the newly created input file as a command line argument.

Export Simulation

If you wish to save a completed SMPDBK input file that is properly formatted then you can choose the **Export SMPDBK File** command. You could then continue to edit/prepare the file in a text editing program such as Notepad or Wordpad and then run it through the SMPDBK program outside of WMS. However, an SMPDBK formatted input file is always generated when running a simulation from within WMS.

Related Topics

- Defining Roughness
- Post Processing
- Automated Floodplain Delineation

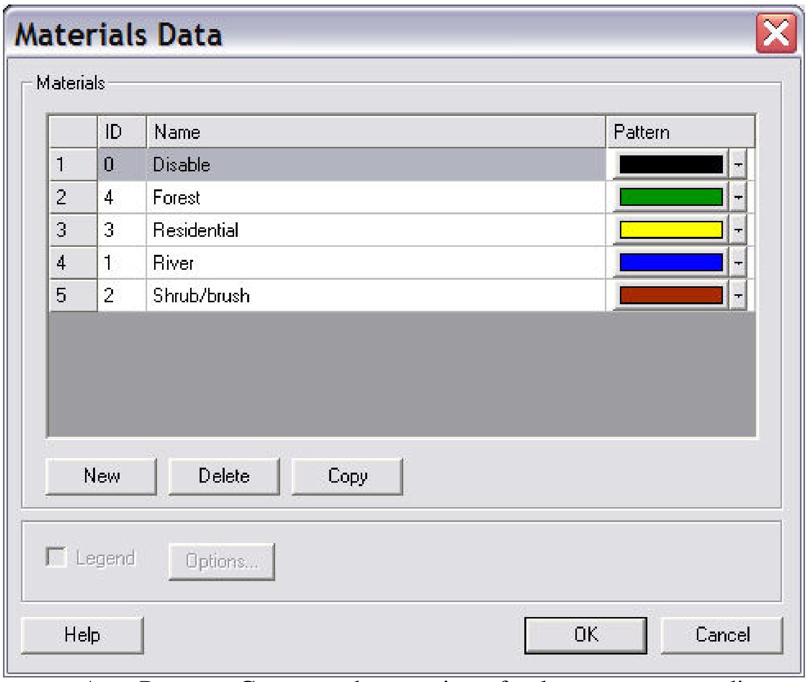
References

[1] http://geo.usace.army.mil/pgis/f?p=397:1:0

Defining Roughness

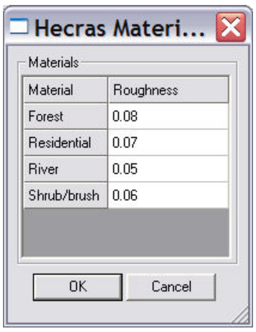
Roughness values are determined for cross sections based on a set of material properties defined for the cross sections. Material properties are defined in a four step process.

1. Create a master set of materials (land use) types that define the possible choices to be found in the floodplain you are modeling. Typically you will want to define as many materials as you will be defining manning's roughness values. This is done using the **Materials** command in the *SMPDBK* menu.

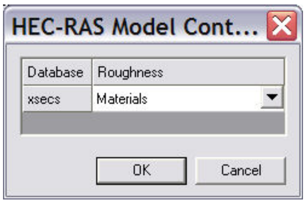


2. Create an Area Property Coverage that consists of polygons corresponding to the different land use (roughness) types found in the floodplain. After building the coverage you will need to define each polygon attribute by double-clicking on the polygon and assigning the appropriate roughness value. Generally these polygons will be a more generalized version of a land use map.

3. Define the manning's roughness values that correspond to each material property. This dialog is accessed from the *SMPDBK* menu using the **Material Properties** command.



4. Assign the created materials properties to the cross section database being used. This will make sure that properties mapped to the cross sections when cut from a digital terrain model have a roughness value assigned. This is done using the **Model Control** option in the *SMPDBK* menu.



When WMS writes the SMPDBK input file, average roughness values are computed based on the conveyance for the elevation/top-width increment.

Related Topics

- Simplified Dam-Break Analysis
- 1D-Hydraulic Centerline Coverage

SMPDBK Post Processing

To view results of an SMPDBK analysis, you should perform the following steps:

1. Run the simulation by selecting **Run Simulation** in the *SMPDBK* menu.

2. Using the Map Module, interpolate the water surface elevations in the 1D Hyd Centerline and 1D Hyd Cross Section coverages. This can be done by selecting each coverage to make it active and then selecting **Interpolate Water Surface Elevations** from the *River Tools* menu.

3. Go to the Terrain Data module and Delineate the floodplain.

4. In *Display Options*, turn on the contours. Turning on color filled contours with transparency and a background image (a contour map or an aerial photograph) will help you to better visualize the flood extents.

5. If there are areas you know are interpolated incorrectly and are not flooded (see Figure 1), you can draw a polygon around these regions so the floodplain will not be delineated in these areas. You can remove these areas from the floodplain delineation by creating a Flood Barrier coverage (see Figure 2 below) and then delineating the floodplain again using the User defined flood barrier coverage option (see Figure 3 below).

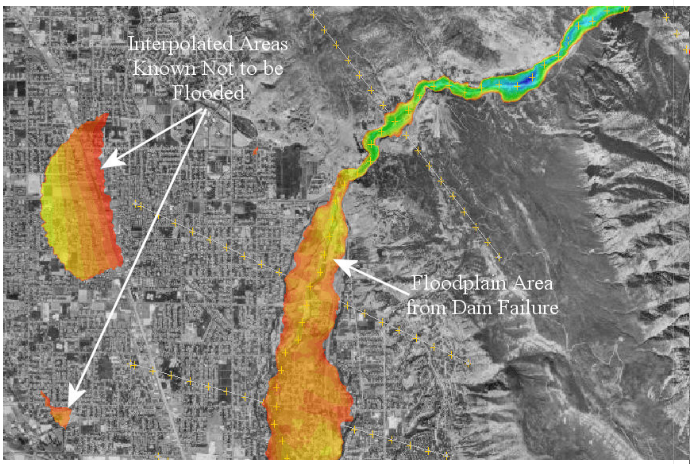


Figure 1

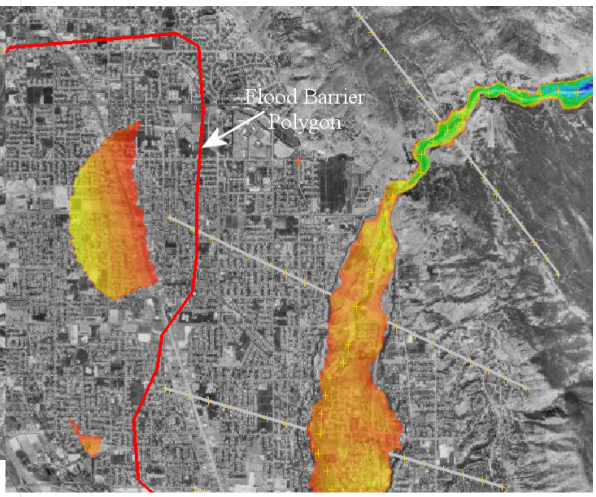


Figure 2

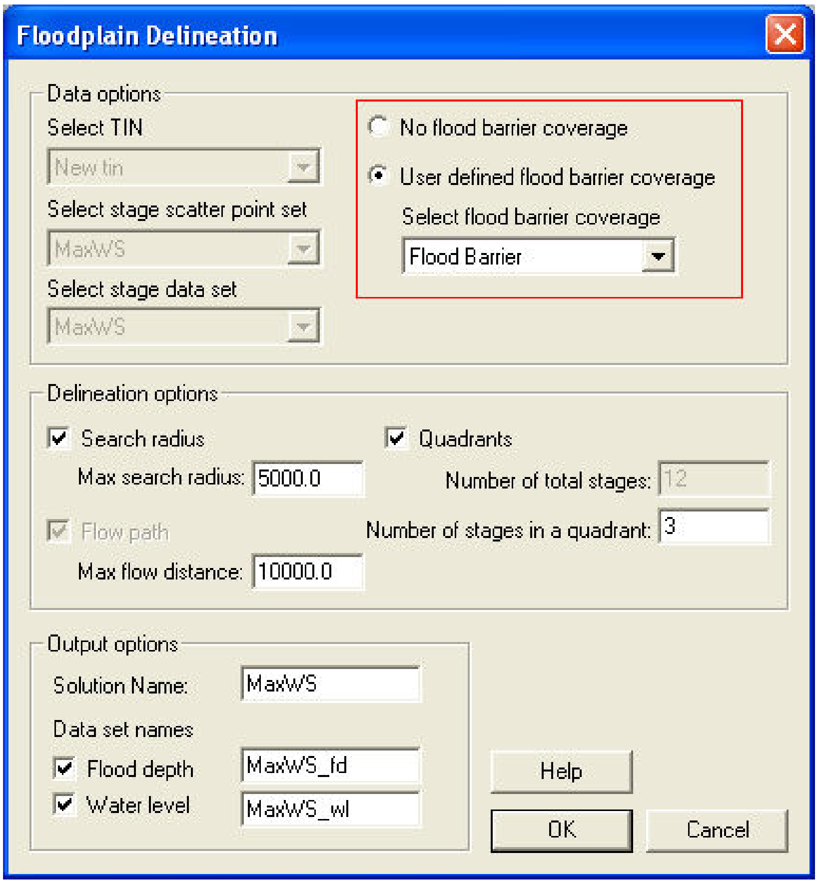


Figure 3

Related Topics

- Simplified Dam-Break Analysis
- Interpolate Results
- Flood Plain Delineation

6.14. TR-20

TR-20

TR20 models can be defined in WMS using the commands in the TR20 menu and resulting dialogs. There are six steps in defining a TR20 model using WMS:

Create a Topologic Tree

In the absence of terrain data, this is done using the commands to build the tree in the **Tree** menu. When using the TIN, Map, or DEM modules to delineate a watershed and sub-basin boundaries, a topologic tree is automatically created. Diversions can not be created directly on a TIN or in the Map module, but must be created using the commands in the *Tree* menu.

Define Job Control Parameters

Job control parameters are used to define the time and length of a simulation, output diagnostic controls, and other miscellaneous items. Definition of these parameters is discussed in detail later in this chapter.

Edit Basin/Outlet/Reservoir/Diversion Data

Parameters for all sub-basin outlets, reach routes, reservoirs, and diversions (hydrograph stations) are entered/edited using the Edit TR20 Parameters dialog accessed from the TR20 menu. This dialog displays current values and lets you edit values for the currently selected hydrograph station.

Model Check

This step allows you to check your TR20 data prior to performing an analysis. It reports errors such as zero area, undefined precipitation, etc. These errors should be corrected by returning to previous steps before actually trying to run TR20.

The **Model Check** command should be issued once you feel that all necessary TR20 data has been defined. It will report any possible errors/inconsistencies in your model so that corrections can be made prior to executing TR20. The list of checks made is not complete and just because no errors are reported does not ensure that a successful and/or accurate analysis will be completed. We encourage you to report any additional checks that might be made as you work through various problems.

Run TR-20

WMS creates TR20 files compatible with any version of TR20. However, a Windows (X-Windows on UNIX and MS-Windows on PC's) compiled version of TR20 is distributed with WMS so that it can be run without leaving the WMS environment.

For an in depth description of the TR20 runoff model itself, refer to the TR20 User's Manual[1].

The version of TR20 distributed with WMS can be run directly from WMS by using the **Run TR20** command in the *TR20* menu. Before running a TR20 simulation, you should run the model checker. The model checker will help you identify serious and potential problems. These problems should be corrected before a successful run of TR20 can be made.

The **Run TR20** command will bring up a dialog allowing you to specify 3 files which are necessary to run TR20. The first file is the TR20 input file. The second is an ASCII output file generated by TR20. This output file can be used to extract specific results. It also contains important information which can be used to correct problems encountered when running TR20. The third file will contain hydrograph results for basins and outlets. You can view these results by reading this file with the **Open** command from the *Hydrographs* menu.

Once these files have been defined and you select **OK**, TR20 will be executed. A separate window will appear and information about the TR20 simulation will be reported. If you are running with Microsoft Windows you must close this window when TR20 terminates. On UNIX workstations the window closes automatically.

If TR20 is not executed successfully when issuing this command then be sure that the path to your TR20.EXE file (tr20 for UNIX) is located in the same directory as your WMS executable file.

If TR20 does not run to a successful completion you can view the ASCII output file using the **View File** command in the *File* menu.

View Hydrographs

Once a TR20 simulation has been run you can view resulting hydrographs using the commands in the *Hydrographs* menu. After viewing the hydrographs, you may wish to repeat the previous steps in order to calibrate a model, or look at different scenarios.

Reading Existing Files

WMS is capable of reading TR20 files created manually using a text editor or other program. However, there are a couple of problems which need to be considered, and may have to be altered either before or after reading in one of these files.

- WMS reads analysis hydrograph results from the *.thy file. Many existing TR20 files will not specify output to this file and you may need to define it for all hydrograph stations before you will be able to read in analysis results. This can be done by selecting all basins/outlets and bringing up the respecting output control dialog. Saving the TR20 file and running it will then create the *.thy file.
- WMS will not read in TR20 files with more than one COMPUT record. Only one COMPUT record will be read in, and it will be the last COMPUT record in the file. If you need to make more than one computation, simply change the necessary input parameters in WMS, save the file, and run TR20 again.
- WMS will not read in data from ALTER, DELETE, INSERT, IPEAKS, PEAKS, or DURINC records.
- WMS adds basin, reservoir, outlet, reach, and diversion names in columns 73 through 78 of TR20 files. These names are not necessary to run TR20 or to read existing TR20 files into WMS.

If you run into any other problems reading TR20 files, please contact the distributor you purchased WMS from.

Saving Files

Once a tree has been created and all of the necessary data entered, a TR20 input file can be generated by selecting the **Save TR20 File** command the *TR20* menu. When writing the file, the proper order for computing, adding, and routing hydrographs is automatically determined. TR20 can be run without any further editing of the input file generated by WMS. Because WMS does not allow input for all TR20 options, it may be necessary to modify the file somewhat before execution. Hydrograph names must be defined in columns 73-80 of all RUNOFF, REACH, DIVERT, ADDHYD, and RESVOR, records and GRAPHICS must be defined in columns 61-68 of the JOB record before output of TR20 can be read back into WMS for display in the *Hydrograph* Window.

Existing files generated outside of WMS can be read it into WMS and a separate topological tree will automatically be generated for the watershed described in the file. Since WMS does not yet support all possible TR20 card types, there may be some incomplete information. However, the basic structure of the watershed will be created and all possible data will be retained. If multiple computations or runs of TR20 are specified in a single file, WMS will prompt you for which one you want to use. With WMS only a single computation sequence may be defined. If additional computations are desired, the parameters can easily be changed within WMS and another run of TR20 made.

Related Topics

- Editing TR20 Parameters

References

[1] http://www.deldot.gov/information/business/drc/pd_files/plan_development/win_tr20_user_guide.pdf

TR-20 Edit Parameters

Attributes or parameters for all TR20 hydrograph stations are defined and/or later edited using the *Edit TR20 Parameters* dialog. This dialog is accessed by selecting the **Edit TR20 Parameters** command from the *TR20* menu or when TR20 is the active model by double-clicking on basin, outlet, or diversion from the *Graphics Window*.

If a basin, outlet, or diversion is selected before issuing the command then data for that object is loaded for editing. This dialog lists the TR20 hydrograph station parameters which can be edited by selecting the corresponding button. When a hydrograph station is selected (basins/outlets/reservoirs/diversions) only the buttons which edit parameters associated with that hydrograph station are active, all others are dimmed.

Once the dialog appears it becomes part of the main screen until you select the **Done** button. Therefore, you can continue to select additional, or other hydrograph stations so that data for that object may be edited without exiting the dialog. You can use the **Previous** and **Next** Hydrograph Station buttons to cycle through hydrograph stations in the order they are computed by TR20. While the dialog is up, all menu commands are active.

Related Topics

- TR20 Overview
- Entering Job Control Parameters
- Entering Basin Data
- Entering Routing Data
- Entering Reservoir Data
- Entering Diversion Data
- Output Control

TR-20 Basin Data

General information for each basin is entered by selecting a basin and choosing the *Basin Data* dialog. If multiple basins are selected, some fields will not be able to be edited (e.g. Basin name and Area), while others will be "grayed out". These other fields are in Multi-select mode. If you select and assign data to these fields, this data is assigned to all the selected sub-basins.

Area

When a TIN, DEM, or Map-based model is present, basin areas and slopes can be computed automatically using the **Compute Basin Data** command in the *TIN* or *DEM* Drainage module menus, or in the *Feature Objects* Map module menu. If a TIN, DEM, or map data are not present, areas and slopes must be entered interactively using the topological tree as a map. Areas should be entered in square miles.

Curve Number

Curve Number refers to the SCS curve number for rainfall/ losses on snow-free ground.

- *NOTE*: Composite Curve Numbers can be computed automatically using a land use and hydrologic soil group coverages.

Name

Each hydrograph station should be identified with a unique name. This name is specific to TR-20 files created by WMS, and is used to identify the basin in the file so that resulting hydrographs from a model run can be read back into WMS and associated with the basin. The name can not be more than six characters long. By default WMS uses the basin ID number followed by a "B" for the name, but a descriptive name is generally more useful.

Time of Concentration

Time of concentration should be in hours for the unit hydrograph. Several different equations exist for determining the time of concentration. Two different methods exist for computing the time of concentration using computed basin data or using map data. The list of basin geometric attributes computed automatically when basins have been delineated from a TIN can be useful in many of these equations. These attributes can be viewed and edited from within the TR20 Basin data dialog by choosing the Basin Geometrical Attributes button.

Basin Geometric Attributes

When drainage data are computed, you can select this button to view and edit the geometric attributes of a basin, including the basin area, the flow length, and the basin slope. The different basin attributes can be viewed and edited only after computing the basin data. The basin data may be computed by selecting the **Compute Basin Data** command in the **TIN** or **DEM** Drainage module menus, or in the **Feature Objects** Map module menu.

Compute TC – Basin Data

Pressing this button will bring up the *Unit hydrograph parameter computation* dialog. In this dialog, you can define the type of computation method to use in computing the time of concentration for TR20. After exiting this dialog, the time of concentration, lag time, time to peak, and Snyder coefficient will be calculated. Many of these coefficients and times can be used in HEC-1 models, and the time of concentration can be used in TR20 watershed models. By de-selecting the Do not auto-recompute parameters option, unit hydrograph parameters will be automatically re-calculated when the basin area is re-computed or other basin parameters are changed.

Compute TC – Map Data

Times of concentration may also be computed using a time computation coverage. The *basin time of concentration* dialog is accessed by selecting the Compute TC – Map Data button.

Cross Sections

Select this option to use a defined cross section for your basin runoff computations. By selecting the **Define Cross Section** button, you can define a cross section to use in the runoff computations. To define a cross section, the elevation, discharge, and end area must be defined for different intervals.

Reservoirs

Select this option to use a defined reservoir for your basin runoff computations. By selecting the **Define Reservoir** button, you can define a reservoir to use in the runoff computations. To define a reservoir, the elevation, discharge, and storage must be defined for different intervals.

Related Topics

- TR-20 Overview
- Editing TR-20 Parameters

TR-20 Diversion Data

TR20 allows flow to be diverted from an outlet or drainage basin. This flow can be thought of as leaving the normal drainage system at that point. It can be retrieved at a downstream outlet where the diverted flow then contributes to the flow at that outlet. If no downstream retrieval outlet point is specified, the flow simply leaves the system at the diverted outlet point and never returns.

Diversion Name

The diversion name is used to associate resulting hydrographs with the appropriate diversion when reading a hydrograph file after a TR20 run. The nave should be unique and no longer than six characters.

Inflow Parameters

The inflow for the diversion can be determined in one of two ways: either by using a cross-section for the main channel or by using a structure for the main channel. Cross-sections are defined using the same dialog as for reaches as described in the section on channel routing. Structures are defined in the same way as structures for reservoir routing.

Outflow Parameters

The outflow for a diversion can be specified in one of two ways. In the first method, a constant flow is diverted. For this method, you must define what this constant outflow is. You can also define a cross section and the decimal fraction of the drainage area to be associated with the main channel output hydrograph (the drainage area fraction) here. In the second method of defining a diversion, you define the cross section for the diversion. TR20 uses a rating curve to divide flow between the two cross sections that comprise the diversion. You can define the decimal fraction of the drainage area to be associated with the main channel output hydrograph (the drainage area fraction) using this method as well.

Related Topics

- TR20 Overview
- Editing TR20 Parameters

TR-20 Reservoir Data

TR20 allows you to route a hydrograph through a reservoir using the RESVOR TR20 file card. You can define these reservoir routing parameters in the *TR20 Reservoir Routing* dialog. Reservoirs and other types of structures (such as detention basins) can be defined from the *TR20 Reservoir Data* dialog.

You can define reservoir data by pressing the **Define reservoir data** button. This will bring up a dialog where you can define the elevation-discharge-storage relationship for the reservoir. You can enter up to 20 elevation-discharge-storage relationships for each reservoir. To assign a particular set of data to a reservoir, select the reservoir you want and select the **OK** button on the *TR20 Reservoir data* dialog.

Reservoir Name

The reservoir name is used to associate resulting hydrographs with the appropriate reservoir when reading a hydrograph file after a TR20 run. The name should be unique and no longer than eight characters.

Routing

Routing can be toggled on by selecting the **Define Reservoir Routing** button. After selecting the **Define Reservoir Routing** button, reservoir routing will be defined at the selected tree nodes. If this option is not selected, routing will not be defined at the selected tree node.

Start Routing Elevation

The start routing elevation is the water surface elevation, in feet, that routing begins for the reservoir or structure.

Input Hydrograph

One or more input hydrographs can be defined for a reservoir. WMS will combine any input hydrographs with other input hydrographs to the reservoir. The resulting hydrograph will then be routed through the reservoir. This option is useful for defining measured stream flows into a reservoir.

Related Topics

- TR20 Overview
- Editing TR20 Parameters

TR-20 Routing Data

Outlet points are used to define locations where hydrographs are combined and then routed downstream. The appropriate combined hydrograph (ADDHYD records) stations are generated automatically when writing a TR20 file. However, routing data must be entered in order to simulate the movement of a flood wave through the river reaches or reservoirs. The effects of storage and flow resistance are accounted for in the shape and timing of the flood wave.

Routing data is entered by selecting an outlet and then selecting the **Routing Data** button from the *Edit TR20 Parameters* dialog.

Routing Method

TR20 has two different routing methods to choose from. If cross-sectional data is available it can be used to establish routing parameters using a “m-value” method. If the cross-section data are not available a Kinematic wave method may be used instead. The method for each outlet is determined by the radio group selection.

Defined Cross Sections

With this routing method a typical cross-section for each reach (outlet) must be defined. This is done using the *TR20 Cross Section Data* dialog, and is accessed by selecting the **Define Cross Section Data** button.

Bankfull Elevation

If you enter the bankfull elevation here, it will trigger a warning message in the TR20 output file if less than two cross section data points are below bankfull. Entering the bankfull elevation is optional.

Zero Damage Elevation

This is used with the TR20 flow duration analysis to flag results at this elevation. Entering the zero damage elevation is optional.

Low Ground Elevation

The low ground elevation is the lowest flood plain elevation in a cross-section. The low ground elevation, which is optional, must not be higher than the bankfull elevation.

Flow Units

The flow can be entered in cubic feet per second per square mile (csm) or cubic feet per second (cfs). To select one of these flows, simply select the type of flow you want by selecting the appropriate radio button in the *Define Cross Section* dialog. If flow is defined in csm, you must define the drainage area of the basin in square miles.

Defining Cross Sections

Once a cross section has been defined in TR20, this cross section can be used in other locations in the TR20 model. A cross-section defined for an earlier outlet can be used later by another outlet simply by selecting the name from the text window or the drop-down box in the *TR20 Routing Data* dialog.

Kinematic Wave

The kinematic wave method uses two different rating coefficients, x and m, in the following equation:

$$Q = xA^m$$

where $Q$is the discharge, $A$is the valley storage area divided by length and $x$and $m$are the coefficient and exponent of the relationship describing the reach and maximum inflow hydrograph peak discharge.

Reach Length

This value should be the length, in feet, along the river reach for which routing takes place. If a TIN, DEM, or Map-based watershed model is present this value is automatically computed and assigned to this field whenever the **Compute Basin Data** command is executed. A cross section must be assigned to the reach length assigned.

Outlet Names

Since outlets are used for both types (adding and routing) of hydrograph stations in the TR20 input file, a separate name for each type of hydrograph must be entered. The name should be eight characters or less and is used to read hydrographs from the results file.

Direct Input Hydrographs

Hydrographs can be input directly, and then routed down stream, using the different routing options. To do this, select the *Direct Input Hydrograph* option. Hit the **New** button in the *Define input hydrographs* dialog to define a new hydrograph. The base flow and contributing drainage area for the hydrograph should be entered for computation purposes.

The hydrograph may be edited by selecting the **Edit** button after selecting the hydrograph to edit. For each input hydrograph in TR20, the discharge in cfs at each time interval must be entered.

Related Topics

- TR20 Overview
- Editing TR20 Parameters
- Entering Basin Data

TR-20 Job Control Parameters

The Job Control parameters include all of the data necessary to run a TR-20 analysis that are not a part of a basin, reach, or reservoir. The following are the specific parameters defined as part of job control:

Precipitation

In TR20, there are two different ways to define precipitation. Precipitation is defined by selecting the **Job Control** command from the *TR20* menu. Then, select the **Define Precipitation** button to define precipitation over the entire watershed. You can define precipitation by using either a standard SCS rainfall distribution or by defining a custom rainfall distribution. If a custom rainfall distribution is used, you must define a custom rainfall distribution in the *XY Series Editor*. If one of the standard SCS rainfall distributions are used, you can select one of the following rainfall types:

- Emergency Spillway and Freeboard
- Type I: 24 hours
- Type II: 24 hours
- Type IA: 24 hours
- Type III: 24 hours
- Type II: 48 hours

For all the standard distributions, you must define the rainfall depth. For the dimensionless distribution, you must also define the rainfall duration. You only need to define the depth and/or duration of the rainfall for a custom distribution if one of the units entered in the *XY Series Editor* is dimensionless.

Time Control

Main Time Increment

The computational time increment defines the length in time between hydrograph ordinates. The interval should be specified in hours.

Starting Time

The starting time of the simulation is defined in this entry. The time should be specified as floating point number of hours past midnight. For example 7:45 am would be entered as 7.75 and 1:20 p.m. as 13.33.

Title

Enter a name and/or project description identifying the model. Two different title records up to 72 characters each can be entered. The title records will appear at the top of the TR20 input file.

Unit Hydrograph Definition

TR20 offers two ways of defining the unit hydrograph used to compute runoff from your watershed. The first way is to use the standard SCS hydrograph provided with TR20. Another way is to define your own input hydrograph. This can be done by selecting the **Define another dimensionless unit hydrograph** radio button and defining the unit hydrograph in the time series editor.

Base Flow

Base flow can be entered in TR20 from the *Job Control* dialog by selecting the **Define Base Flow** button. There are two ways for defining base flow in TR20. One way is to define a constant base flow. The second way is to define a triangular base flow in the watershed by specifying a volume in inches, a peak time in hours, and a base time in hours. By selecting the appropriate option, you can define either a triangular or a constant base flow for your watershed.

Related Topics

- TR-20 Analysis
- Editing TR20 Parameters
- Obtaining Precipitation Data
- Basin Data
- Reach Data
- Reservoir Data
- Diversion Data

TR-20 Output Control

For each hydrograph station (basin hydrographs, combined hydrographs, routed hydrographs, reservoir hydrographs, and diversion hydrographs) different output controls can be specified. Selecting the Output Control dialog button from the individual hydrograph station dialog accesses this dialog. Entries that can be defined in this dialog are described below.

Peak Discharge and Runoff Volume

Selecting this option will output the following data at the selected hydrograph station(s):

- Peak discharge in cfs.
- Peak time in hours.
- Volume of water above constant base flow under the hydrograph in inches depth, acre-feet, and cfs-hours.
- Constant base flow value.

Discharge Hydrograph

Selecting this option will produce a discharge versus time hydrograph at the selected hydrograph station(s).

Elevation of Hydrograph

Selecting this option will produce an elevation versus time hydrograph in tabular form. This table will be produced for the hydrograph station(s) selected.

Flow Duration Data

Selecting this option will produce a table showing discharge versus duration of time that a discharge is equaled or exceeded. This table will be produced for the hydrograph station(s) selected.

Save Results for Summary Tables

This option puts the standard control operation results in summary tables 1 and 3 of the ECON2/URB1 generated files (see TR-20 reference manual). Only the largest peak discharge for each hydrograph is listed in these files.

Individual Basins and Reaches

Control of the output files can be specified individually for each of the reaches or outlets. The output control parameters for the inflow hydrograph corresponds to the ADDHYD record whereas the parameters for the outflow hydrograph correspond to the REACH record. Output control parameters for all hydrograph stations are identical and are discussed in an earlier section.

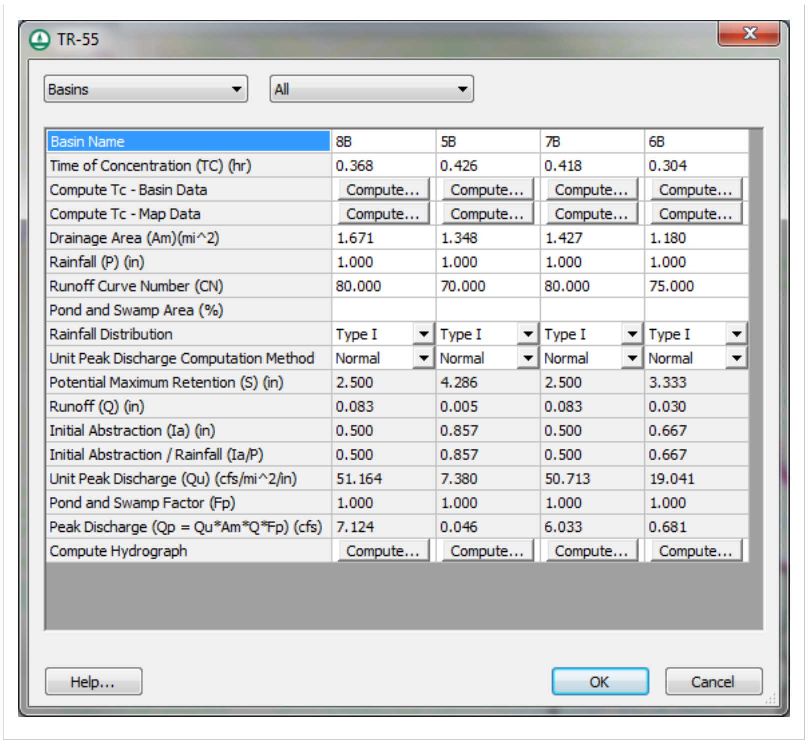
Related Topics

- TR-20 Overview
- Editing TR-20 Parameters

6.15. TR-55

TR-55

TR-55 models can be defined in WMS using the commands in the *TR-55* menu and resulting dialogs. The TR-55 model has long been used to determine the increase in runoff resulting from the development of rural land into urban land. The WMS interface to TR-55, combined with the basin delineation and time of concentration calculations from feature objects (see Lag Time and Time of Concentration), makes it simple to set up and run several different scenarios in a relatively short time period.



This section describes the mechanics of the WMS interface to TR-55 and is not a complete reference to the methodology, uses, and limitations. For more in depth information you should read “Technical Release 55: Urban Hydrology for Small Watersheds, 2nd Edition.” [1]

WMS uses its own format for saving or reading a TR-55 simulation. When you are in the Hydrologic Modeling Module, select *TR-55* from the drop-down list of models to make TR-55 your active model. To edit the TR-55 parameters and run a TR-55 simulation, select the outlets or basins you wish to edit and/or run and select the **Run Simulation** menu item from the *TR-55* menu.

Related Topics

- Basin Data
- Outlet Data
- Hydrographs

References

[1] ftp://ftp.wcc.nrcs.usda.gov/wntsc/H&H/other/TR55documentation.pdf

TR-55 Basin Data

The following information must be determined and appropriate values entered for each sub-basin.

Drainage Basin Area

Enter the area of the basin in this edit field. If you have delineated a watershed from feature objects, a DEM, or a TIN, the drainage area can be computed and will automatically be updated in this field.

Time of Concentration

Three different options exist for determining the basin time of concentration (tc).

1. The time of concentration can be computed outside of WMS and entered into the appropriate edit field.
2. The Compute Tc – Basin Data button can be chosen and one of the time of concentration (or lag time) equations specified (this option is only available when you have computed basin data from either a TIN or a DEM).
3. Finally, a series of time computation arcs may be used to define overland, sheet, and channel flow with in a basin and then travel times for each arc are summed to compute the total travel time or time of concentration for the basin. The Compute Tc – Map Data button accesses the dialog which allows you to combine arcs within the currently selected basin to compute a time of concentration. A description of the time computation coverage can help in understanding how to do this. See the Overview of Map Data Equations help page for a detailed description of the TR-55 travel time equations that can be assigned to arcs in a time computation coverage.

SCS (NRCS) Curve Number

The NRCS (SCS) Curve Number (CN) should be entered in this field. CN is a function of hydrologic soil group and land use. Composite CN values may be computed for each basin and mapped to this field. For more information on computing CNs from GIS data layers.

Rainfall

24-hour rainfall depth and an appropriate time distribution curve

Both the total rainfall and a dimensionless mass distribution must be specified to define precipitation for a basin. The total rainfall is entered in the rainfall edit field, and the distribution type is specified by selecting one of the standard NRCS (SCS) rainfall distribution types from the drop down list. The TR-55 reference manual shows a map (Appendix B-2 of the TR-55 reference[1]) of distribution types for the US.

Pond/Swamp Factor

Optionally, the effects of water bodies can be determined by entering the percent of the watershed covered by lakes/ponds.

The pond and swamp factor will affect the peak discharge. The percentage of area for the selected basin that is made up of ponds, lakes, swamps, etc. should be entered in the appropriate edit field.

Related Topics

- TR-55 Overview
- Outlet Data
- Hydrographs

- Overview of Map Data Equations

TR-55 Outlet Data

WMS allows you to define your watershed as a single basin or you may also subdivide the watershed into multiple sub-basins. Besides computing peak flows using the standard TR-55 equations, storm hydrographs may also be computed for each basin and then combined/lagged to downstream outlets (junctions) using the TR-55 tabular hydrograph method.

When you create a watershed with multiple sub-basins you must define the time of travel from one outlet (junction) to the next. The travel time for an outlet is defined by selecting the outlet and then entering the appropriate travel time. You may wish to use a time computation arc to compute the travel time.

Related Topics

- Channel Travel Time
- TR-55 Overview
- Basin Data
- Hydrographs

TR-55 Hydrographs

Using the TR-55 tabular hydrograph method (see TR-55 reference manual for details), hydrographs for a selected basin or outlet may be computed. The **Compute Hydrograph(s)** button is chosen once all of the necessary input for basins and outlets have been entered and a new set of hydrographs will be computed.

If you have a basin selected when you compute hydrographs, then a hydrograph for only the selected basin will be computed. However, if you have an outlet selected when you compute hydrographs, then a new hydrograph for each upstream basin will be computed along with the routed hydrographs at outlet points.

You may notice a slight difference between hydrographs computed by WMS-TR-55 implementation and those in the TR-55 reference manual. This difference occurs because the I_a/Pvalue is rounded to the nearest hydrograph value for the standard version of TR-55. In WMS hydrographs are determined by linear interpolation between two hydrographs using the actual I_a/Pvalue.

Information about discrepancies between the TR-55 Hydrograph peak flow values and the Unit Peak Discharge method peak flow values

One question that is frequently asked about TR-55 hydrographs is why the peak flow computed from the TR-55 unit peak discharge method is different from the peak flow on the TR-55 computed hydrograph. Though the TR-55 tabular hydrograph method, which is used to compute TR-55 hydrographs, may give similar results to the TR-55 unit peak discharge method, which is used to compute a peak flow, the peak flows from these two different methods will usually be different. WMS uses the method described in Exhibit 5 in the TR-55 manual to compute a hydrograph using the tabular hydrograph method. Hydrograph ordinates used in the the TR-55 tabular hydrograph method were originally obtained from a series of TR-20 runs with different times of concentration and other values, as described in the TR-55 manual. See the TR-55 manual [1] for more information.

Additional Information

WMS uses exhibit 5 in the TR-55 manual for the tabular hydrograph method.

Values for the T_c, T_t, I_a/P, and $A_m * Q$are used as input values for this table, and all hydrograph values are linearly interpolated or extrapolated from the values for T_c, T_t, and I_a/P. The difference between the WMS method and the method described in the manual is the WMS hydrograph is linearly interpolated. The method described in the manual just rounds the values for T_c, T_t, and I_a/P. So you should basically get the same results, but the WMS results will be more accurate and can go beyond the minimum and maximum values for T_c, T_t, and I_a/P(this is not recommended).

Related Topics

- TR-55 Overview

TR-55 Channel Travel Time

The TR-55 travel time is the amount of time a flood wave takes to move from the outlet to the downstream most point of the watershed. This is different than other models in WMS which has you enter the travel time from outlet to outlet. Computed hydrographs at each basin and each outlet point represent the amount of the hydrograph at the outlet point contributed from each basin or each outlet point. Thus, the hydrographs you see when you run TR-55 in WMS are already lagged. The upstream hydrographs are then simply added together to determine the hydrograph at the watershed outlet point. No lagging is done in WMS; tabulated TR-55 hydrographs are pre-lagged.

Like time of concentration, travel times between outlets may be computed using a series of feature objects with equations such as Manning's defined for each.

Related Topics

- TR55 Overview
- Entering TR55 Parameters
- Outlet Data

TR-55 Edit Parameters

Once all of the input necessary to run a TR-55 simulation is defined, peak flow and hydrograph computations are made using the same dialog. The first time you select the *TR-55 data* window lists the remaining information that must be entered before a peak flow is calculated. Once all of the items listed in this window are entered, WMS performs the calculation for peak discharge and displays the result in the same data window. Additional information (help) about any line selected in the *TR-55 data* window is displayed in the *TR-55 help* window.

The **Data window display options** button allows you to control which information are displayed in the data window when the peak discharge is computed. The primary purpose for including the different display options is so that all results can be copied to the clipboard and then pasted into a report document.

Name

The basin name is used to identify individual basins within a larger watershed. As with all basins created automatically in WMS, the name is defaulted to “ID”B where ID is an internal identification number and B stands for basin. The name can be changed to something more identifiable at any time. While TR-55 does not require that basin names are unique, other models supported by WMS do.

Related Topics

- TR55 Overview
- Drainage Area
- Curve Number
- Time of Concentration
- Channel Travel Time
- Rainfall
- Pond/Swamp Factor

6.15.a. Map Data Equations

Overview of Map Data Equations

The TR-55 equations for travel time are one set of commonly used equations to compute time of concentration. Others, including those used by the Federal Highways Administration, are variations of the same type of equations. These sets of equations form a library of predefined equations in WMS. However, you can also enter your own equation or modify one of the existing equations.

Equations consist of a type and sub-type (user-defined equations do not require a sub-type). Both are specified using the drop-down combo boxes in the attribute dialog.

Equations

FHWA:

Sheet Flow Equation

Shallow Concentrated Flow

Channel Flow

TR55:

Sheet Flow Equation

Shallow Concentrated Flow Equation

Channel Flow Equation

Maricopa County

User Defined

Related Topics

- Travel Times from Map Data
- Assigning Equation to Arcs
- TR-55

Travel Times from Map Data

The Time Computation coverage can be used to create arcs representing flow path segments when computing time of concentration or lag time for a basin or reach. Within a basin the time of concentration or lag time is usually determined by combining the time of travel across one or more flow path segments. Travel time equations are generally functions of the length and slope of the flow path segment as well as surface roughness (i.e. Manning's roughness coefficient) and channel shape and roughness. Since length and slope (providing there is a background TIN or DEM) are easily determined from arcs, the time computation coverage provides a simple and powerful method for computing basin time of concentration and/or lag time. Travel times between consecutive outlet points may also be computed using the same tools.

If you have developed your watershed model from a TIN or DEM you can have WMS automatically create flow path arcs from selected points using the Node→Flow Arcs and Stream→Flow Arcs commands.

The process of computing a time of concentration involves two primary steps.

1. Compute travel times for individual arc segments.

2. Combine the travel times of all arcs within a basin to compute the time of concentration for the basin, or combine arcs between outlet points to compute the travel time along a stream reach.

Related Topics

- Lag Time and Time of Concentration
- Travel Times from Basin Data
- Combining Arc Travel Times
- Overview of Map Data Equations

User Defined Time of Concentration

Because it would be impossible to contain all possible equations used for computing travel times, a user defined equation may be defined for any arc segment. When “user defined” is selected for the arc type the **Modify Equation** button in the *Time Computation Attributes* dialog is active. By selecting this button you can use the *Modify Equation* dialog to create/modify a suitable equation. User defined equations can be created by typing in the equation edit field using the following rules for precedence (order of operations):

1. Parts of the equation in parentheses have the highest precedence.
2. Multiplication and division have higher precedence than addition and subtraction.
3. Equations are evaluated from left to right.

Besides typing in the equation you can also select one of the pre-defined equations from the Sample equations drop-down list and add it to the equation line as a starting point to create a new equation. You may also use any of the variables that WMS can manage as part of the equation (e.g. length, slope, rainfall intensity, etc.). However, you cannot add a new "variable" since there is no way for WMS to manage it. If a variable that is not managed by WMS is used in your equation you must determine what the appropriate value for the selected arc would be and enter it as a constant in the user defined equation. Variables are added to an equation by either typing the abbreviation or selecting the variable you want to use and clicking on the **Add to Equation** button. In a similar fashion, mathematical operators can either be typed or the corresponding button selected to add them to the equation.

Related Topics

- Overview of Map Data Equations

FHWA Sheet Flow Equation

Sheet flow generally occurs for the first 300 feet at the headwater of streams. The following equation is used to describe sheet flow:

$$T_t = \left(\frac{K}{i^{0.4}}\right)\left(\frac{nL}{\sqrt{S}}\right)^{0.6}$$

where:

T_t= travel time for open channel flow segments.

K= emprical coefficient equal to 0.933 for English units and 6.943 for Metric.

i= rainfall intensity (in/hr).

n= Mannings roughness coefficient for overland flow. Suggested values are given in Table 3-2 of the FHWA HEC 22 manual and : arerepeated in the table below.

L= length of the overland flow segment (ft).

S= ground slope of the flow segment (ft/ft).

Manning's Roughness for overland sheet flow

Surface Description	n
Smooth asphalt	0.011
Smooth concrete	0.012
Ordinary concrete lining	0.013
Good wood	0.014
Brick with cement mortar	0.014
Vitrified clay	0.015
Cast iron	0.015
Corrugated metal pipe	0.024
Cement rubble surface	0.024
Fallow (no residue)	0.05
Cultivated soils	
Residue cover £ 20%	0.06
Residue cover > 20%	0.17
Range (natural)	0.13
Grass	
Short prairie grass	0.15
Dense grasses	0.24
Bermuda grass	0.41
Woods	
Light underbrush	0.40
Dense underbrush	0.80

The rainfall intensity is actually a function of the travel time for the flow segment. In order to iteratively solve for the travel time you must define an IDF curve (function) to be used in conjunction with the equation. IDF curves are defined in WMS using Hydro 35, NOAA Atlas 2, or user defined rainfall intensities for specific durations. The IDF Curves dialog used as part of the rational method is used to set up equations relating $i$to T_t.

Related Topics

- Overview of Map Data Equations

FHWA Shallow Concentrated Flow

After 300 feet, sheet flow usually turns to shallow concentrated flow. The following equation, based entirely on the length and slope of the arc, is used to compute the travel time for the shallow concentrated segment of flow:

$$T_t = \frac{L}{60k\sqrt{S}}$$

where:

T_t= travel time for open channel flow segments.

L= Length of flow segment.

k= intercept coefficient (values are given in Table 3-3 of the FHWA HEC 22 manual and are repeated in the table below).

S= slope of the ground surface as a percent.

Intercept coefficients for velocity vs slope relationships

Land Cover / Flow Regime	k
Forest with heavy ground litter; hay meadow	0.076
Trash fallow or minimum tillage cultivation; contour or strip cropped; woodland	0.152
Short grass pasture	0.213
Cultivated straight row	0.274
Nearly bare and untilled; alluvial fans in western mountain regions	0.305
Grassed waterways	0.457
Unpaved	0.491
Paved area; small upland gullies	0.619

Related Topics

- Overview of Map Data Equations

FHWA Channel Flow

Travel time for open channel flow segments is computed using the following form of Manning's equation for open channel flow:

$$T_t = \frac{Ln}{60KR^{\frac{2}{3}}\sqrt{S}}$$

where:

T_t= travel time for open channel flow segments.

L= open channel flow length.

n= Manning's roughness coefficient for channel flow. Suggested values are given in Table 3-4 of the FHWA HEC 22 manual and are repeated in the table below.

K= empirical coefficient equal to 1.49 for English units and 1.0 for Metric.

R= hydraulic radius (length, ft or m).

S= channel slope (length/length).

Values of Manning's coefficient for channels and pipes

Conduit Material	n
Closed conduits	
Asbestos-cement pipe	0.011-0.015
Brick	0.013-0.017
Cast iron pipe	
Cement lined & seal coated	0.011-0.015
Concrete (monolithic)	0.012-0.014
Concrete pipe	0.011-0.015
Corrugated-metal pipe (0.5-2.5 inch corrugations)	
Plain	0.022-0.026
Paved invert	0.018-0.022
Spun asphalt lined	0.011-0.015
Plastic pipe (smooth)	0.011-0.015
Vitrified clay	
Pipes	0.011-0.015
Liner plates	0.013-0.017
Open channels	
Lined channels	
Asphalt	0.013-0.017
Brick	0.012-0.018
Concrete	0.011-0.020
Rubble or riprap	0.020-0.035
Vegetal	0.030-0.040
Excavated or dredged	

Earth, straight and uniform	0.020-0.030
Earth, winding, fairly uniform	0.025-0.040
Rock	0.030-0.045
Unmaintained	0.050-0.14
Natural channels (minor streams, top width at flood stage < 100 feet)	
Fairly regular section	0.03-0.07
Irregular section with pools	0.04-0.10

The hydraulic radius may be computed using the channel calculator. In this case you will have to assume an approximate depth of flow or flow rate in order for the channel calculator to be able to compute the appropriate hydraulic radius.

Related Topics

- Overview of Map Data Equations

TR-55 Sheet Flow Equation

Sheet flow generally occurs in the headwater of streams. The following equation is used to describe sheet flow:

$$T_t = \frac{0.007(nL)^{0.8}}{(P_2)^{0.5} s^{0.4}}$$

where:

T_t= travel time (hr).

n= Manning's roughness coefficient (see Table 3-1 of the TR-55 manual [1]).

L= flow path length (ft).

P_2= 2year, 24 hour rainfall (in).

s= slope of the hydraulic grade line (ground slope) in ft/ft.

2 year, 24 hour rainfall values can be determined from the NOAA Atlas 2 or NOAA Atlas 14 maps for many US locations or from the map in appendix B-4 of the TR-55 reference manual [1] for eastern US locations.

Generally the sheet flow equation should not be used for lengths greater than 300 feet. This simplified form of the Manning's kinematic solution is based on the following:

1. Shallow steady uniform flow,
2. Constant intensity of rainfall excess (that part of a rain available for runoff),
3. Rainfall duration of 24 hours, and
4. Minor effect of infiltration on travel time.

Related Topics

- Overview of Map Data Equations

TR-55 Shallow Concentrated Flow Equation

After a maximum of 300 feet, sheet flow usually becomes shallow concentrated flow. The average velocity for this flow can be determined using the equations below, in which average velocity (V) is a function of watercourse slope (s) and type of channel:

For Unpaved Channels: $V(ft/s) = 16.1345(s)_{0.5}$

For Paved Channels: $V(ft/s) = 20.3282(s)_{0.5}$

Tillage can affect the direction of shallow concentrated flow. Flow may not always be directly down the watershed slope if tillage runs across the slope. After determining average velocity, WMS uses the following equation to estimate travel time for the shallow concentrated flow segment:

$$T_t = \frac{L}{3600V}$$

where:

T_t= travel time (hr).

L= flow path length (ft).

V= average velocity (ft/s).

3600= conversion factor from seconds to hours.

Related Topics

- Overview of Map Data Equations

TR-55 Channel Flow Equation

Open channels are assumed to begin where surveyed cross section information has been obtained, where channels are visible on aerial photographs, or where blue lines (indicating streams) appear on United States Geological Survey (USGS) quadrangle sheets. Manning’s equation or water surface profile information can be used to estimate average flow velocity. Average flow velocity is usually determined for bankfull elevation.

Manning's Equation is:

$$V = \frac{1.49r^{\frac{2}{3}}s^{\frac{1}{2}}}{n}$$

where:

V= average velocity (ft/s).

r= hydraulic radius (ft) and is equal to a/p_w, where:

a= cross sectional flow area (ft^2)

p_w= wetted perimeter (ft)

s= slope of the hydraulic grade line (channel slope, ft/ft)

n= Manning’s roughness coefficient for open channel flow.

Manning’s n values for open channel flow can be obtained from standard textbooks such as Chow (1959) or Linsley et al. (1982). The FHWA Channel Flow page also lists Manning's n values for various channel and pipe materials.

The hydraulic radius may be computed using the channel calculator. In this case you will have to assume an approximate depth of flow or flow rate in order for the channel calculator to be able to compute the appropriate hydraulic radius.

After average velocity is computed using Manning's equation, the travel time T_t for the channel segment can be estimated using the following equation:

$$T_t = \frac{L}{3600V}$$

where:

T_t= travel time (hr).

L= flow path length (ft).

V= average velocity (ft/s).

3600= conversion factor from seconds to hours.

WMS combines Manning's equation and the travel time equation into a single equation to compute the travel time using the TR-55 open channel equation.

Related Topics

- Overview of Map Data Equations

Maricopa County Time of Concentration

The Maricopa County, Arizona equation for computing time of concentration has also been included in WMS as a pre-defined equation.

$$tc = 11.4\sqrt{L}K_b^{0.52}S^{-0.31}i^{-0.38}$$

where:

tc= time of concentration.

L= length of the flow path (ft).

K_b= representative watershed resistance coefficients. Values are computed using the equation $K_b = m\log(A) + b$where $m$and $b$are defined in the Table below of the Maricopa County drainage manual (shown below) and $A$is the drainage area in acres.

S= ground slope of the flow segment (ft/ft).

i= rainfall intensity (in/hr).

Parameters for Estimating

Type	Description	Typical Applications	Parameters m	Parameters b
A	Minimal roughness: Relatively smooth and/or well graded and uniform land surfaces. Surface runoff is sheet flow.	Commercial/Industrial areas, Residential area, Parks and golf courses	-0.00625	0.04
B	Moderately low roughness: Land surfaces have irregularly spaced roughness elements that protrude from the surface but the overall character of the surface is relatively uniform. Surface runoff is predominately sheet flow around the roughness elements.	Agricultural fields Pastures Desert rangelands Undeveloped urban land	-0.01375	0.08
C	Moderately high roughness: Land surfaces that have significant large to medium sized roughness elements and/or poorly graded land surfaces that cause flow to be diverted around roughness elements. Surface runoff is sheet flow for short distances draining into meandering drainage paths.	Hillslopes Brushy alluvial fans Hilly rangelands Disturbed land, mining, etc. Forests with underbrush	-0.025	0.15

D	Maximum roughness: Rough land surfaces with torturous flow paths. Surface runoff is concentrated in numerous short flow paths that are oblique to the main flow direction	Mountains Some wetlands	-0.030	0.20

Conversion of watershed area between square miles and acres may be required in order for this equation to compute the proper time of concentration.

Because the appropriate rainfall intensity value is a function of t_c, you will need to define an IDF curve (equation). The IDF Curves dialog used as part of the rational method is used to set up equations relating i to t_c.

Related Topics

- Overview of Map Data Equations

7. Appendix

What's new in WMS version 8.1

The WMS software development team is excited about the release of WMS 8.1! This page lists the exciting new features that have been added to WMS 8.1.

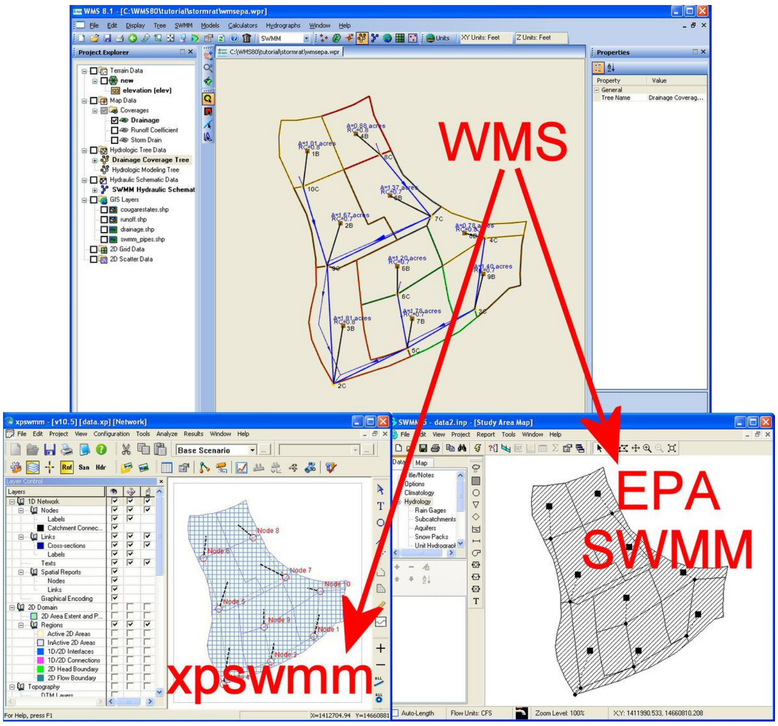
Model Enhancements

Hydrologic Modeling

Storm water quality and quantity modeling with xpswmm and EPA SWMM interfaces

The Storm Water Management Model (SWMM) is a popular model used to simulate the hydrology and hydraulics of storm water runoff. SWMM is primarily used for urban areas, and can be used to model single event or long-term (continuous) simulation of runoff quantity and quality.

With WMS 8.1, you can delineate a watershed and then export the delineated watershed boundaries and the WMS-computed watershed data to xpswmm [1] or EPA SWMM [2].



SWMM Interface Example

GSSHA™ Overland and Stream Contour Animation

=420|bgcolor=white|scale=noborder|align=right</flash>

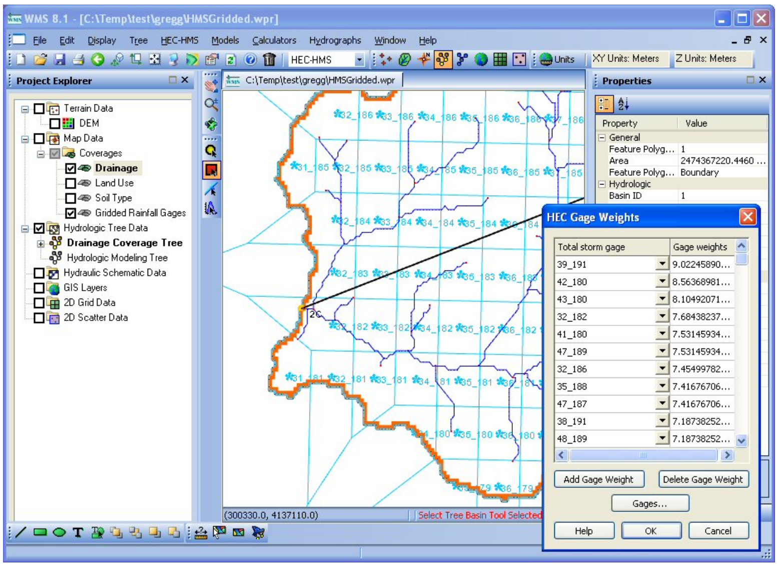
Spatially Distributed Hydrologic Modeling with GSSHA™

New to WMS 8.1 is full support and documentation for the distributed hydrologic model GSSHA™. Several improvements have been made to the GSSHA™ interface in WMS 8.1. The entire process of creating GSSHA™ models and running GSSHA™ models has been streamlined in the Hydrologic Modeling Wizard. The water quality modeling interface and capabilities of GSSHA™ have been significantly improved. Nutrient modeling capabilities have also been improved. Several post-processing features have also been added that allow you to more effectively view water depth along each of the stream channels in the GSSHA™ model. We have added two new tutorials and a new volume (Volume 6) to our list of WMS tutorials that walk you through the process of setting up a basic GSSHA™ model with overland flow and stream routing and using NEXRAD Radar precipitation with your GSSHA™ model. A new web site, **gsshawiki.com** [1], has also been launched. This web site contains all the latest

GSSHA™ documentation and a wealth of information for GSSHA™ modeling.

HMS gridded rainfall (NEXRAD) support

WMS 8.1 supports an exciting new feature that makes it easier to work with quasi-distributed hydrologic models. This feature makes it simple for you to use NEXRAD RADAR rainfall data in XMRG format, such as archived rainfall data from the National Weather Service [1]. This feature allows you to read a set of NEXRAD rainfall grids and compute the Thiessen polygon-based gage weights for each sub-basin and the total precipitation (or a precipitation distribution) for each gage based on the rainfall grids.



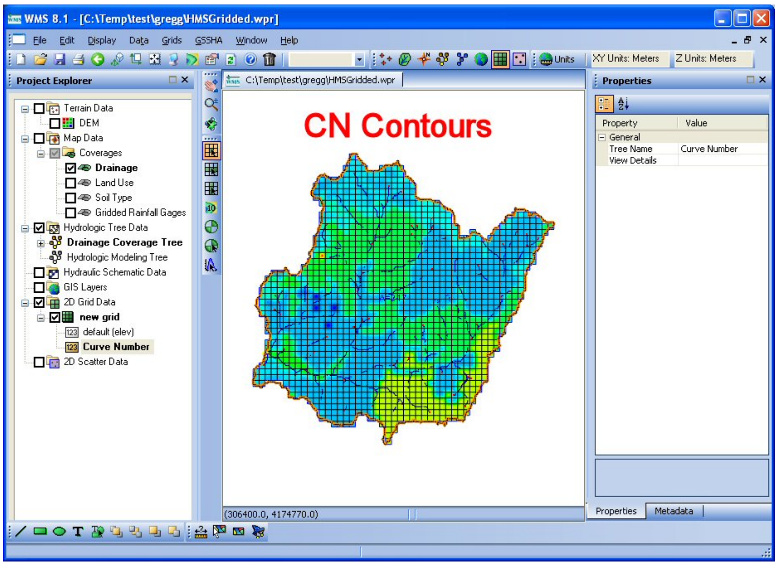
NEXRAD Rainfall Gage Example

WMS 8.1 also has features that allow you to convert radar-derived precipitation grids directly to formats that can be used in two different hydrologic models: The quasi-distributed MODClark model, which is part of the HMS interface and the distributed GSSHA™ model.

Improved HMS interface and MODClark modeling support

Several improvements have been made to the HMS interface in WMS 8.1.

One of the most exciting improvements allows you to define a grid representing an HMS MODClark model at any resolution and compute a Curve Number, rainfall value, and travel time at each cell on the grid. After defining the MODClark model in WMS, you can export the model to HMS and run the MODClark simulation. The powerful MODClark modeling capabilities in WMS are not available in any other software product.

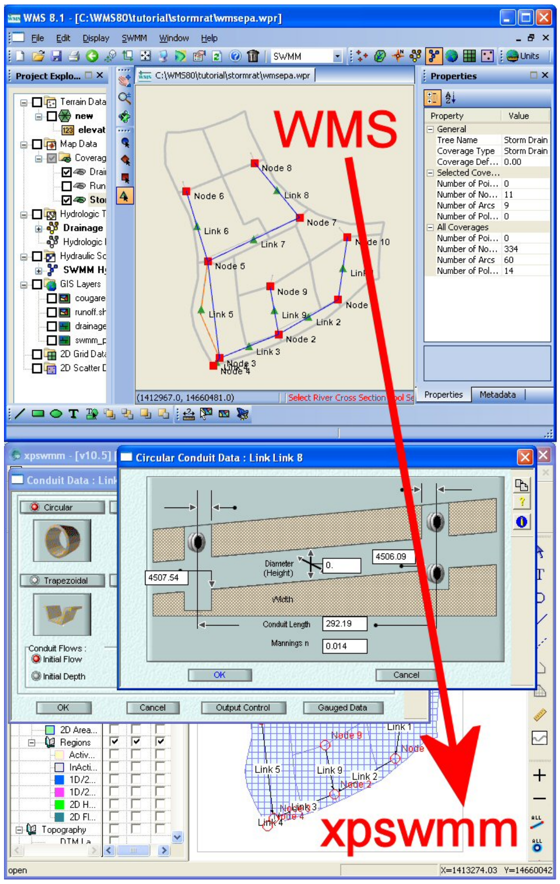


MODClark Model Example

Hydraulic Modeling

Storm water quality and quantity modeling with xpswmm and EPA SWMM interfaces

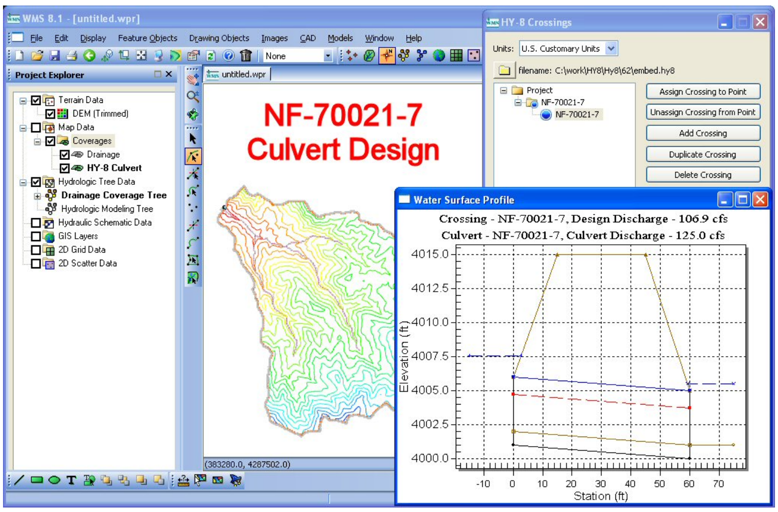
Besides supporting hydrologic modeling with xpswmm and EPA SWMM, WMS supports hydraulic modeling through the "Storm Drain" coverage and the River module. You can import or draw a conceptual model of your storm drain networks in the storm drain coverage and then convert your conceptual model to a schematic in the River module. From the river module, you can export your storm drain network to xpswmm or EPA SWMM where the model can be run. Using the new WMS feature to export (and import) xpswmm and EPA SWMM files, you can build your model and perform model computations in the familiar and GIS-enabled WMS interface. After building and defining your model in WMS, you can export your model to SWMM and finish your model and view the results in the SWMM interface.



WMS To xpswmm Example

Updated HY-8 7.1 culvert analysis computations and an improved interface

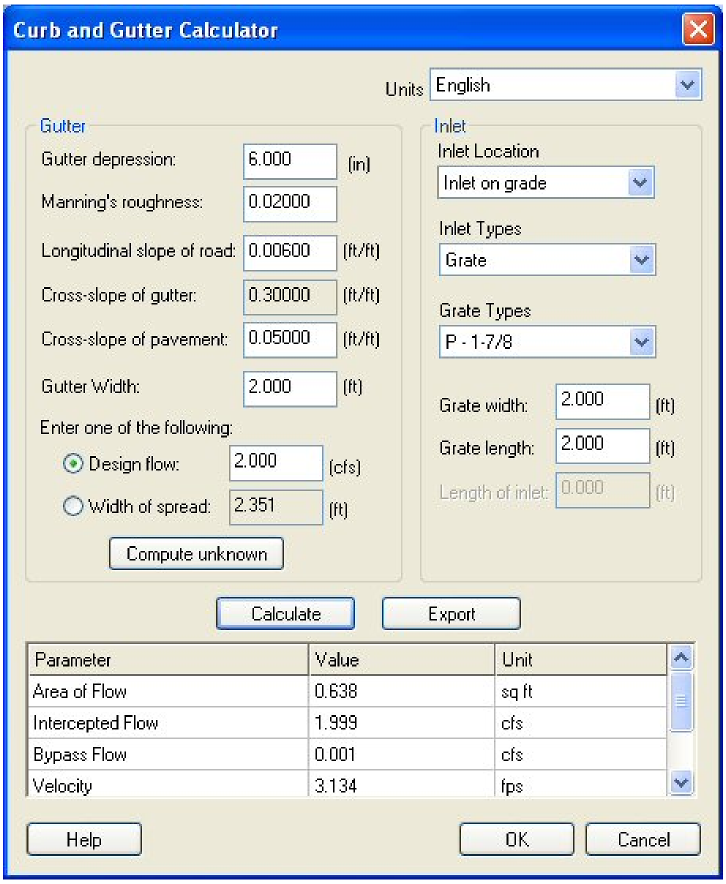
WMS 8.1 has an improved interface with the FHWA HY-8 culvert analysis program. Since HY-8 7.1 was developed by Aquaveo (the developers of WMS), we were able to integrate the HY-8 7.1 culvert analysis interface and code directly into WMS. With the new HY-8 Culvert coverage, you can define the location of your culvert and edit the data associated with a culvert. The WMS software developers have also linked the HY-8 culvert coverage with the detention basin calculator, allowing you to route a hydrograph through a culvert and determine the effect of the culvert on the hydrograph. The WMS HY-8 interface offers report generation, energy dissipation structure analysis capabilities, and many other powerful features directly within the WMS interface.



HY-8 7.1 WMS Culvert Design Example

Improved support of FHWA hydraulic modeling tools

WMS 8.1 has improved the support of the FHWA hydraulic modeling tools. The channel calculator, weir calculator, and the curb and gutter calculator have all been improved to more effectively coincide with FHWA's HEC 22 computation guidelines.

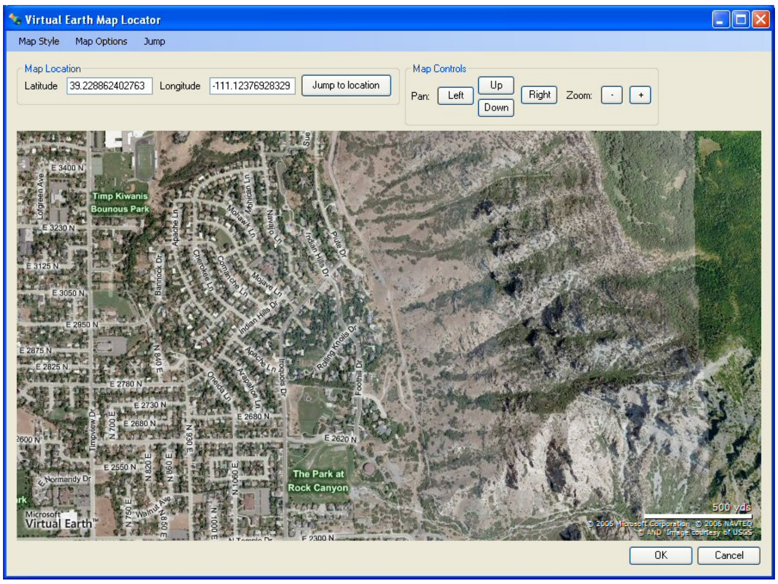


WMS Curb and Gutter Calculator

General Enhancements

In WMS 8.1, the "Get Data" toolbar has been expanded to include two new buttons: The "Get Data From Map" button and the "Hydrologic Modeling Wizard" button:

Virtual Earth map locator tool



The WMS Virtual Earth Map Locator Tool

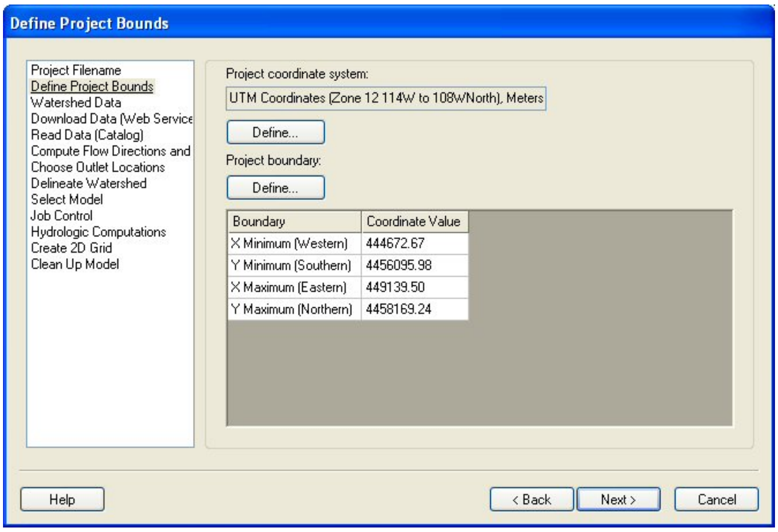
The "Get Data From Map" button brings up the WMS Virtual Earth map locator tool. This tool is powered by Microsoft Virtual Earth, and accesses data from the internet to allow you to view maps anywhere in the world. Once you zoom into the region you're interested in modeling, WMS will bring up a web data service client which will allow you to select from several data download options. From this dialog, you can select to download NED (DEM), Land Use, or Microsoft TerraServer aerial imagery and topographic data. The data you select is downloaded and read into WMS. This powerful combination of tools will allow you to streamline your data collection process so much of your data collection can be done within the WMS interface.

Improved web service client tools

Along with the virtual earth map locator tools, the web service client capabilities of WMS have been greatly improved. In previous versions of WMS, WMS had difficulties downloading data. Many of these difficulties originate from the web service servers, and WMS cannot improve much on these problems. However, the WMS developers have streamlined our web service client tools to give you feedback on the status of your downloads and to fix several problems that were in previous version of the web service client.

Hydrologic modeling wizard

The "Hydrologic Modeling Wizard" button in the Get Data toolbar steps you through the process of building a hydrologic model. This powerful wizard starts by helping you define your project coordinate system and gather your data, then steps you through the process of delineating your watershed and computing the parameters for your hydrologic model. When you are done with the wizard, you will have a hydrologic model that is completed or nearly completed for any of the hydrologic models supported in WMS.

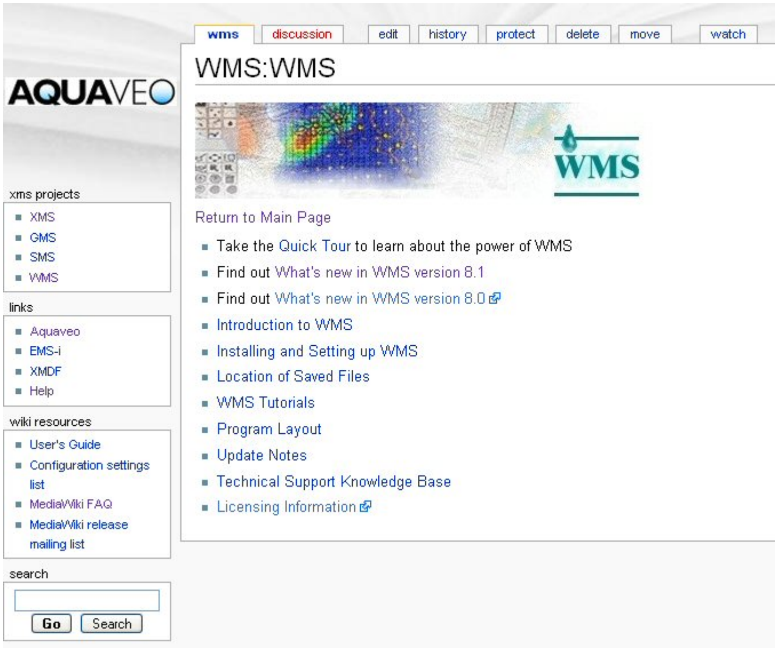


The Hydrologic Modeling Wizard

Improved graphics

The WMS graphics display speed has been dramatically increased. Previous versions of WMS used a software-based approach to rendering your data, but WMS 8.1 uses your graphics hardware to render your data. Along with this modification, the WMS software developers have added Windows Vista graphics support to WMS 8.1.

A new user-editable WMS "Wiki" online help



The http://xmswiki.com WMS Help Page

The WMS software developers have developed the xmswiki.com [3] home page. Instead of the traditional WMS help file, all the help is now available online from the xmswiki.com site. This help site uses the "Mediawiki" engine, which is the same code base used by Wikipedia [4] to run their site. This means that anybody who knows about WMS, including you, can modify a help topic or create a new topic. Be assured that the WMS software developers will monitor changes to the WMS wiki, so the xmswiki.com content will be accurate. But with this powerful online editing tool, the WMS help will be much more comprehensive and up-to-date. The xmswiki.com site also features a Google search engine, bringing the power of Google search to the WMS help pages.

XMDF file I/O

The WMS 8.1 file format has been modified to use the XMDF file format as the default format. This format provides for faster WMS file I/O and good WMS file viewing capabilities with the NCSA HDFView [5] editor which can be used to edit any XMDF-formatted file.

References

[1] http://www.xpsoftware.com/
[2] http://www.epa.gov/ednnrmrl/models/swmm/index.htm
[3] http://www.xmswiki.com
[4] http://en.wikipedia.org
[5] http://hdf.ncsa.uiuc.edu/hdf-java-html/hdfview/

What's new in WMS version 8.2

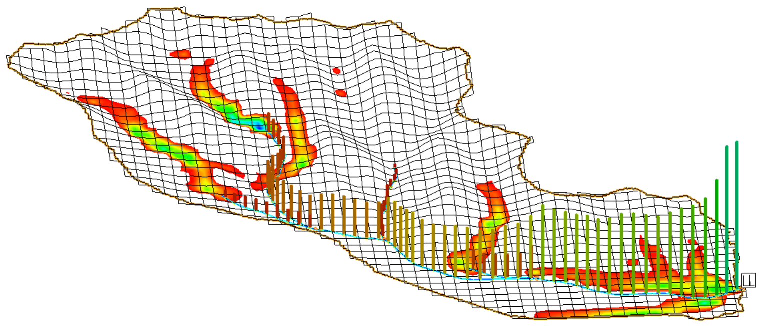
The WMS software development team is excited about the release of WMS 8.2! This page lists the exciting new features that have been added to WMS 8.2.

Model Enhancements

Hydrologic Modeling

Support of calibration and batch mode processes (stochastic modeling) in the GSSHA interface

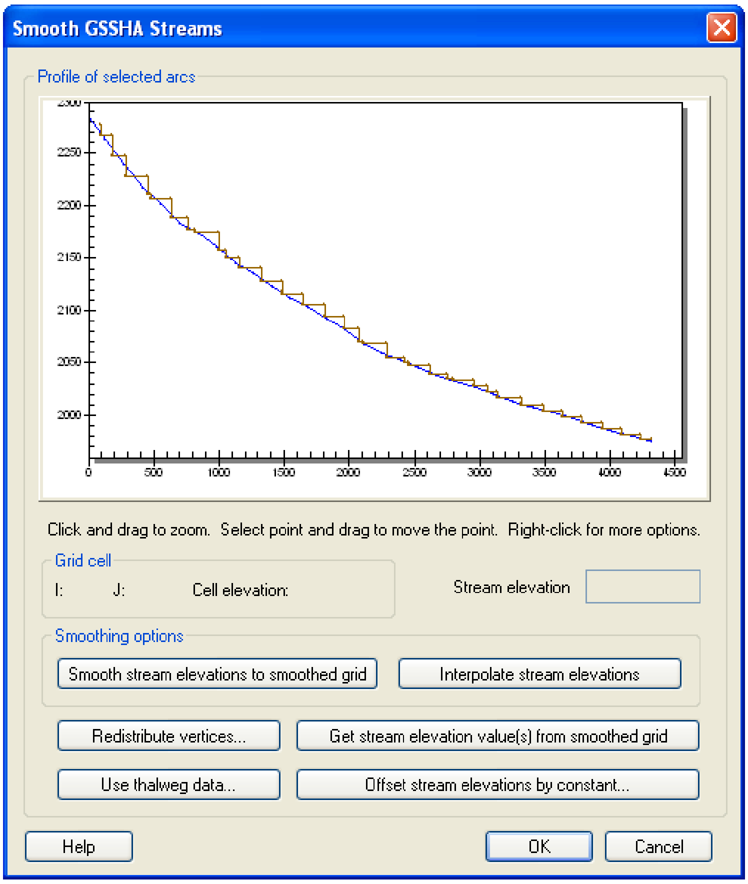
Using WMS 8.2, you can run GSSHA in calibration mode to determine optimum parameters or in batch mode to run using a range of input values.



Calibration and Batch Mode Processes

Improved stream model checker, better stream visualization, stream adjustment to match grid, and improved stream smoothing

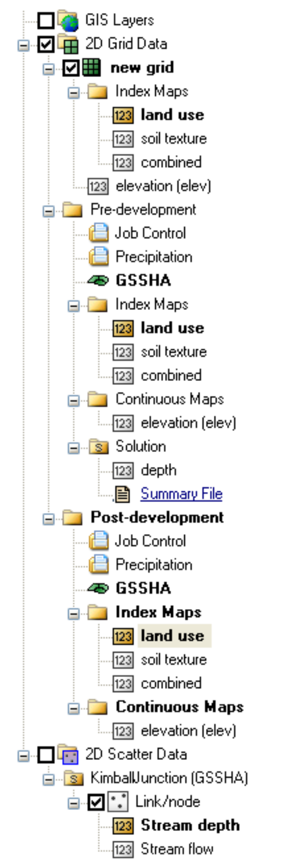
The hydrologic modeling wizard has been improved in WMS 8.2 to allow for editing stream elevations before creating a 2D grid. When the 2D grid is generated, the grid cell elevations are set to match the stream elevations. Matching up these elevations reduces the chance for errors in running your GSSHA model when stream routing is defined.



Improved Stream Smoothing and Grid Matching Capabilities

Ability to create multiple GSSHA scenarios (multiple GSSHA job controls, sets of index maps, mapping tables, solutions, etc.)

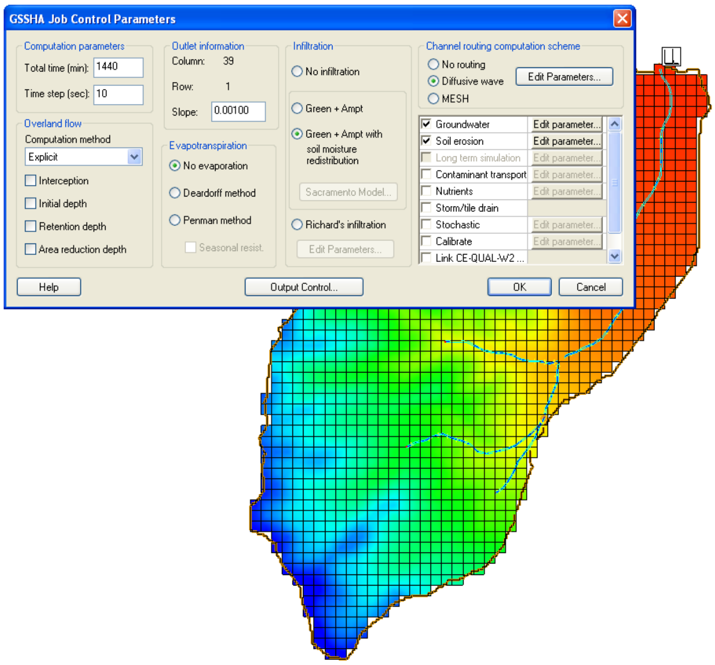
Multiple scenarios, such pre-development and post-development conditions, can be defined in a single instance of WMS. Each of these scenarios can be run and the results from each scenario can be compared.



Example of Multiple Scenarios in the Project Explorer

Support of wetlands, sediment transport, and groundwater/surfacewater interaction routines in GSSHA [1]

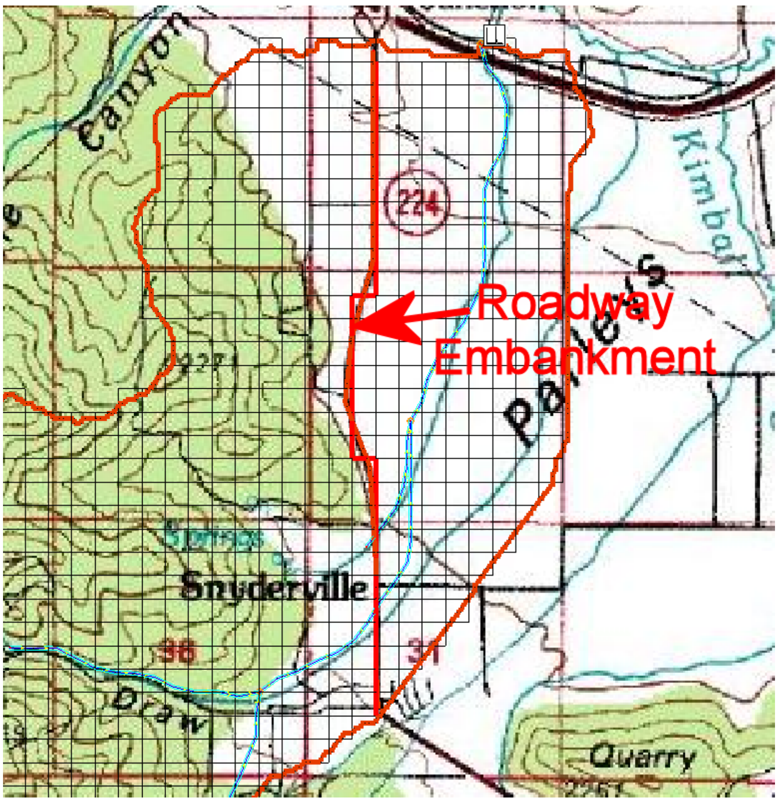
WMS 8.2 contains full support for wetland, sediment transport, and groundwater/surfacewater interaction capabilities in GSSHA. Tutorials for each of these capabilities are available on gsshawiki.com [1].



Groundwater and Sediment Transport Example

Improved ability for defining embankment arcs

WMS 8.2 has improved ability for defining embankment arcs and running GSSHA embankment routines that was not available in previous version of WMS.

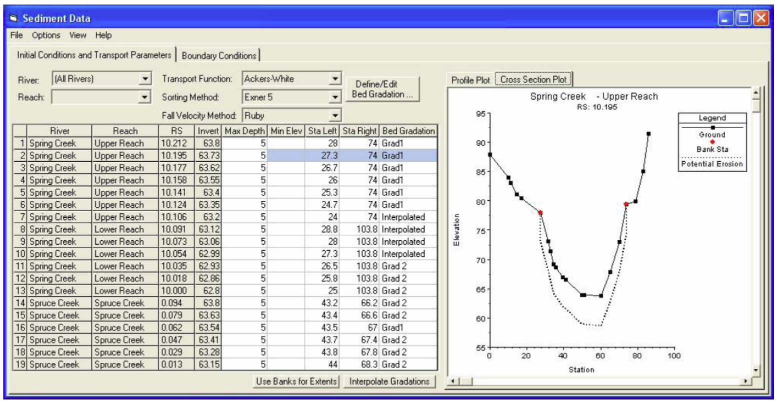


Example Roadway Embankment

Hydraulic Modeling

Support of HEC-RAS 4.0 and sediment transport

WMS 8.2 contains support for the latest version of HEC-RAS, version 4.0, which has sediment transport capabilities.

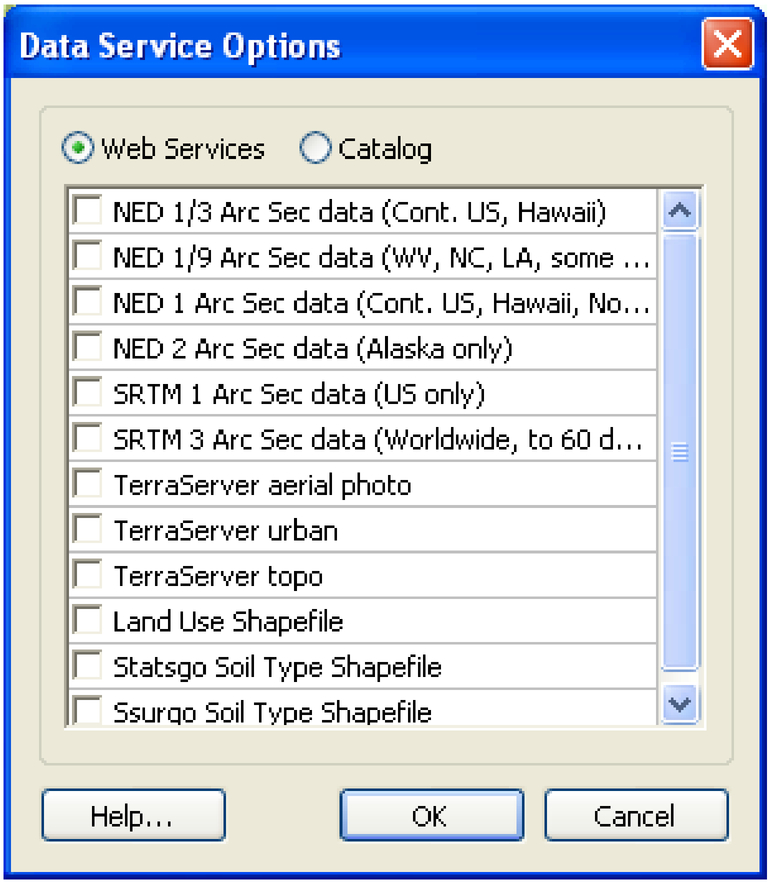


HEC-RAS 4.0 Sediment Transport

General Enhancements

Improved web service client tools

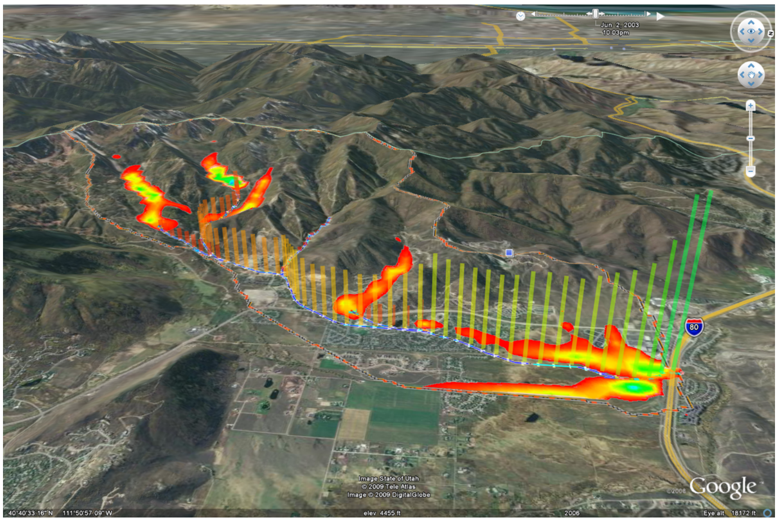
WMS 8.2 Includes all types of NED and SRTM elevation data downloads, Terraserver images, and a web catalog that allows you to download any type of data if the data is available. These data sets can be downloaded directly from WMS and from the hydrologic modeling wizard. In WMS 8.2, practically all of the data required for a hydrologic study can be downloaded directly from WMS for any location in the world.



Web Data Service Options

Output animated KMZ files to Google Earth

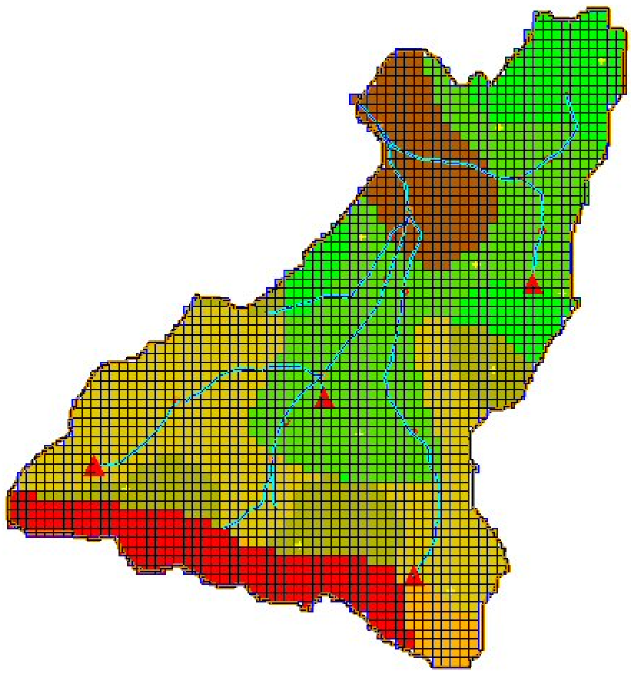
One of the results from running a GSSHA model is water depth at any location of your watershed for any time during your simulation. With WMS 8.2, you can export an animation of the water depth contours and any other parameter that is computed in GSSHA to a file that can be read by Google Earth. This capability is great for presentations of GSSHA model results.



Animation Exported to Google Earth

Dataset zonal classification

Using WMS 8.2, you can combine datasets such as erosion and deposition to view areas of low erosion and high deposition or high erosion and low deposition based on certain criteria. You can view the results of these queries throughout your watershed.



Zonal Classification

References

[1] http://gsshawiki.com

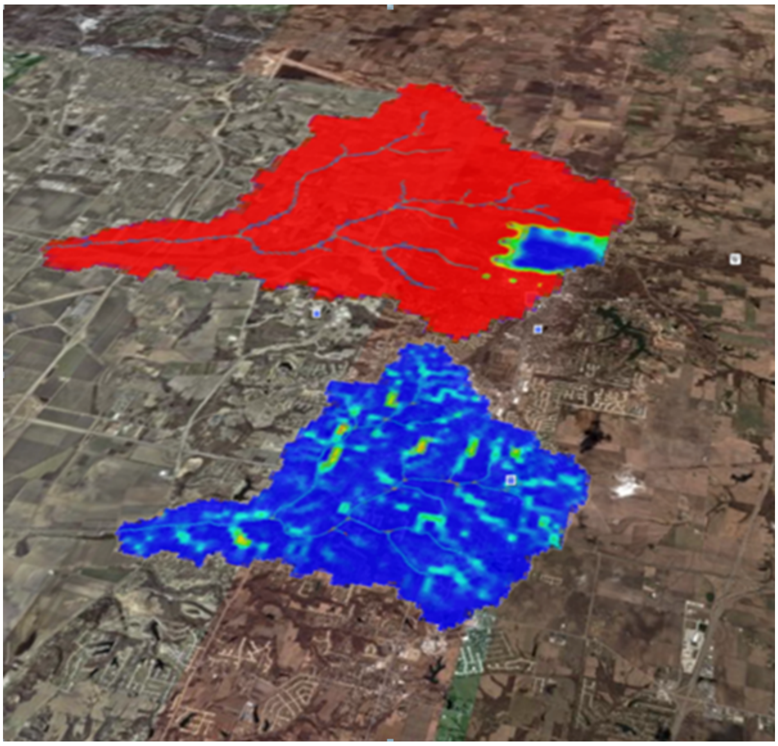
What's new in WMS version 8.3

The WMS software development team is excited about the release of WMS 8.3 (beta)! This page lists the exciting new features that have been added to WMS 8.3.

WMS Enhancements

Export rainfall datasets to KMZ (Google Earth) animation files

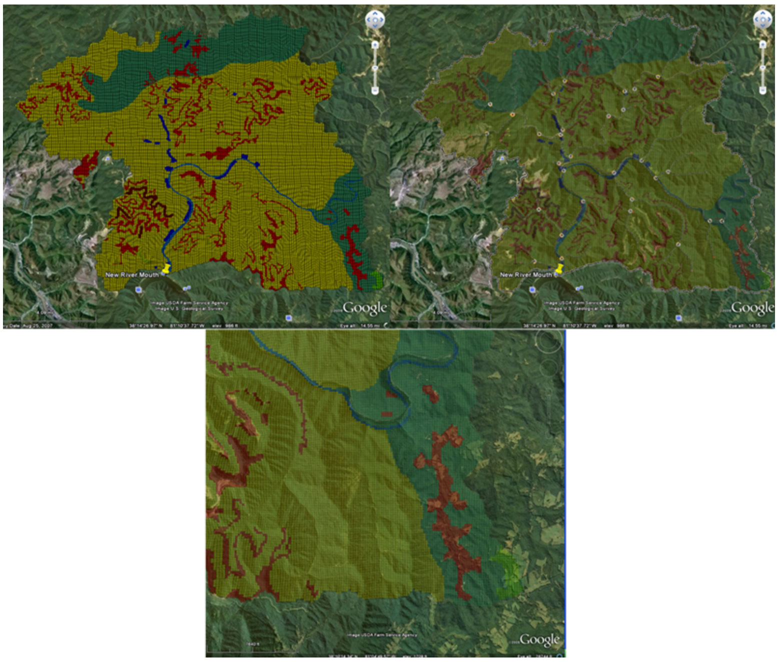
Using WMS 8.3 you are able to select the solution dataset, the scattered data dataset, and the rainfall dataset and export all three to a KMZ file. In the film loop dialog, you can specify the display elevation associated with your rainfall dataset, and view rainfall intensity at the same time you are viewing its effects in the overland plane and the river network.



KMZ export to Google Earth. The upper layer represents the radar rainfall dataset and the ground-level layer represents the overland water depths.

Export index map cells to Google Earth as vector polygons

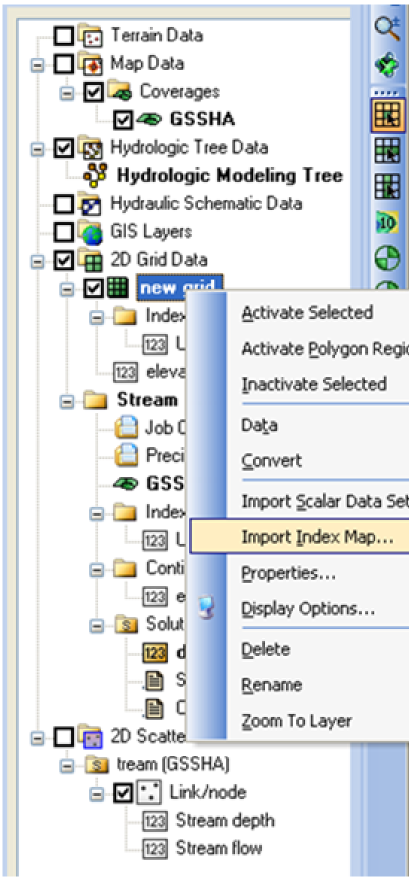
With WMS 8.3 you get a variety of visualizing options. For example, the different index maps that can be created in WMS can now be exported as .kmz files that can be displayed in Google Earth.



Land Use Index Map exported to Google Earth. You can set different transparency levels.

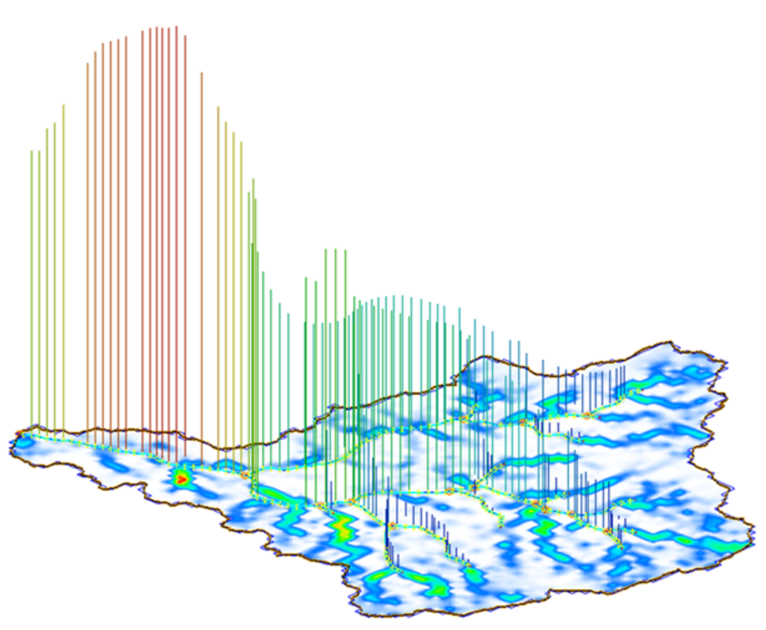
Import GSSHA index map files

WMS 8.3 allows you to import any index map file, such as Land Use, Soil Type, Combined Index Map, etc. from one GSSHA project to another GSSHA project with the same grid.



Menu item to import an index map

Support of Contaminant Transport and NSM modeling in GSSHA [1]

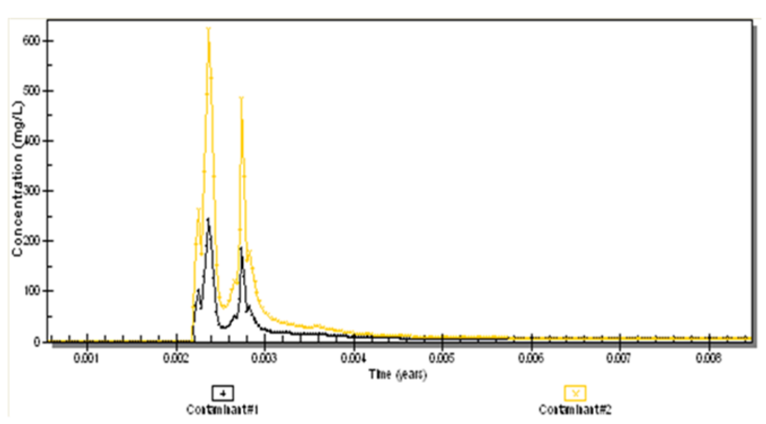


Constituent Mass in the overland plane and channel network

WMS 8.3 supports constituent transport modeling, which can be simulated as simple first order reactants or with the full nutrient cycle using NSM. You can input a constituent concentration at any point in your watershed (point and/or non point source) or on a stream and then GSSHA outputs maps of mass and concentrations at each point in your watershed and in your stream network.

NSM can be coupled with any hydrologic and hydrodynamic model transport component; this means that NSM deals with transforming processes of water quality constituents in the overland plane and receiving water bodies. Tutorials for each of these capabilities are available on gsshawiki.com [1].

Ability to create mass and concentration plots for NSM constituents and for simple first order constituents

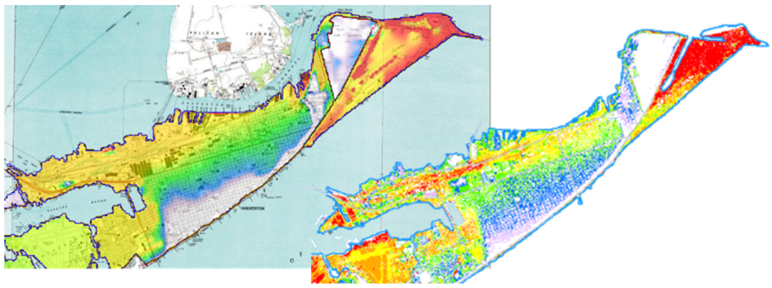


Plot of contaminant concentration for two different contaminants at the outlet location

Not only can constituents be visualized in the overland flow plane at any cell in your watershed, but specific plots of constituent mass and concentrations can be generated in the outlet of your watershed using WMS 8.3. This works both for simple first order reactants and for NSM constituents such as nitrogen, phosphorus and carbon species.

Support of Overland flow boundary conditions in GSSHA

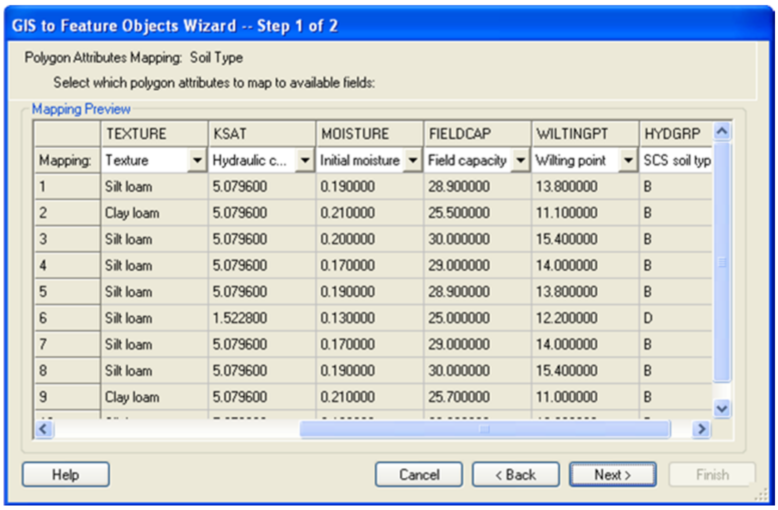
Using WMS 8.3 you can model coastal storm surges or areas around a standing water body using overland flow boundary conditions, which can be defined as constant slope, constant stage (water surface elevation), or variable stage (water surface elevation). GSSHA will take a map of the boundary conditions and apply that condition at the beginning of each time step.



Coastal storm surge simulation in Galveston Island Texas. The figure on the left shows GSSHA inundation depths using overland boundary conditions. The figure on the right is an inundation map for the same event produced by NOAA.

Improvements on Joining SSURGO Data

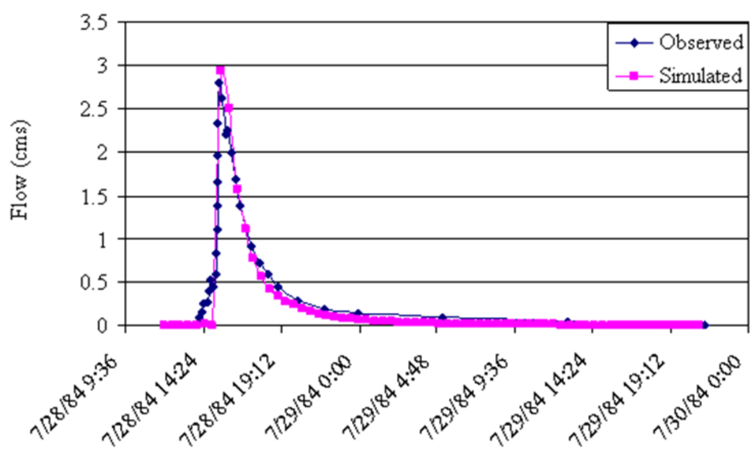
Using the “Join SSURGO Data” command in WMS 8.3, you can now join more attributes besides those associated with the *.dbf file. Attributes such as hydrologic soil group (HYDGRP), texture, hydraulic conductivity (KSAT), moisture, field capacity (FIELDCAP) and wilting point (WILTINGPT) can now be mapped to the WMS soil coverage.



This figure shows the different values extracted from the SSURGO soil that are mapped in the WMS soil coverage.

Improvements on GSSHA Stochastic Model Calibration

Besides having the capability of running in calibration mode or batch mode, with WMS 8.3 you can automatically read in and run a model with the calibrated parameters and decide whether or not to substitute them into your model.



Plot of observed data and calibrated data for Reynolds Creek

Capability to define an input hydrograph and/or contaminant source at any location in the stream network

As mentioned previously, WMS 8.3 allows you to specify a contaminant source at any point in your stream network that can account for point-sources in the stream network. In addition, you can define an input hydrograph at any point in your stream network, which can be particularly useful if you want to account for the hydrology of upstream areas into your watershed.

What's new in WMS version 8.4

What's new in WMS 8.4

- Updated to be compatible with the latest version of GSSHA. New tutorials are available for use with WMS.
- Integration with FHWA's HY-8 and Hydraulic Toolbox software.
- HY-8 modeling wizard: Allows you to design or analyze culverts using the linkage between WMS, HY-8, and the Hydraulic Toolbox.
- Integration of the Time Series Editor with WMS. This tool allows you to find and download data from the internet and to modify existing time series data.
- New method of computing Flow Directions/Accumulations for basin delineation using TauDEM, allowing for multiprocessor computation of Flow Directions and accumulations.
- Full compatibility with 64-bit computers, allowing for faster processing and the ability to process larger datasets than ever before.

Bug fixes

DEM contours to feature objects crash in WMS 8.3: 2180

Save GSSHA Group Dialog: 2183

Multi-Select index map grid cells: 2143

converting dem contours to feature objects: 2175

Issues with GSSHA Automated Calibration: 2151

Error reading GSSHA Stochastic Simulation results: 2147

WMS crashes when right clicking on a TIN Tree Item for a TIN that's been deleted: 2141

Error building pyramids: 2131

DEM File won't read in: 2084

Error message when trying to open an image: "The application has failed to start because gmp-vc90-mt.dll was not found": 2075

Zoom tool not working in Edit DEM Elevations plot window: 2071

The "Select Shapes Tool" in the GIS module should be an active tool but it is inactive.: 2065

Changes to GSSHA .cmt file requested by Chuck: 277

Allow users to set the GSSHA NUM_INTERP value

Sediment interface enhancements: GSSHA

Error in GSSHA Calibration: 2156

GSSHA calibration output files: 2153

Allow editing polygon-selected index map ID's in the properties window: 2060

Hydrologic modeling wizard Define project boundary: 2288

WMS does not read observed data file: 2248

Check while reading parameter and calibration file in GSSHA automated calibration: 2249

Add GSSHA Calibration Parameters in WMS: 2247

flow vectors don't read in: 2244

Save File button not working in Coverage Overlay dialog: 2239

Crash when deleting GSSHA model: 2236

Report the name of the contaminant when reading the contaminant transport solution: 2235

Add option to select whether to export contaminant mass and concentration to GSSHA output control: 2234

WMS not writing all lakes to file: 2217

Simple dam Break not finding Cross Section: 2215

Tutorial change request: 2184

Display options: 2366

problems numbering Branch: 2332

Run GSSHA model button in the Hydrologic modeling wizard not working: 2321

Contour Options button not working: 2311

Distinguish Data/Model/Solution in GSSHA project explorer. 1) Use a different color scheme, 2) Separate the sections out, and 3) Put a symbol M for models as we have S for solutions

If groundwater head and aquifer bottom is available in the model, display the following in the Smooth stream arcs dialog and allow user to edit all of these

Add the new Tc method to MODRAT

In Hydrologic Modeling wizard, the coordinates for project boundary are not defaulted properly. Dr Nelson thinks it is good idea to default them to 0

In the same dialog of the modeling wizard, if you change the project coordinates, it looks like WMS tries to transform the coordinates in the project bound coordinates list also causing WMS to crash

While saving GSSHA project, the .cmt file had a soil type index map assigned to several processes even though those processes were not turned on. The processes that were not turned on include evapotranspiration, soil erosion, and possibly others. GSSHA r

In GSSHA Job Control/Storm-tile drain Edit parameter button, change the text to "Allow GSSHA to redistribute Superlink Vertices"

Super link/ Super Junction numbering is still not working. If you create the superlinks and turn the Storm/Tile Drain option on Job control, the numbering is not correct. Turning the Storm/Tile Drain option on Job control and then creating the arcs seemed

In Job Control/Edit parameters for storm/tile drains, put an option to select a method that specifies how water gets into the drains. The default option should be Cook Method (need to make sure if the name is correct) and next option should be Drain Mod M

Be able to specify a depression mask for running cleandam so you don’t change areas for specified zones

Change the user interface to turn this option on, specify the XY boundary condition files, and specify the larger GSSHA model so the mask can be written to this location

Change the user interface to turn this option on for the larger model and to specify the small GSSHA model so the mask can be used in running the simulation.

Read and write the project file cards and data files required to implement this option

Read and write DEMs as GeoTIFF files

Create a display option to display the wetland parameters from the GSSHA polygon attributes dialog if this display option is selected

WMS cannot load images after converting DEM to TIN: 2629

Severe WMS/Global Mapper bug when converting DEM to TIN: 2627

Modrat Wrapper not finishing: 2624

Joining SSURGO data bug: 2623

When loading the hydrograph solution to HEC-1 WMS Gives an error: 2597

WMS crashes when converting DEM to TIN after editing TOPAZ generated streams: 2578

WMS crashes when reversing directions of a stream network: 2577

The Cross Section Attributes dialog gives bogus help strings: 2566

Click on GSSHA Smooth Stream Dialog causes a crash: 2565

Add Autocheck for updates to WMS interface

Write KMZ file animations as Super-overlays

Fixed several MODRAT bugs

WMS crashing when loading GIS data: 2436

Build Polygons Crash: 2400

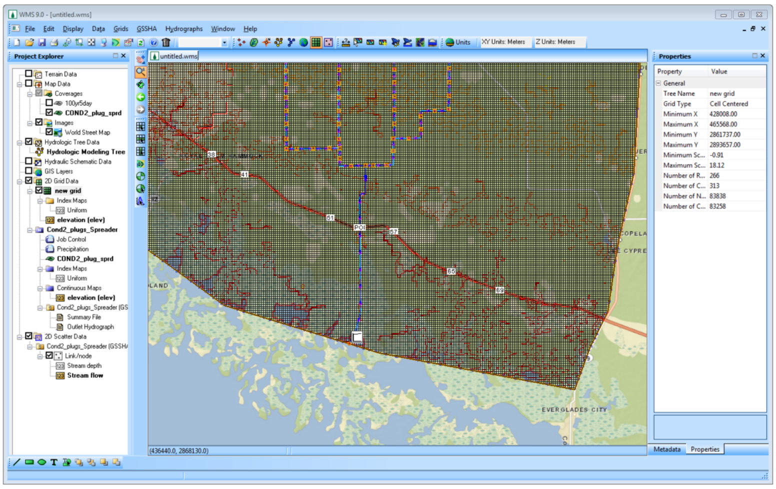
WMS 8.3 unable to Connect to Sever when selecting from web service: 2389

What's new in WMS version 8.5

The WMS software development team is excited about the release of WMS 9.0! This page lists the exciting new features that have been added to WMS 9.0.

What's new in WMS 9.0

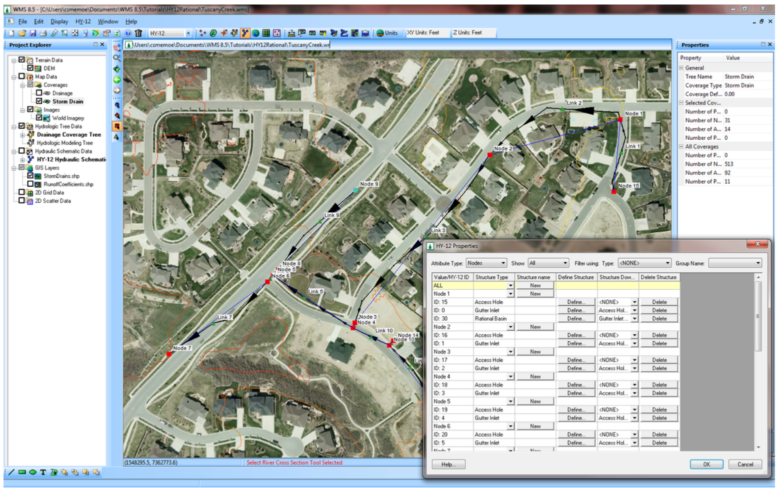
Updated Interfaces



The GSSHA model is a 2D watershed model that includes 1D hydraulic modeling, groundwater modeling, and other advanced watershed modeling components.

The NSS (National Streamflow Statistics) interface has been updated to use the latest NSS database. The MODRAT interface has also been updated. Renumbering has been improved in the MODRAT interface and several issues have been fixed. Several new features and enhancements have been added to the GSSHA interface. First, the storm and tile drain modeling capability of GSSHA has been improved by adding the capability to add multiple pipes in a "superlink", which represents a network of pipes in GSSHA. The algorithm for determining embankments has also been reworked to make the algorithm more efficient and more accurate. The GSSHA tutorials have also been updated and improved to include the latest enhancements in the GSSHA code. Also, GSSHA itself has been reworked to make the program more stable.

FHWA HY-12 Storm Drain Modeling

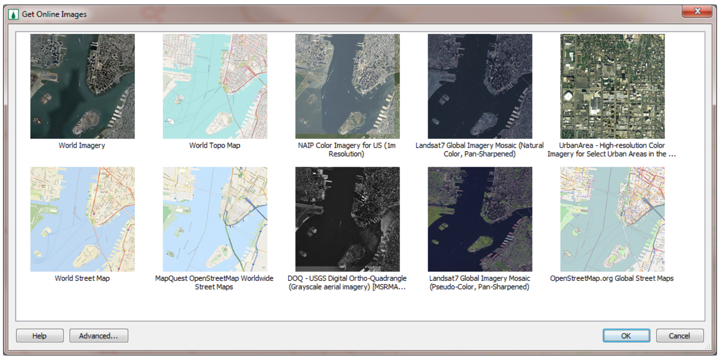


The HY-12 model is a storm drain model developed by Aquaveo and the FHWA (US Federal Highway Administration).

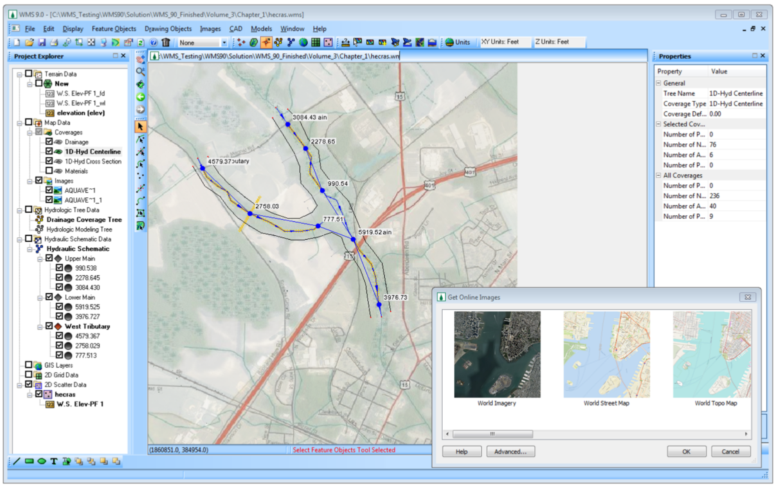
WMS now includes an interface to FHWA's HY-12 storm drain modeling program. This interface allows you to create a storm drain layout and assign rational method computations, curbs and gutters, access holes, pipes, channels, and other storm drain network features to the storm drain layout. Much of the data can be computed automatically using the interface, and other data values can be entered in the easy-to-use windows. The other storm drain modeling interfaces, including the SWMM and xp-swmm interfaces, have also been upgraded to make your model easier to build and maintain.

Online Image (Web Map Service) Capabilities

One of the most exciting new features in WMS is the Get Online Maps tool  in the Get Data Toolbar  . This tool allows you to open a web map service as an "online image" and use it as you would any other image in the WMS interface. Since the web can be a little slow, there is an option to convert the online image to a static (locally saved) image that is saved with your WMS project and that displays much faster than the online image.

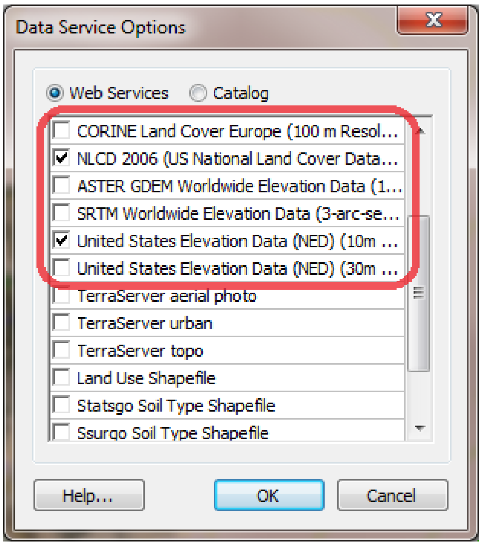


The online image capabilities in WMS 9.0 allow you to view and use a map showing any part of the world with your data.



The online image capabilities in WMS 9.0 allow you to download and use georeferenced images for any part of the world with your data.

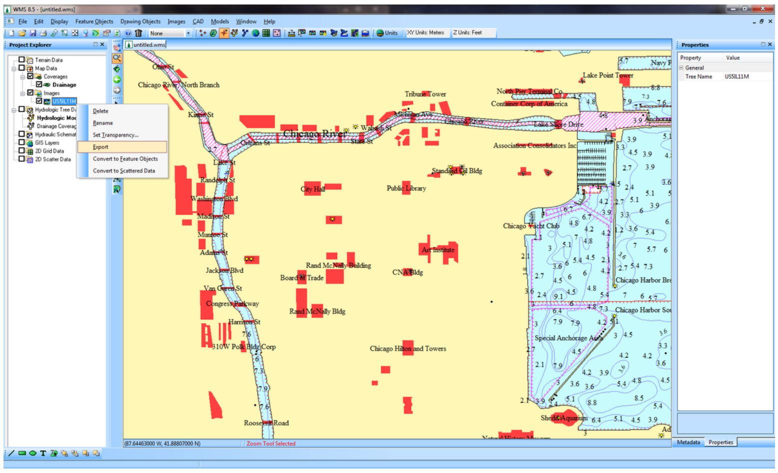
Improved Web Service Tools



Six new web service data sources are available in WMS 9.0. These services allow you to download data for many parts of the world.

Six new web services have been added to the list of web data that can be downloaded from the WMS Get Data Toolbar  . These web services include some that were previously available, such as United States National Elevation Datasets, as well as newly available datasets, such as the CORINE European land cover database, the NLCD US National Land Cover database, and the ASTER Worldwide Elevation Data database. These new datasets are much faster than previously available datasets and include a progress bar so you can view the progress of your data download.

Support of New File Formats



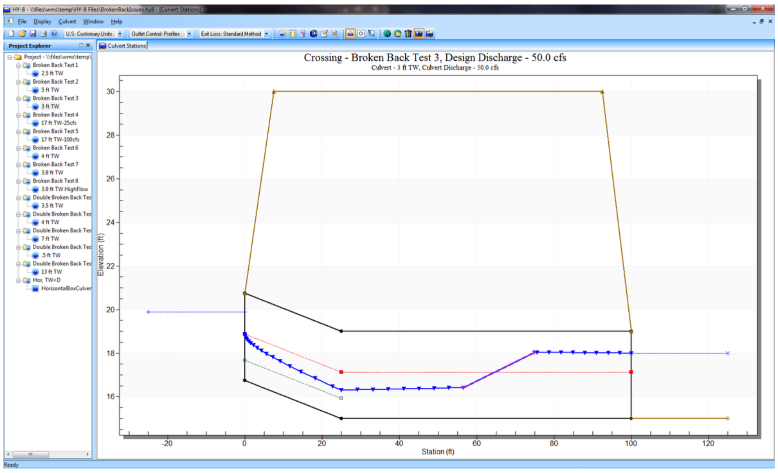
You can read and display almost any commonly used vector data source as a vector image in WMS.

We have added support for almost all the commonly used vector file formats in the new version of WMS. Some vector files, such as DXF, DWG, and ESRI Shapefile format, still read the way they have always read into WMS. But support for "Vector-based images" has been added to WMS that allows you to read any file that can be read using the Global Mapper software in WMS (A license to Global Mapper is not required). For a complete listing of the vector formats supported in WMS, visit the Global Mapper web site [1]. Right-clicking on a vector image allows you to export to one of many formats or convert linear data to feature objects or scattered data (XYZ). The following formats can be exported using the right-click command:

- DXF Files (*.dxf)
- Area Shapefiles (*.shp)
- Line Shapefiles (*.shp)
- Point Shapefiles (*.shp)
- Google Earth KMZ Files (*.kmz)
- MapInfo MIF/MID Files (*.mif)
- MapInfo TAB/MAP Files (*.map)
- Simple ASCII Text Files (*.txt)
- CSV (Comma-separated value) Files (*.csv)

- SVG Files (*.svg) [1]

FHWA HY-8 7.3 and Hydraulic Toolbox 3.0 Support



HY-8 7.3 is supported in the latest WMS.

The FHWA's HY-8 7.3, which includes many new culvert modeling capabilities, is supported in the new version of WMS. The new version of the FHWA's Hydraulic Toolbox (3.0) is also supported with the new version of WMS. The new tools in HY-8 7.3 include modeling of hydraulic jump profiles, broken back culverts (culverts with a change in slope), and horizontal and adverse slopes in culverts. Documentation showing the capabilities included in the latest version of HY-8 is included on the HY-8 wiki. The newest version of the hydraulic toolbox includes tools for culvert assessment and for determining a riprap or streambed gradation curve using a digital image.

Bug fixes

To view the list of bugs fixed in WMS 9.0, visit the WMS bugfix page.

References

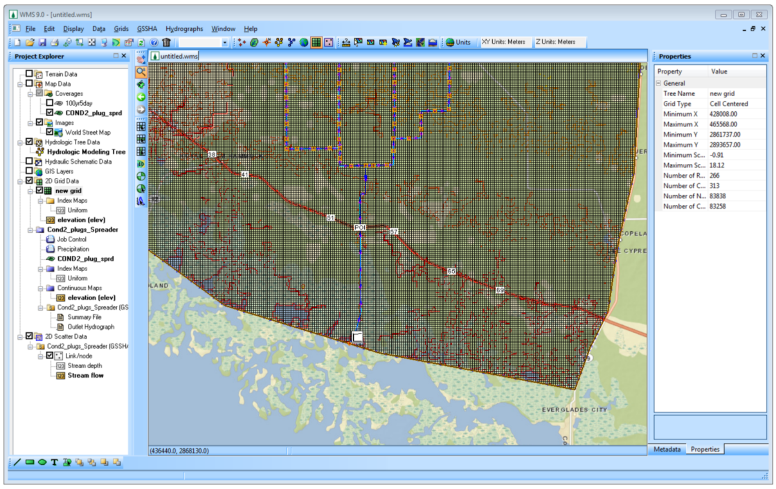
[1] http://www.globalmapper.com/product/formats_vector.htm

What's new in WMS version 9.0

The WMS software development team is excited about the release of WMS 9.0! This page lists the exciting new features that have been added to WMS 9.0.

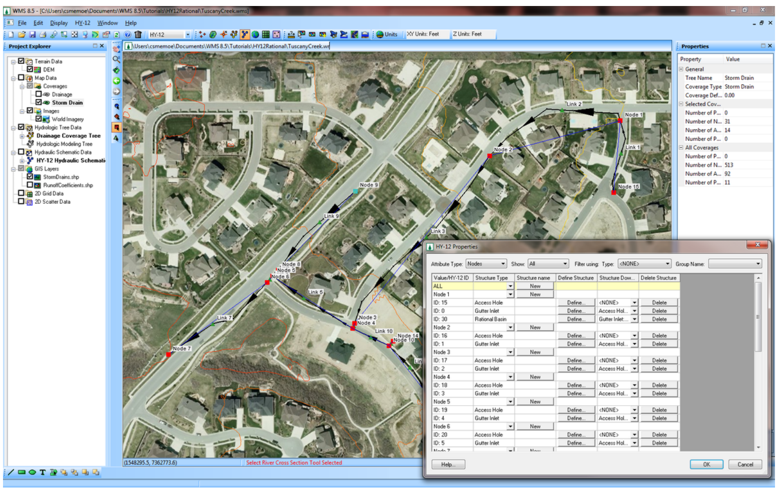
What's new in WMS 9.0

Updated Interfaces


The GSSHA model is a 2D watershed model that includes 1D hydraulic modeling, groundwater modeling, and other advanced watershed modeling components.

The NSS (National Streamflow Statistics) interface has been updated to use the latest NSS database. The MODRAT interface has also been updated. Renumbering has been improved in the MODRAT interface and several issues have been fixed. Several new features and enhancements have been added to the GSSHA interface. First, the storm and tile drain modeling capability of GSSHA has been improved by adding the capability to add multiple pipes in a "superlink", which represents a network of pipes in GSSHA. The algorithm for determining embankments has also been reworked to make the algorithm more efficient and more accurate. The GSSHA tutorials have also been updated and improved to include the latest enhancements in the GSSHA code. Also, GSSHA itself has been reworked to make the program more stable.

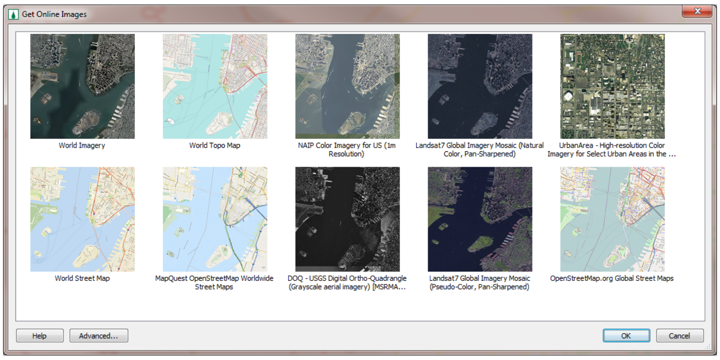
FHWA HY-12 Storm Drain Modeling


The HY-12 model is a storm drain model developed by Aquaveo and the FHWA (US Federal Highway Administration).

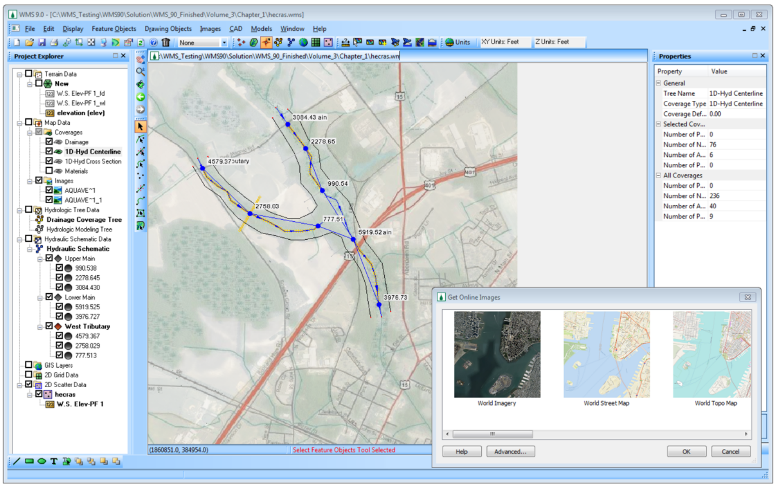
WMS now includes an interface to FHWA's HY-12 storm drain modeling program. This interface allows you to create a storm drain layout and assign rational method computations, curbs and gutters, access holes, pipes, channels, and other storm drain network features to the storm drain layout. Much of the data can be computed automatically using the interface, and other data values can be entered in the easy-to-use windows. The other storm drain modeling interfaces, including the SWMM and xp-swmm interfaces, have also been upgraded to make your model easier to build and maintain.

Online Image (Web Map Service) Capabilities

One of the most exciting new features in WMS is the Get Online Maps tool  in the Get Data Toolbar  . This tool allows you to open a web map service as an "online image" and use it as you would any other image in the WMS interface. Since the web can be a little slow, there is an option to convert the online image to a static (locally saved) image that is saved with your WMS project and that displays much faster than the online image.

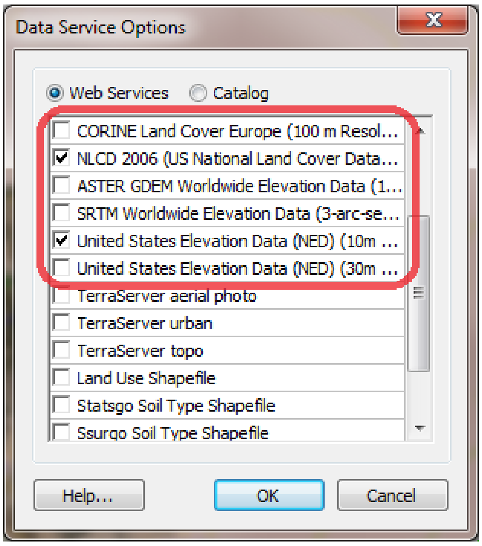


The online image capabilities in WMS 9.0 allow you to view and use a map showing any part of the world with your data.



The online image capabilities in WMS 9.0 allow you to download and use georeferenced images for any part of the world with your data.

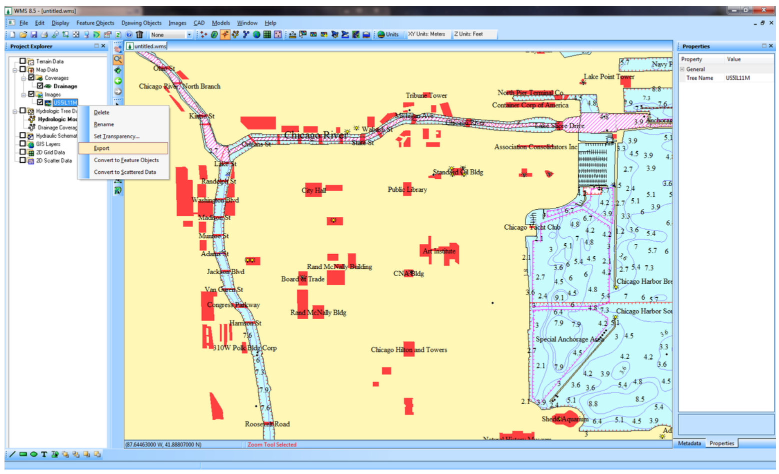
Improved Web Service Tools



Six new web service data sources are available in WMS 9.0. These services allow you to download data for many parts of the world.

Six new web services have been added to the list of web data that can be downloaded from the WMS Get Data Toolbar  . These web services include some that were previously available, such as United States National Elevation Datasets, as well as newly available datasets, such as the CORINE European land cover database, the NLCD US National Land Cover database, and the ASTER Worldwide Elevation Data database. These new datasets are much faster than previously available datasets and include a progress bar so you can view the progress of your data download.

Support of New File Formats



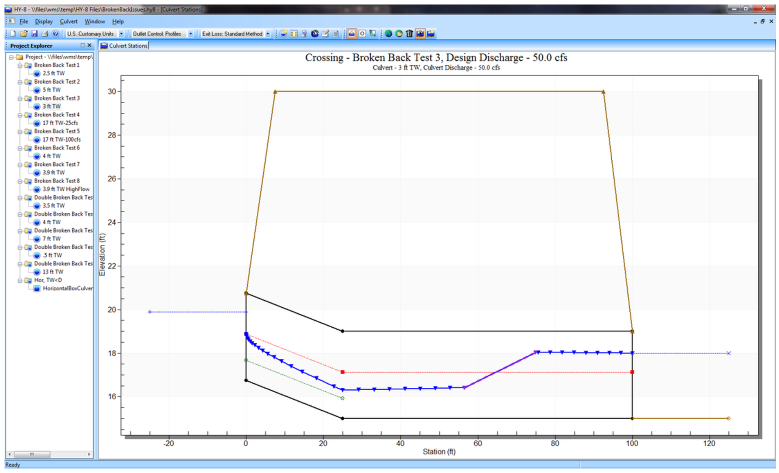
You can read and display almost any commonly used vector data source as a vector image in WMS.

We have added support for almost all the commonly used vector file formats in the new version of WMS. Some vector files, such as DXF, DWG, and ESRI Shapefile format, still read the way they have always read into WMS. But support for "Vector-based images" has been added to WMS that allows you to read any file that can be read using the Global Mapper software in WMS (A license to Global Mapper is not required). For a complete listing of the vector formats supported in WMS, visit the Global Mapper web site [1]. Right-clicking on a vector image allows you to export to one of many formats or convert linear data to feature objects or scattered data (XYZ). The following formats can be exported using the right-click command:

- DXF Files (*.dxf)
- Area Shapefiles (*.shp)
- Line Shapefiles (*.shp)
- Point Shapefiles (*.shp)
- Google Earth KMZ Files (*.kmz)
- MapInfo MIF/MID Files (*.mif)
- MapInfo TAB/MAP Files (*.map)
- Simple ASCII Text Files (*.txt)
- CSV (Comma-separated value) Files (*.csv)

- SVG Files (*.svg) [1]

FHWA HY-8 7.3 and Hydraulic Toolbox 3.0 Support



HY-8 7.3 is supported in the latest WMS.

The FHWA's HY-8 7.3, which includes many new culvert modeling capabilities, is supported in the new version of WMS. The new version of the FHWA's Hydraulic Toolbox (3.0) is also supported with the new version of WMS. The new tools in HY-8 7.3 include modeling of hydraulic jump profiles, broken back culverts (culverts with a change in slope), and horizontal and adverse slopes in culverts. Documentation showing the capabilities included in the latest version of HY-8 is included on the HY-8 wiki. The newest version of the hydraulic toolbox includes tools for culvert assessment and for determining a riprap or streambed gradation curve using a digital image.

Bug fixes

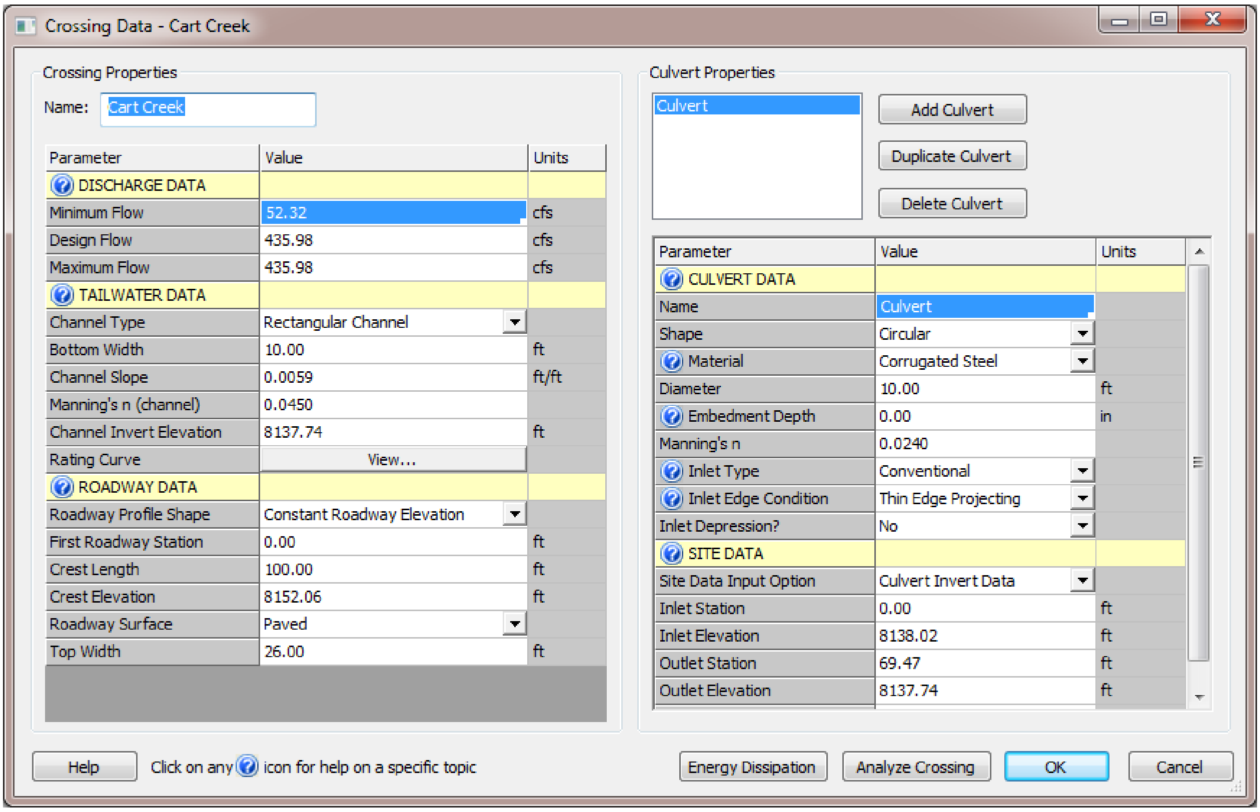
To view the list of bugs fixed in WMS 9.0, visit the WMS bugfix page.

7.1. WMS to HY-8

Path of Data from WMS To HY8

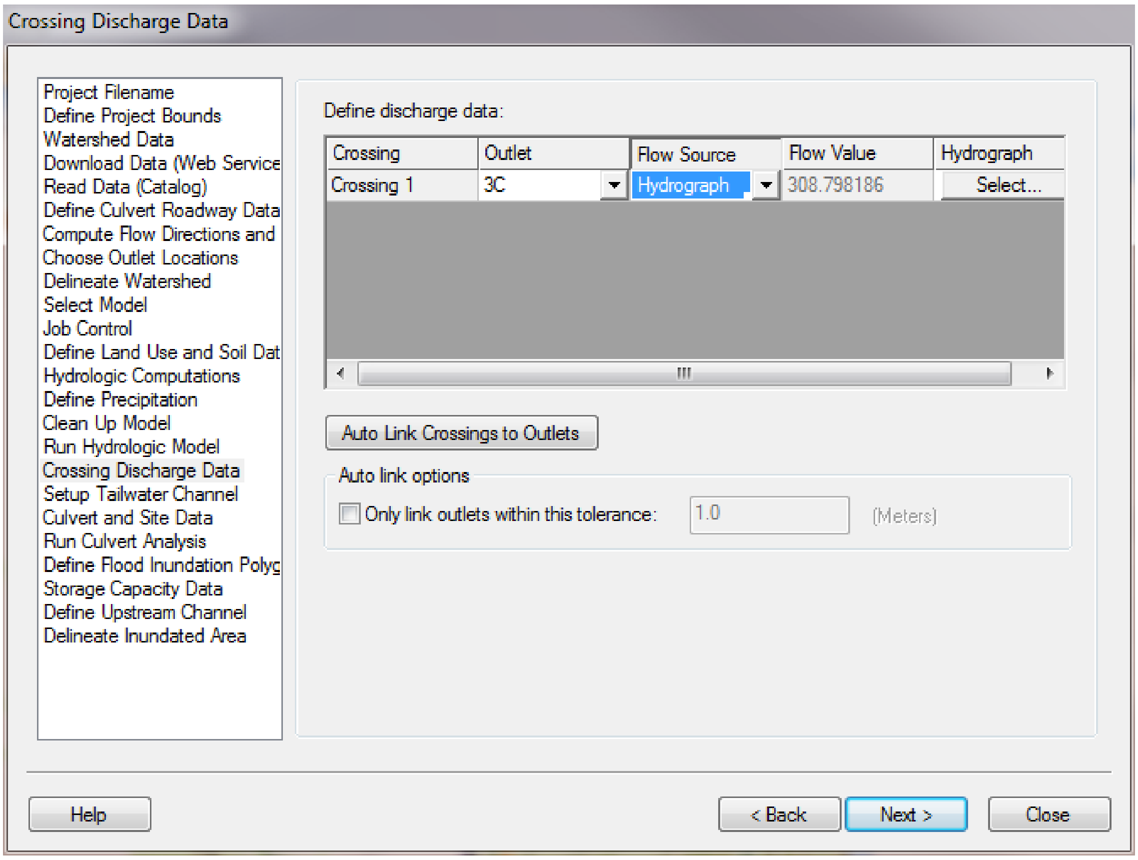
The Path of Data from WMS to HY8 in the HY8 Modeling Wizard

The HY8 Modeling Wizard in WMS gathers data to setup an HY8 model. Some data can be changed at multiple locations, and all variables can be changed in the HY-8 model when the model is setup. This data is categorized as follows: Discharge Data, Tailwater Data, Roadway Data, Culvert Data, and Site Data. These categories can be seen in the HY-8 interface.



Discharge Data

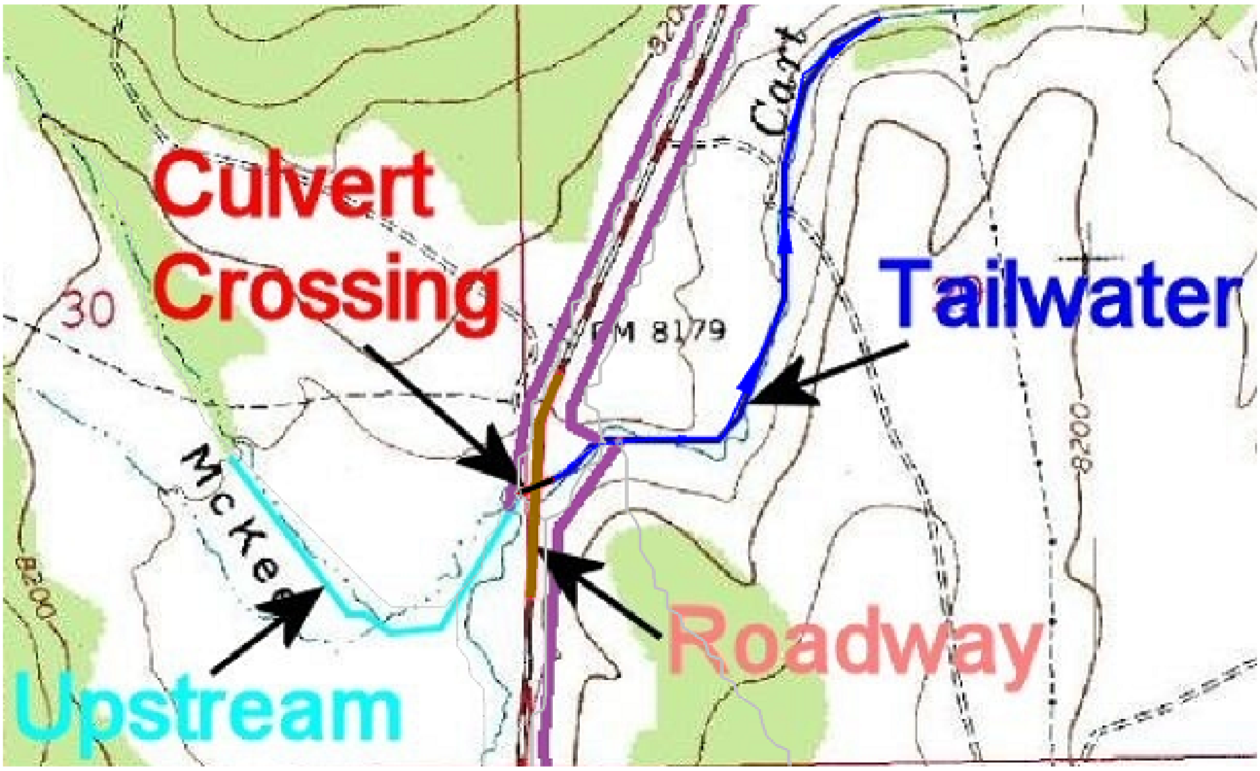
This information is pulled from the HY-8 Wizard on step Crossing Discharge Data.



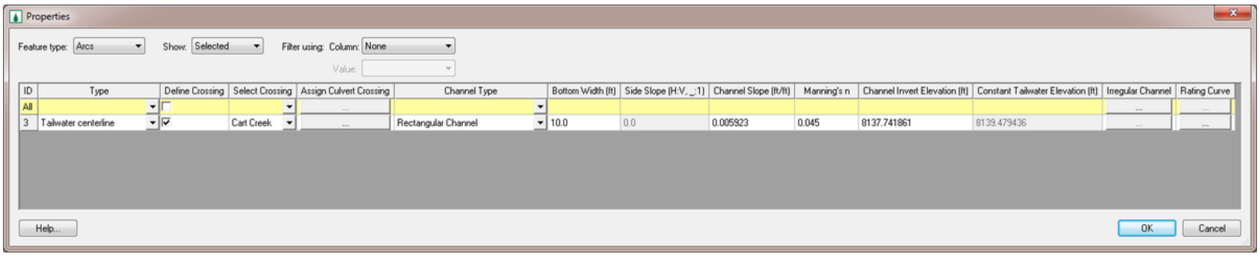
If the user specifies the flow source as 'Design Flow', the minimum flow value will be zero and the design and maximum flow value will be the value entered by the user. If the user specifies the flow source as 'Hydrograph', the minimum flow in the hydrograph will be the minimum flow value in HY-8, while the maximum flow in the hydrograph will be both the design and maximum flow value in HY-8. Finally, if the user specifies 'Design Flow & Hydrograph' as the flow source, WMS will use the minimum and maximum flow values from the hydrograph and allow the user to enter the design value.

Tailwater Data

The information for the tailwater is saved in the tailwater arc.

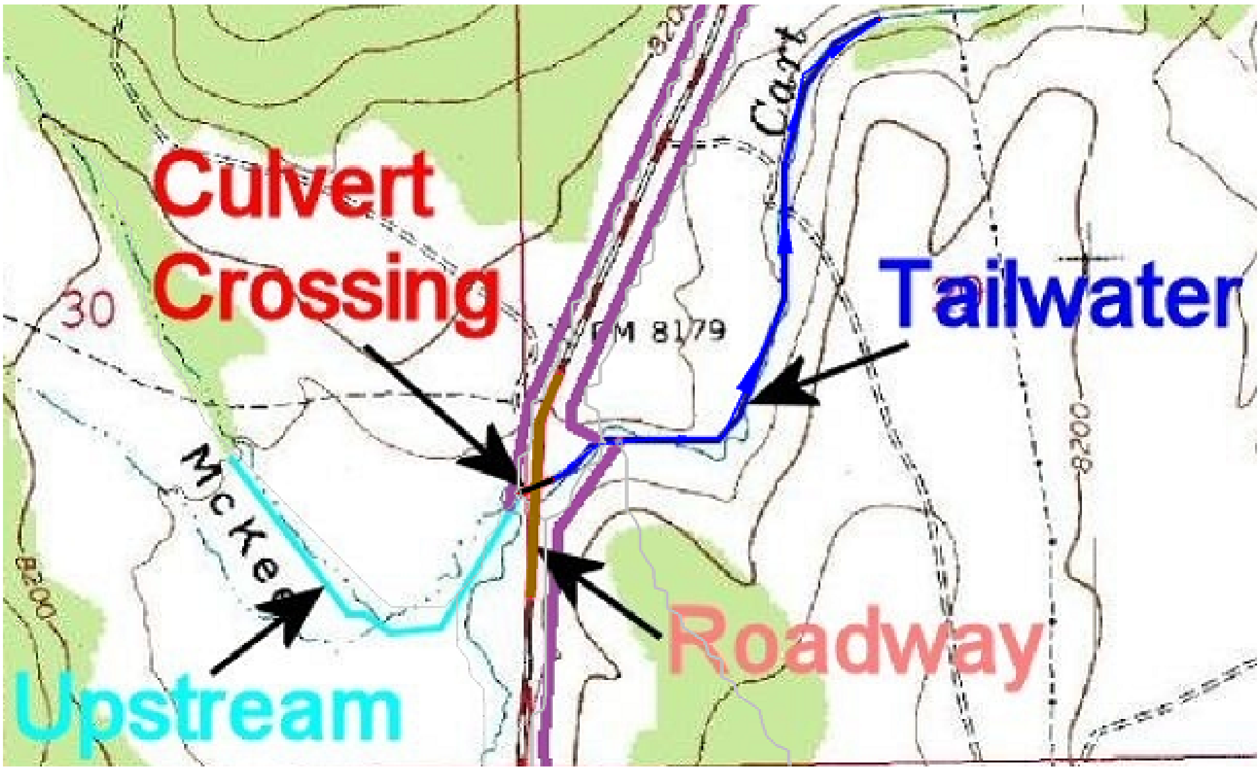


The 'Channel Slope' is determined from the slope of the tailwater arc. The 'Channel Invert Elevation' is initialized with the value of the elevation at upstream end of the tailwater arc.

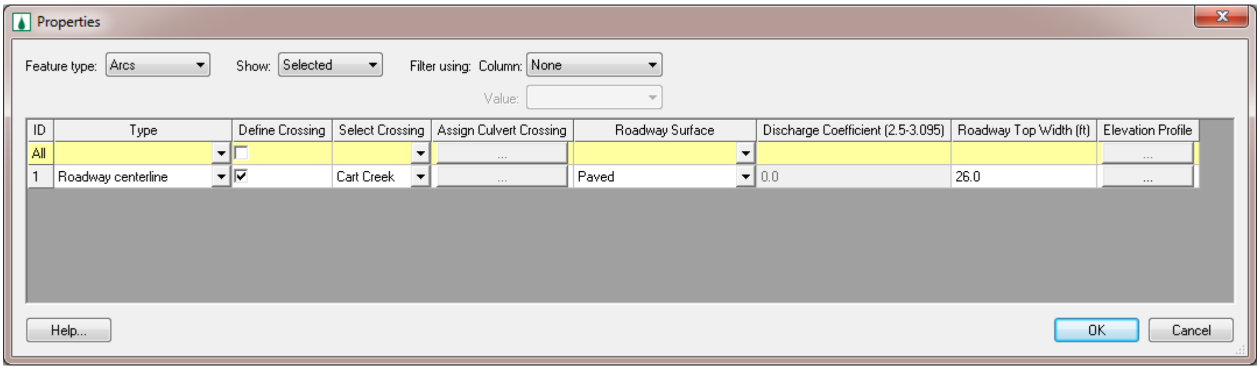


The 'Channel Type', 'Bottom Width', 'Side Slope', 'Manning's n', 'Constant Tailwater Elevations', 'Irregular Channel Cross-Sections', and 'Rating Curves' are set in the attribute table for the arc.

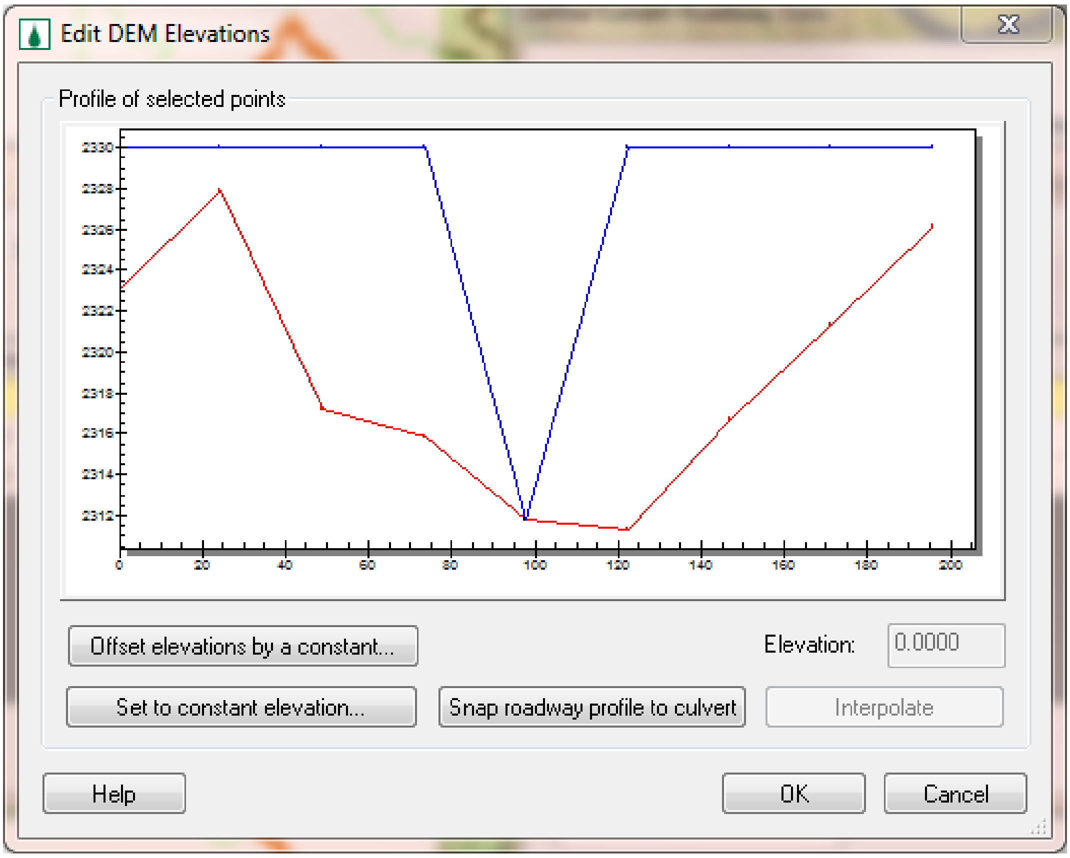
Roadway Data



The length of the roadway, or 'Crest Length', is determined from the length of the tailwater arc.



The 'Roadway Surface', 'Discharge coefficient', 'Roadway Top Width', and 'Elevation Profile'. The elevation profile will set whether the roadway has a constant roadway elevation or if it is an irregular shape.



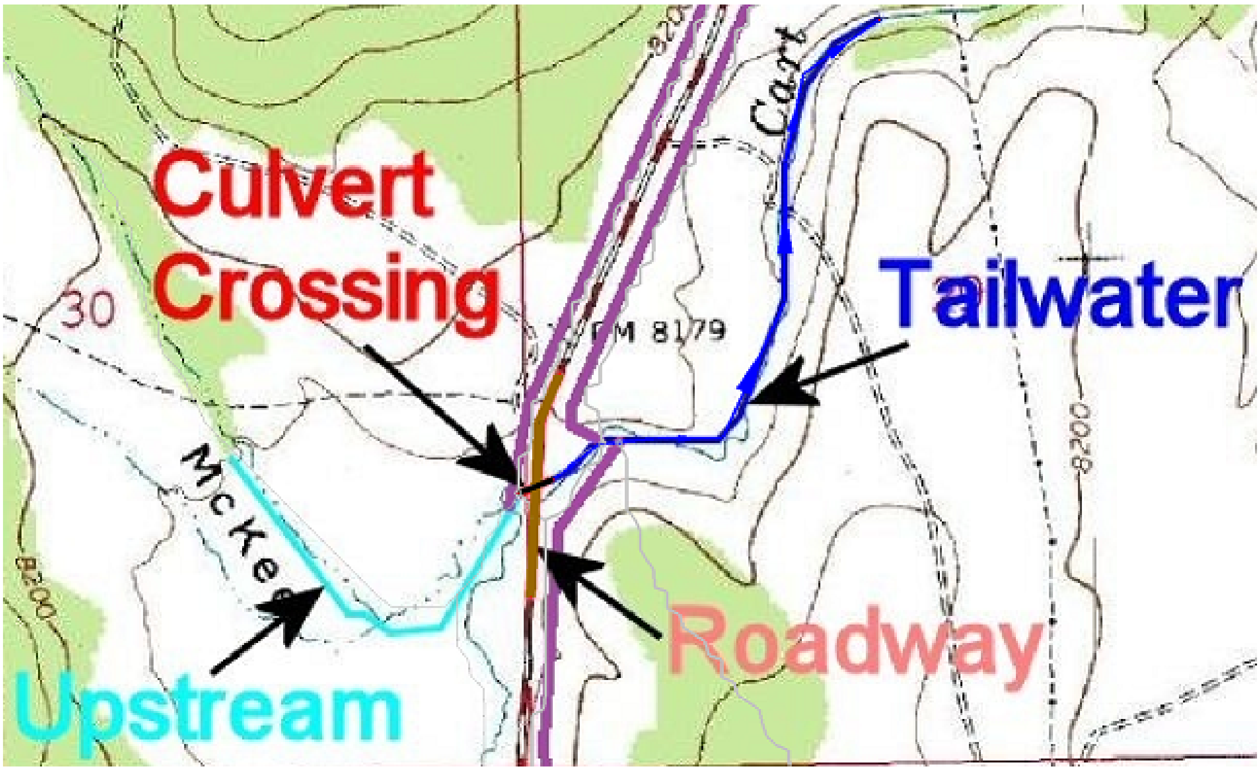
If the user clicks to edit the elevation profile, the user will see a plot with a red line and a blue line. The red line represents the elevation of the DEM while the blue line represents the finished roadway elevation. When the user clicks okay on this dialog, WMS will change the DEM under the roadway arc to match the finished roadway. For this reason, the user must leave one point lower, preferably at the culvert invert elevation, and at the same location as the culvert crossing. When Topaz determines flow directions and accumulations, the water will be able to route through the roadway at the proper location. When WMS brings the roadway profile to HY8, it will remove the lowest point and then remove any points that are unnecessary (if there are three or more points with the same elevation, it will remove all but the points at the end of the group that are the same elevation). HY8 requires that the irregular roadway has between 3 and 15 points. If the profile has 2 points after removing the lowest and unnecessary points, WMS will assign it to be a constant roadway elevation. Otherwise it will assign it to be an irregular roadway profile. If the profile has more than 15 points, WMS will remove the least significant points until it has only 15 points remaining.

Culvert Data

There is no culvert data that is set in WMS. The user must enter all culvert data in the HY-8 interface.

Site Data

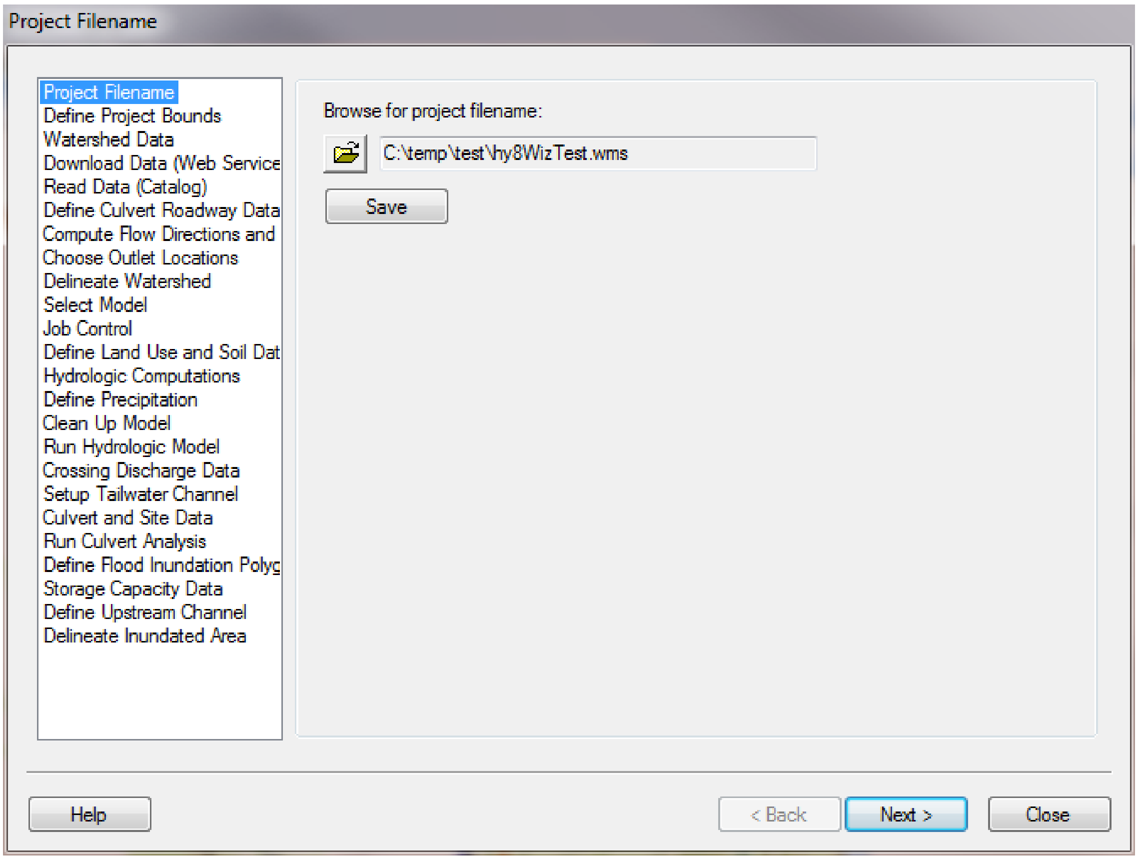
Some of the parameters for the site data is saved in the culvert crossing arc and some of the parameters are not changed by WMS.

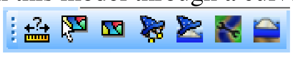


WMS will take the length of the culvert crossing arc and set the 'Inlet station' to zero and the length to the 'Outlet Station'. The elevations are then set to the elevations of the DEM at those locations of the culvert crossing arc. The 'Number of Barrels' and all of the parameters for the 'Embankment Toe Data' option are not handled by the WMS interface and must be updated in the HY-8 interface.

HY8 Modeling Wizard

Overview



The WMS HY-8 Modeling Wizard () is a simple tool that walks you through all the steps involved in creating a hydrologic model and routing the results from this model through a culvert. It can be accessed by selecting the HY-8 modeling wizard tool in the get data toolbar ().

Steps

The following steps are included in the HY-8 modeling wizard:

1. Project Filename
2. Define Project Bounds
3. Watershed Data
4. Download Data (Web Service Client)
5. Read Data (Catalog)
6. Define Culvert Roadway Data
7. Compute Flow Directions and Flow Accumulations
8. Choose Outlet Locations
9. Delineate Watershed
10. Select Model
11. Job Control
12. Define Land Use and Soil Data
13. Hydrologic Computations
14. Define Precipitation

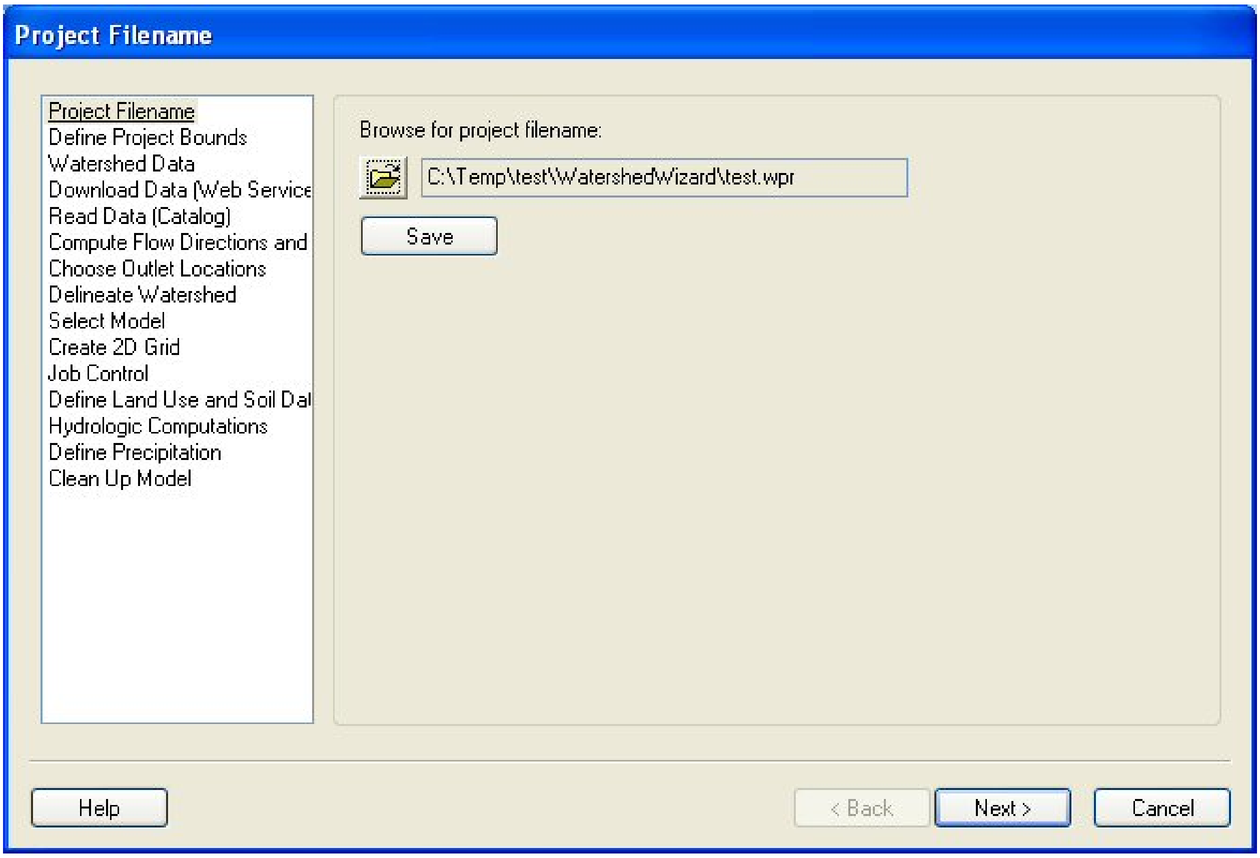
15. Clean Up Model
16. Run Hydrologic Model
17. Crossing Discharge Data
18. Setup Tailwater Channel
19. Culvert and Site Data
20. Run Culvert Analysis
21. Define Flood Inundation Polygon
22. Storage Capacity Data
23. Define Upstream Channel
24. Delineate Inundated Area

General Information

If you would like to see where each variable for the HY-8 model originates in the WMS HY-8 modeling wizard, please read the following page: Path of Data from WMS To HY8.

Project Filename

Overview



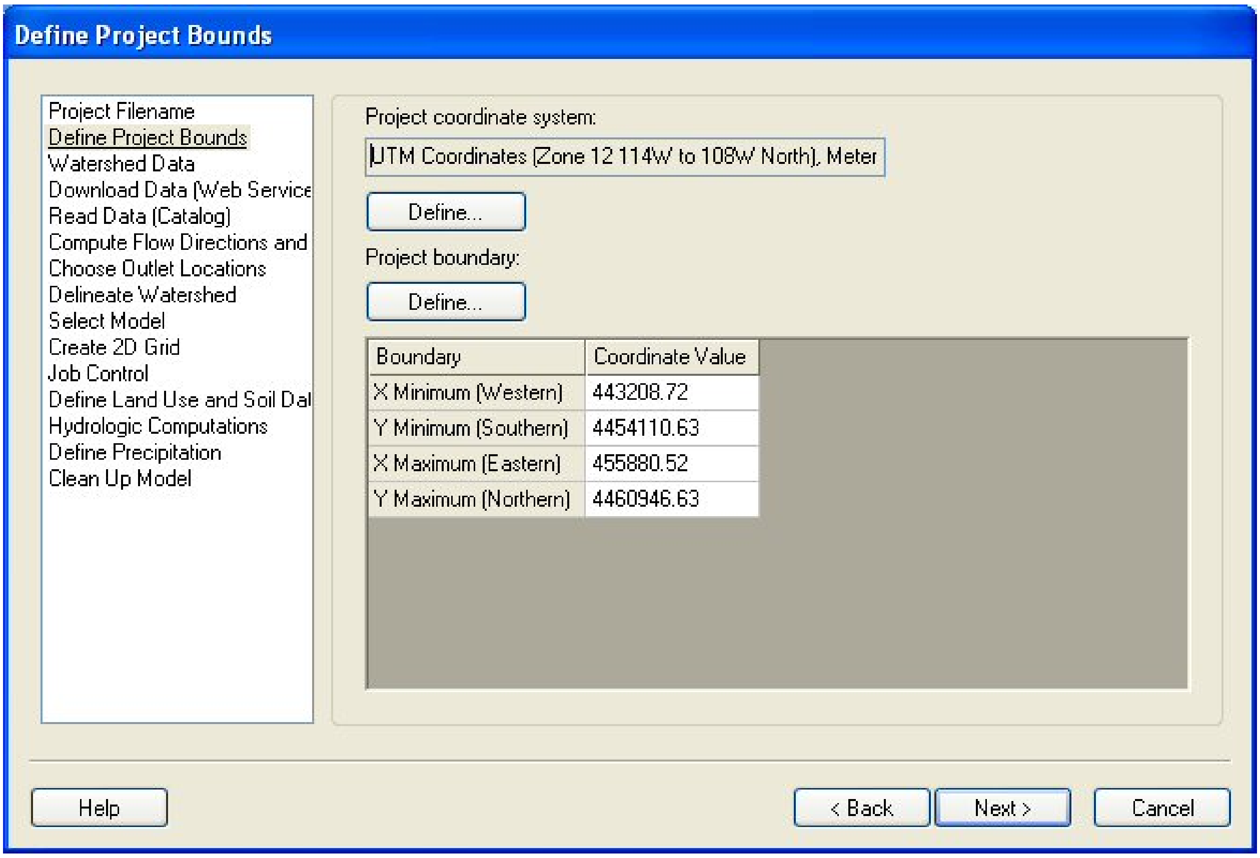
The project filename step is used for defining a project filename. This filename is used when saving files from the hydrologic modeling wizard.

Help

Save – This button saves the project in your current state.

Define Project Bounds

Overview



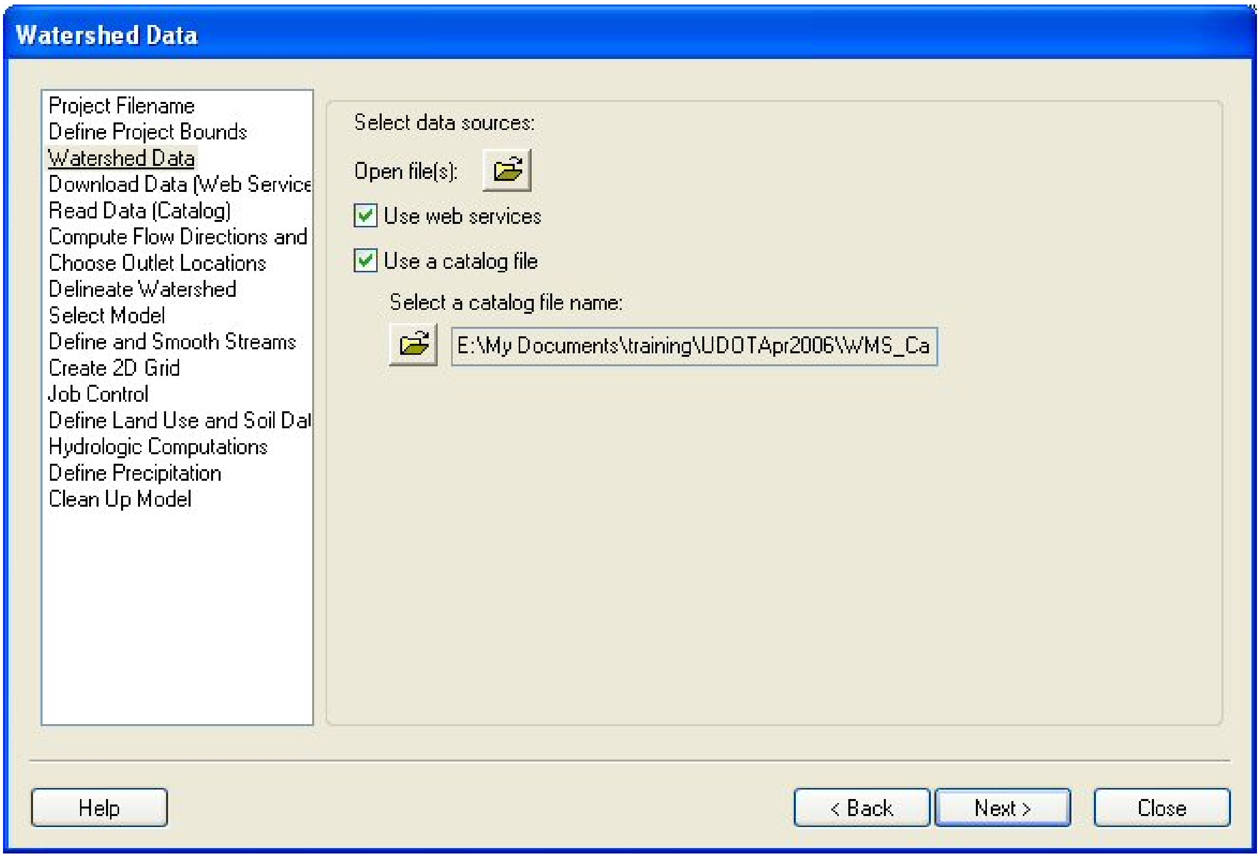
The define project bounds step is used for defining your project boundary.

Help

Top **Define** button - This button is used to define your project coordinate system, if it is known. Bottom **Define** button - This button is used to define your project boundary in the Microsoft Virtual Earth web service client. Find the area you are interested in modeling and WMS will enter the minimum and maximum coordinates of the box you have defined in the Virtual Earth window.

Watershed Data

Overview



The watershed data step is used for defining how you will read data into WMS. You can obtain data using the WMS web service client, a catalog file, or by simply opening files.

Help

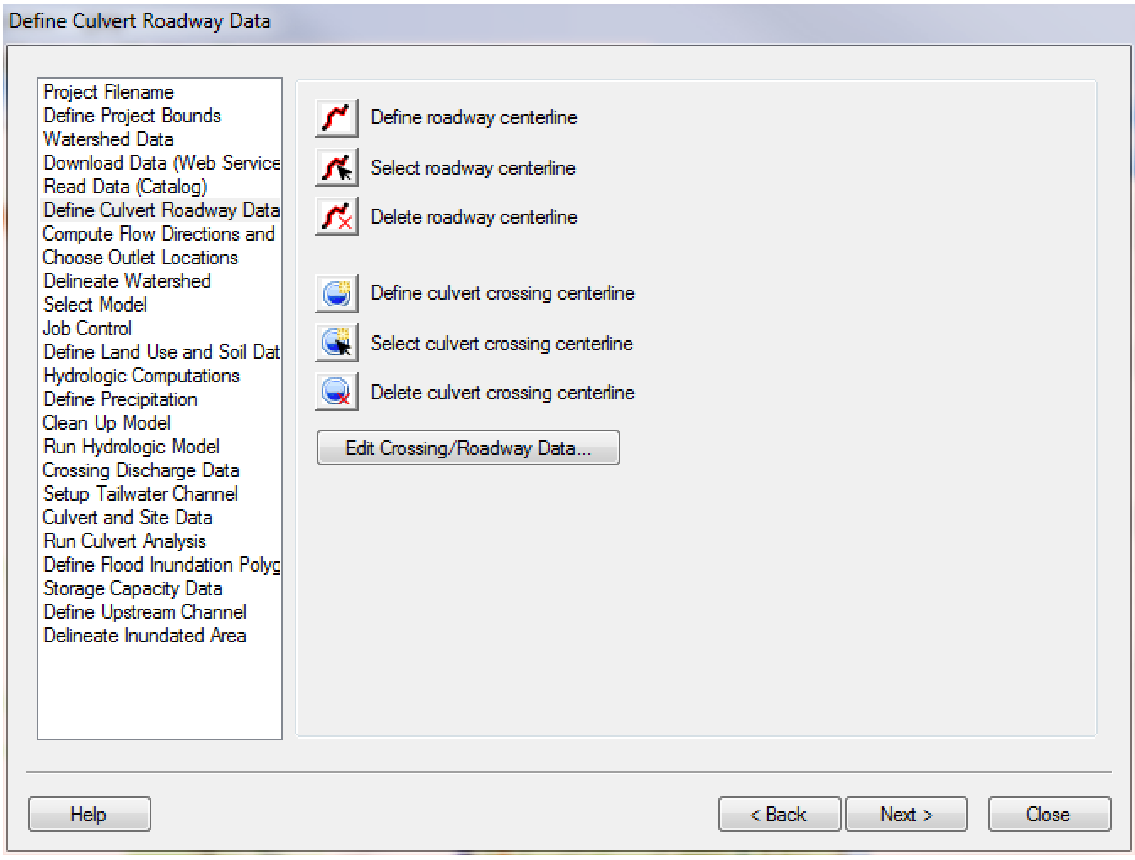
Open file button - Use this button in the same way as the File | Open command in the WMS menus. After selecting this button, a file browser appears and WMS opens the files you select.

Web services - WMS will use the built-in web service client for obtaining data for watershed modeling.

Catalog file - WMS will use a catalog file for obtaining data for watershed modeling.

Define Culvert Roadway Data

Overview



The Define Culvert Roadway Data step is used to define the culvert and roadway centerlines as arcs and the data associated with these structures.

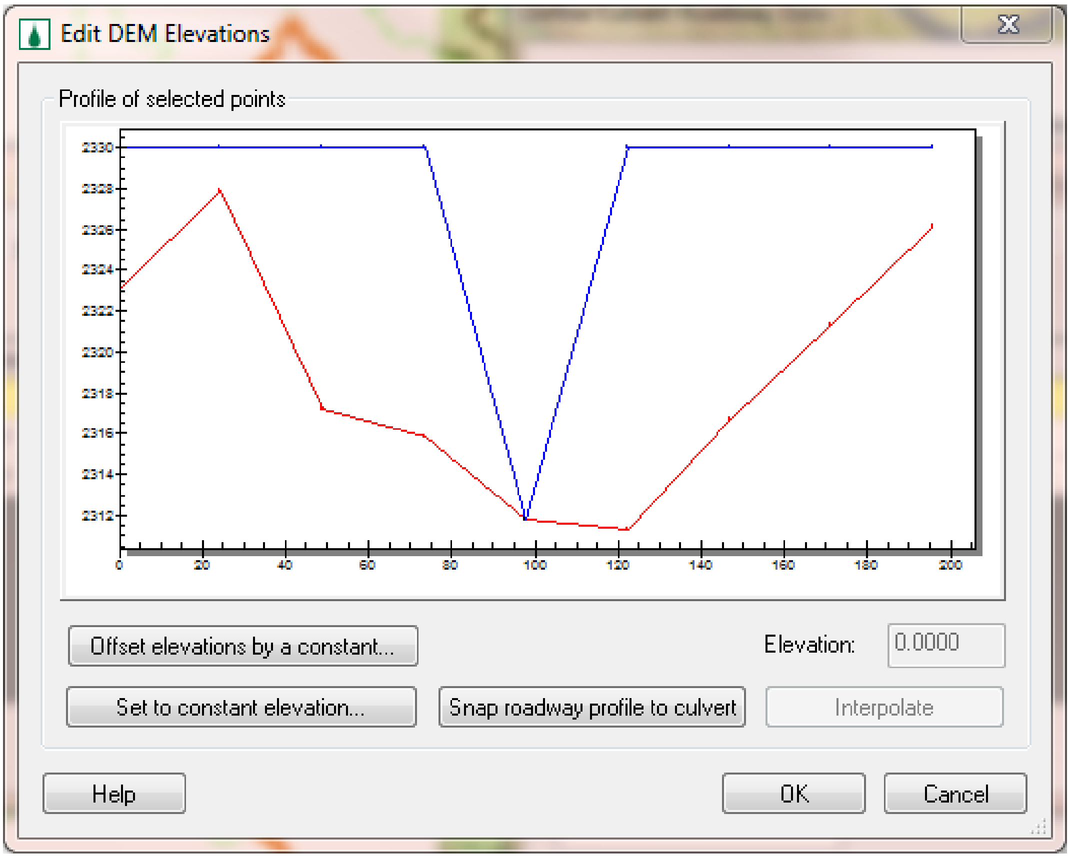
Help

When clicking on the define roadway centerline button, WMS either gets the HY8 coverage and makes it the active coverage or creates a new HY8 coverage if one does not exist and makes this the active coverage. Any roadway centerline arcs should be created in the HY8 coverage. Separate display options exist for roadway arcs, upstream arcs, downstream (tailwater) arcs, culvert arcs and flood inundation polygons.

Selecting the define culvert crossing centerline button works in much the same way as the define roadway centerline button, except a culvert arc is created when the user clicks in the graphics window. Culverts and all water conveyance arcs in the HY-8 coverage should be created from upstream to downstream, and an arrow shows the flow direction of each type of arc.

If no roadway arcs are selected, clicking on the edit crossing/roadway data button selects all the roadway arcs in the HY8 coverage and brings up the arc attributes for these arcs. If roadway arcs are selected, only the selected roadway arc attributes are shown. The culvert and roadway arc attributes are shown in a spreadsheet window.

For the roadway attributes, a button exists that brings up the roadway elevation profile, as shown here:

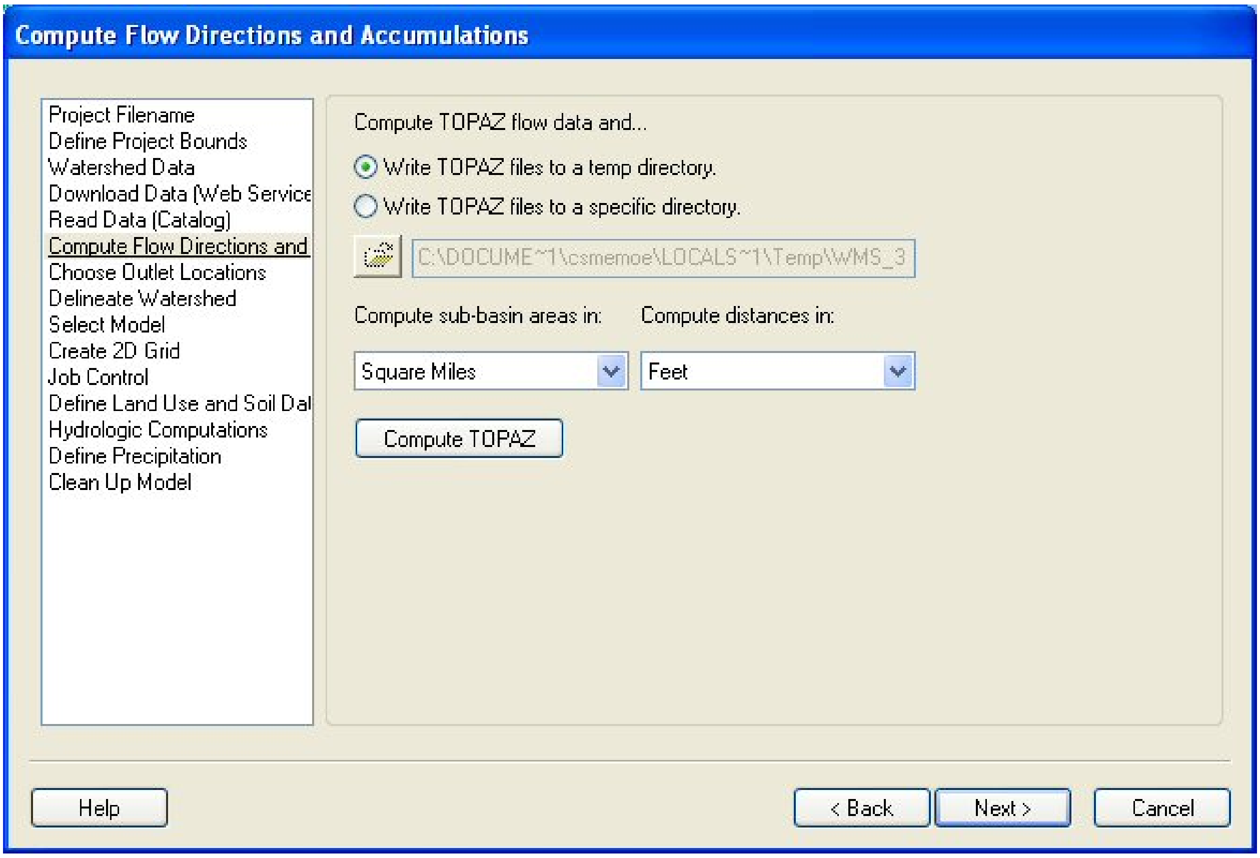


This dialog works in much the same way as the Edit Elevations dialog in the DEM menu. If the Snap roadway profile to culvert button is selected, WMS determines the intersection of the roadway and the culvert and returns this elevation to the original DEM elevation.

For more information on how WMS uses this data to set the HY8 Roadway Data, please see: Path of Data from WMS to HY8

Compute Flow Directions and Flow Accumulations

Overview



The Compute Flow Directions and Flow Accumulations step is used for running TOPAZ to compute flow directions and accumulations.

Help

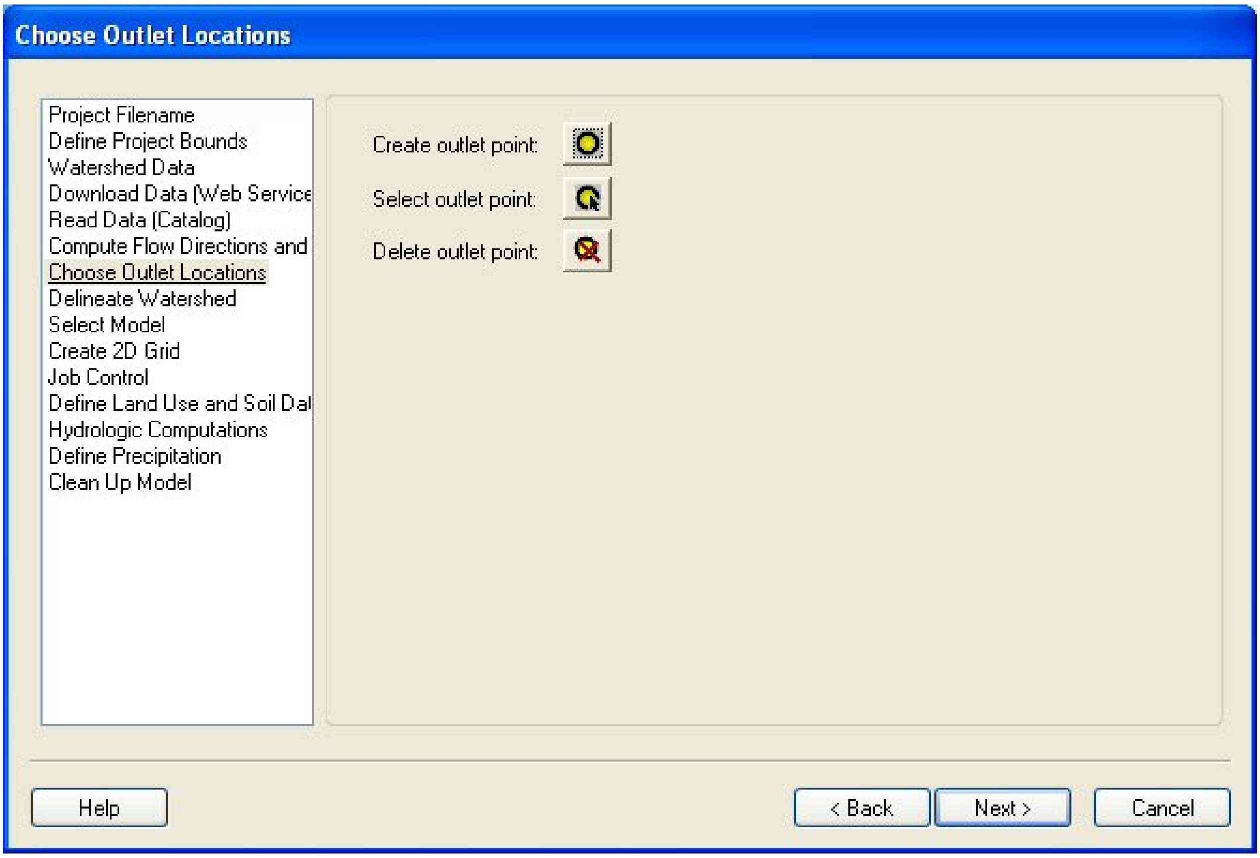
Compute TOPAZ flow data and... - You can choose either to write your TOPAZ output to a specific directory or to the temp directory. Most frequently, you will just write your TOPAZ output to a temp directory.

Units - Use the units combo boxes to define your model's computation units.

Compute Topaz - This button runs TOPAZ using the settings you have selected.

Choose Outlet Locations

Overview



The Choose Outlet Locations step is used to define, move, or delete outlet locations.

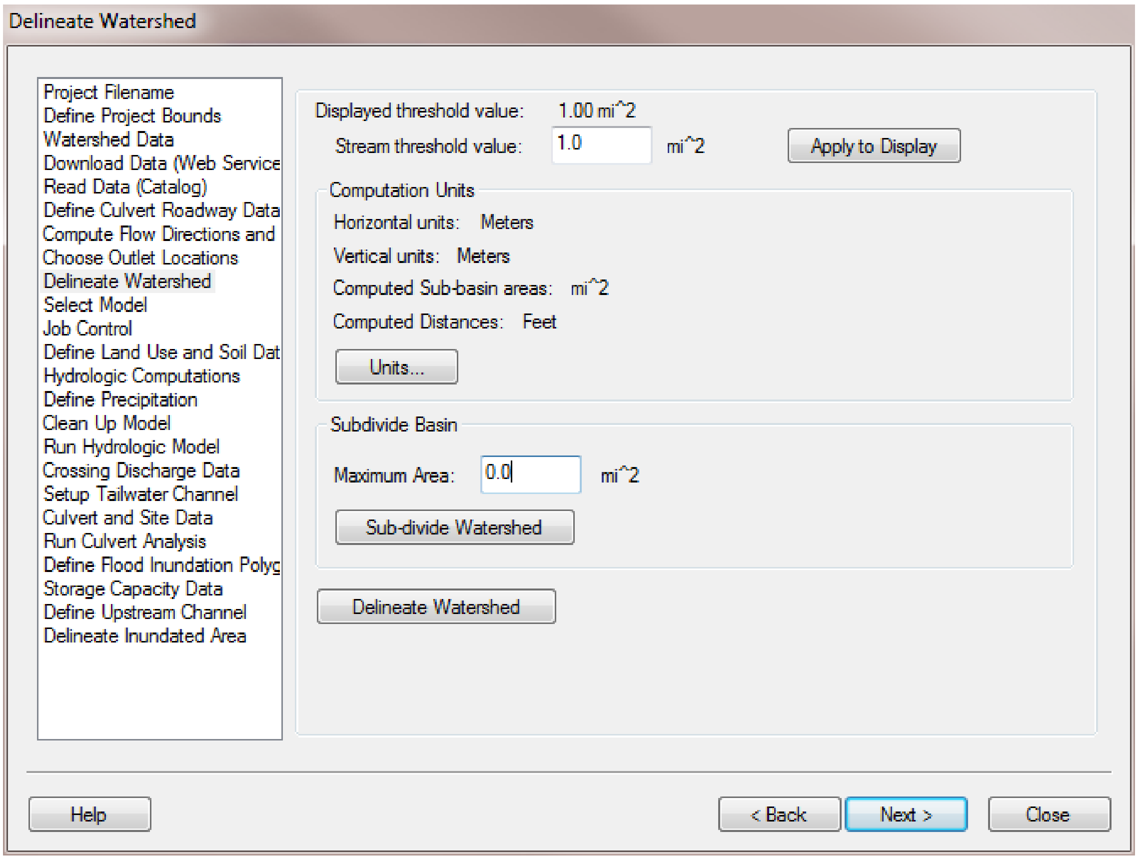
Help

Create, **Select**, and **Delete** outlet point buttons - To create an outlet point, select the tool you want to use and then click on a flow accumulation cell. To select an outlet point, select the tool and click on a point or drag a point. To delete a point, select a tool and select an outlet point to delete.

In the HY-8 Modeling Wizard, this dialog has a **Define Outlets from Culvert Locations** button. This button defines an outlet at the point on a flow accumulation cell that is closest to the upstream end of the culvert arc.

Delineate Watershed

Overview



The Delineate Watershed step is used to delineate a watershed.

Help

Stream threshold value - This value is used to modify the stream density. Lower values will cause the streams to be more dense while higher values will create fewer streams in the completed model.

Apply to Display - Select this button to apply the stream threshold value entered to the display.

Create Tc Coverage - If this toggle is selected, a Time Computation coverage containing an arc with the longest flow path is created after delineating the watershed and sub-basins.

Units - This button is used to define your model and computation coordinates and units. You can turn on the option to create a Tc coverage and change which data is computed by selecting the *Drain Data Compute Opts...* button in the Units dialog.

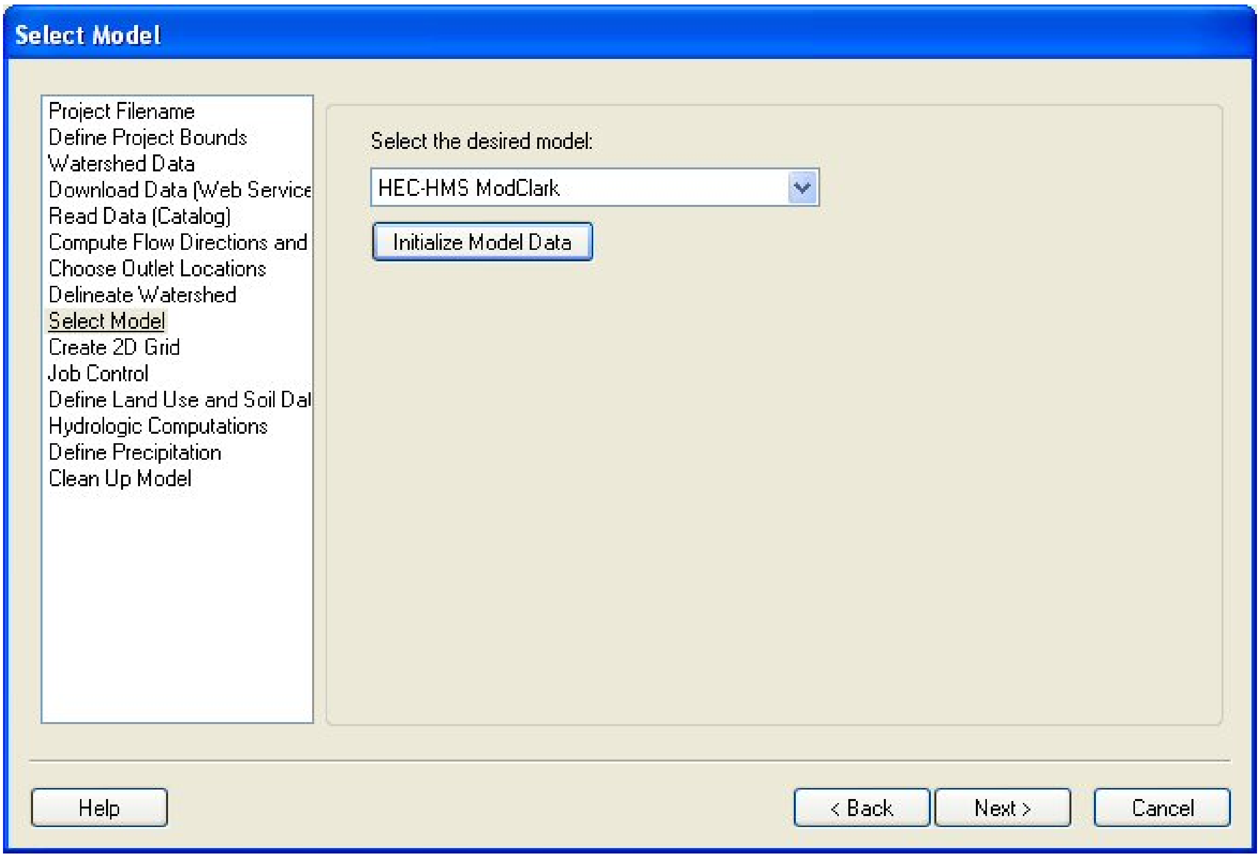
Sub-divide Watershed - This button subdivides your watershed into sub-basins based on the maximum sub-basin area entered.

Delineate Watershed - This button delineates the watershed and computes each sub-basin's data based on the selected watershed delineation parameters.

After delineating a watershed, it is possible to manually edit the extents of the watershed.

Select Model

Overview



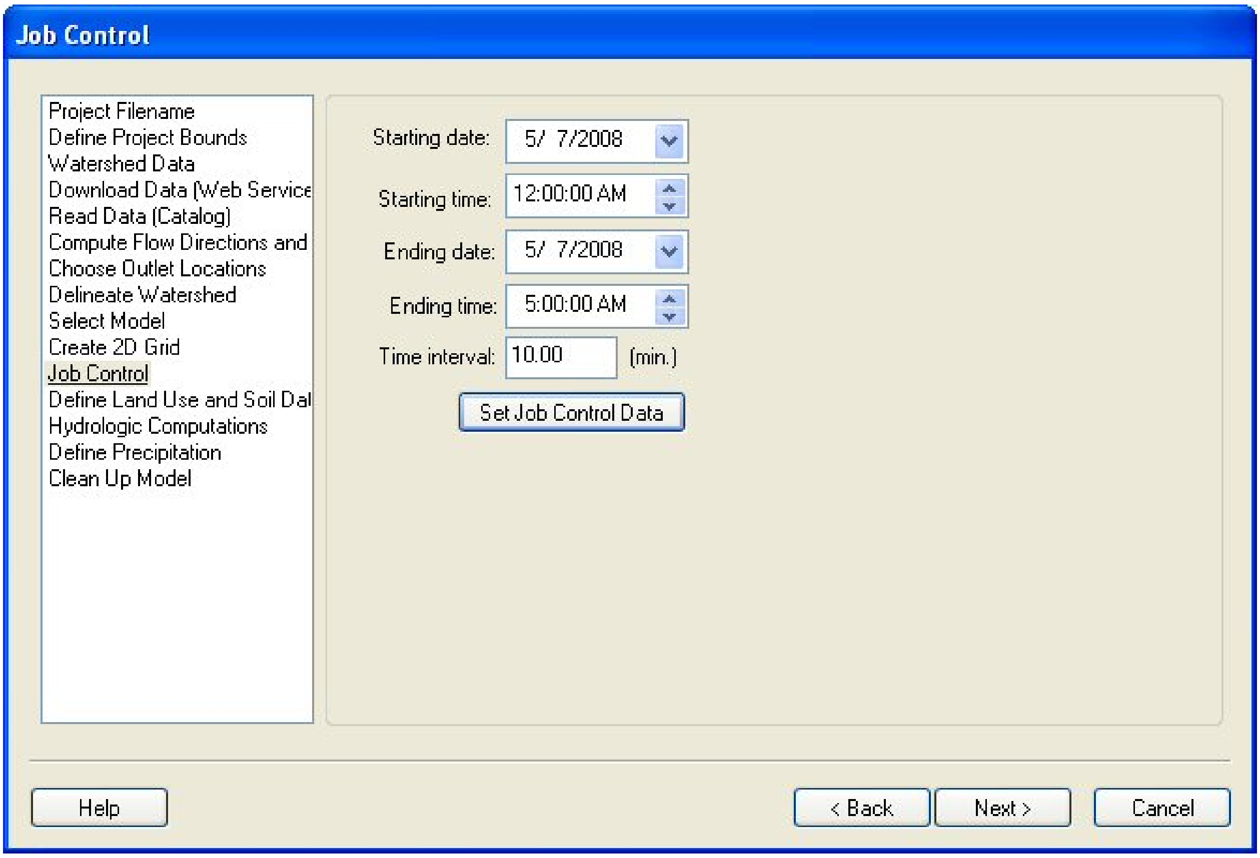
The Select Model step is used for defining which model you are running with your delineated watershed.

Help

Initialize Model Data - Click on this button to set your model to the selected option, initialize your model data to default values, and set the module to the correct module.

HMW Job Control

Overview



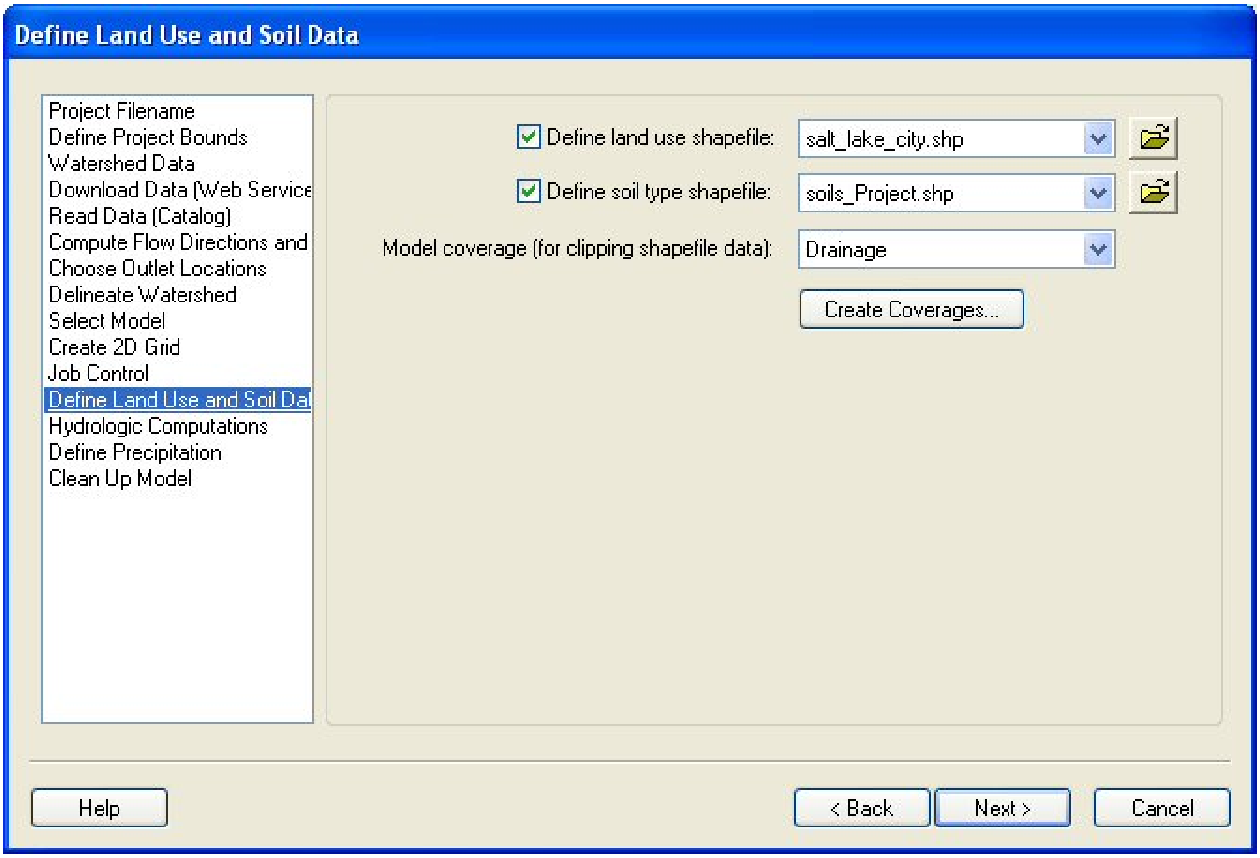
The job control step is used define the time parameters for running your model. Define the start and end time and date and select **Set Job Control Data** to set your job control parameters for the model you have selected.

Help

Set Job Control Data - Sets your job control parameters for the model you have selected.

Define Land Use and Soil Data

Overview



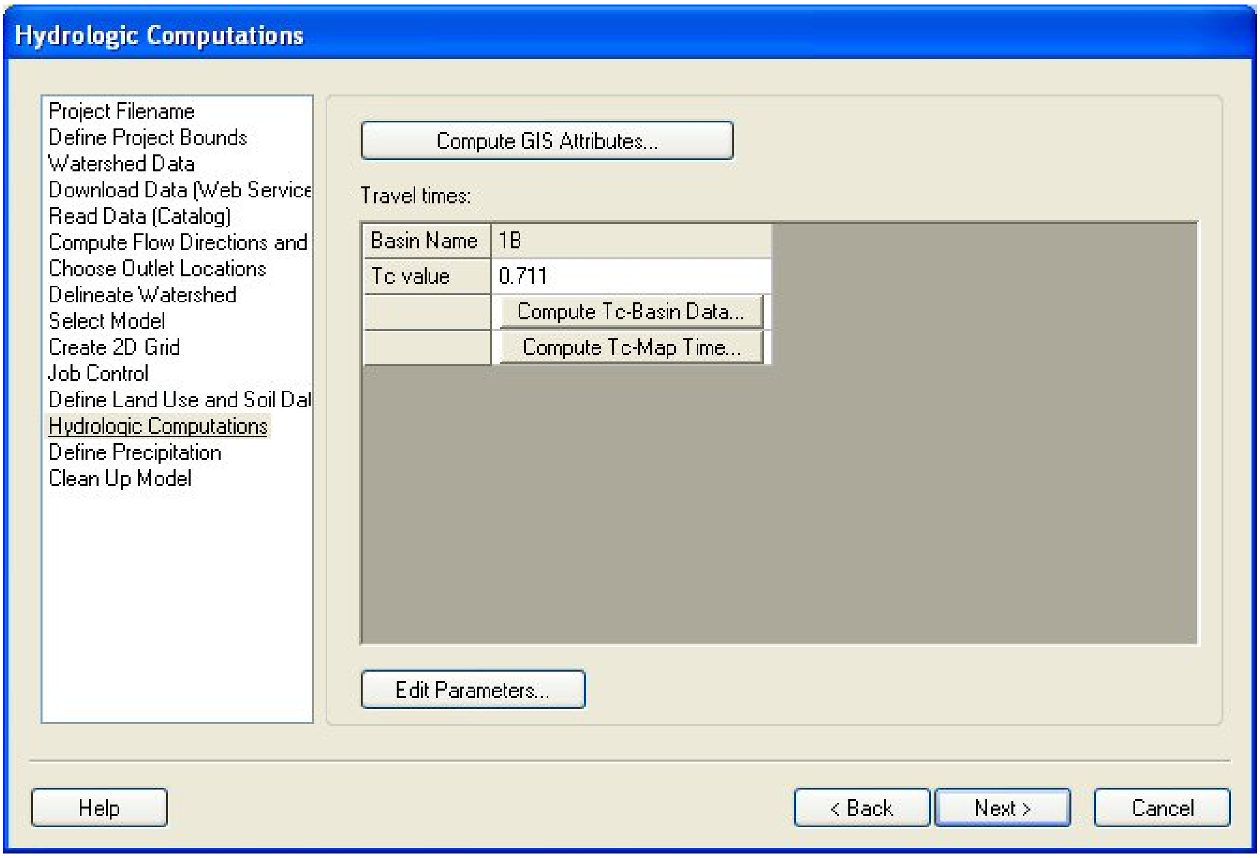
The Define Land Use and Soil Data step converts GIS Module shapefiles to data in the WMS Map module. WMS uses the boundary of your watershed to clip the shapefile data for the selected files.

Help

Create Coverages - Define your land use and soil type shapefiles and a model coverage and your land use/soil data will be transferred to coverages in the map module and clipped to your watershed boundary when you select this button.

Hydrologic Computations

Overview



The Hydrologic Computations step is used to compute hydrologic parameters for your watershed and sub-basins.

Help

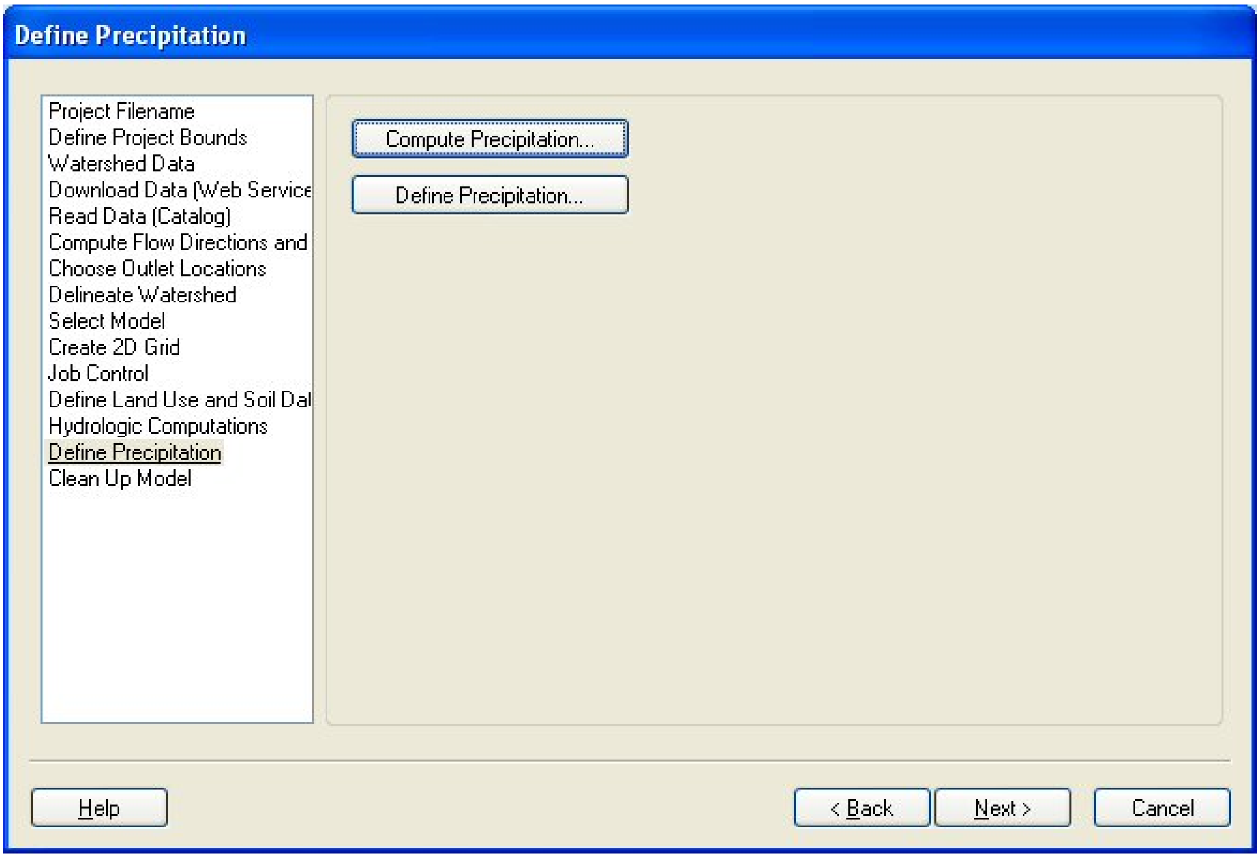
Compute GIS Attributes - Depending on the model you are using, this button brings up a dialog allowing you to compute attributes from your land use and/or soil type data.

Travel times - This spreadsheet allows you to compute the Time of Concentration for all the sub-basins in your watershed using either the basin data or map data method.

Edit Parameters - This button brings up the edit parameters dialog for your model, allowing you to change modeling parameters for all the sub-basins in your watershed.

Define Precipitation

Overview



The Define Precipitation step is used to compute and define precipitation for your model.

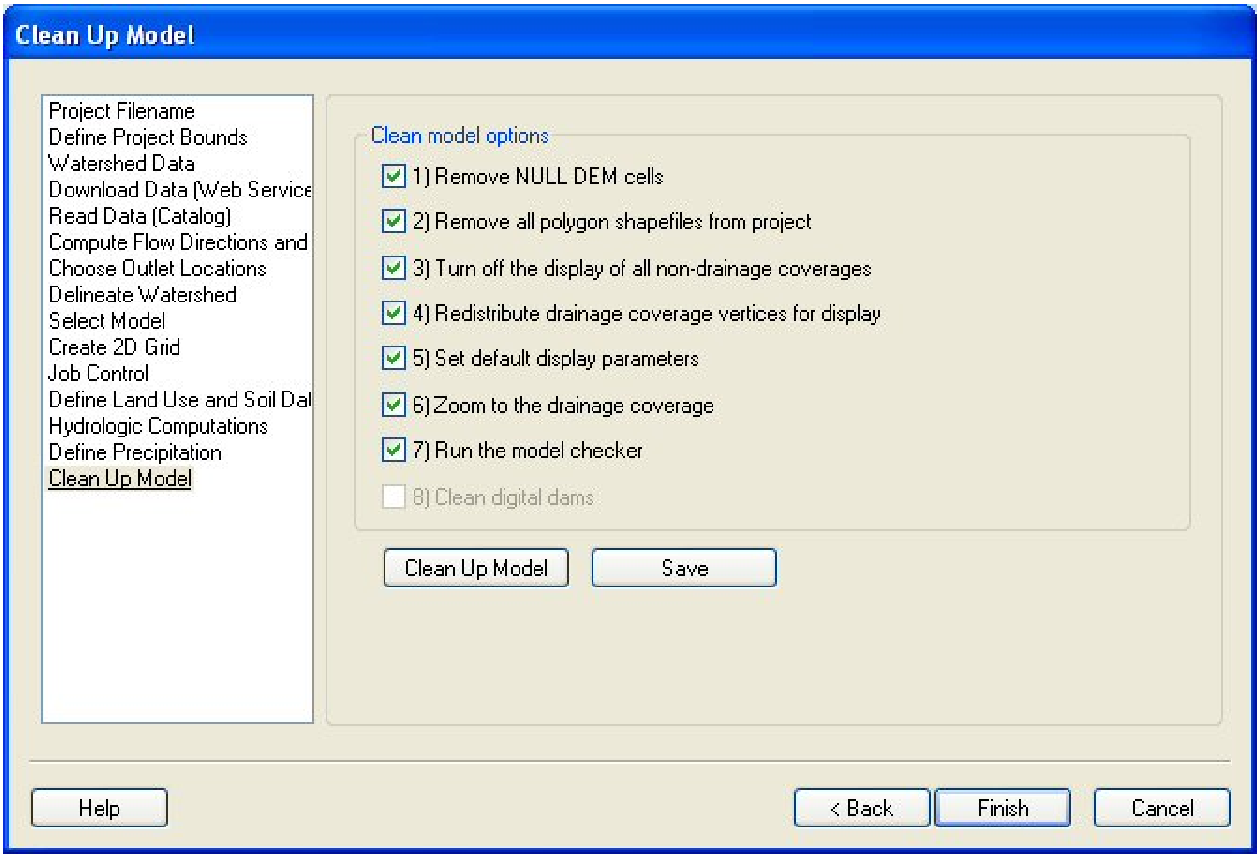
Help

Compute Precipitation (certain models only) - This button allows you to compute the precipitation for your model from a NOAA Atlas 2 or any other type of rainfall grid.

Define Precipitation - This button allows you to define precipitation for your selected model.

Clean Up Model

Overview



The Clean Up Model step is used to clean up your model by doing tasks that are typically done when your model is finished.

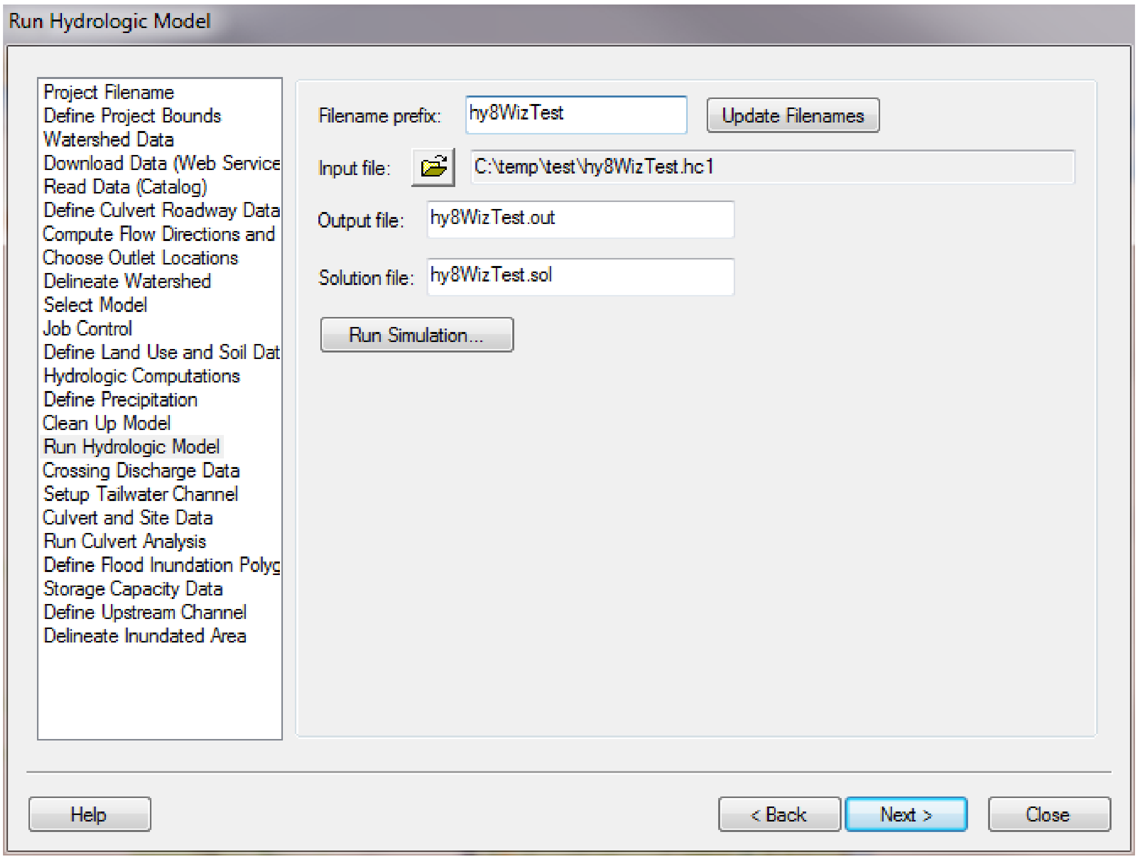
Help

Clean Up Model - This button runs only the selected tasks listed above the button.

Save - This button saves the project in your current state.

Run Hydrologic Model

Overview



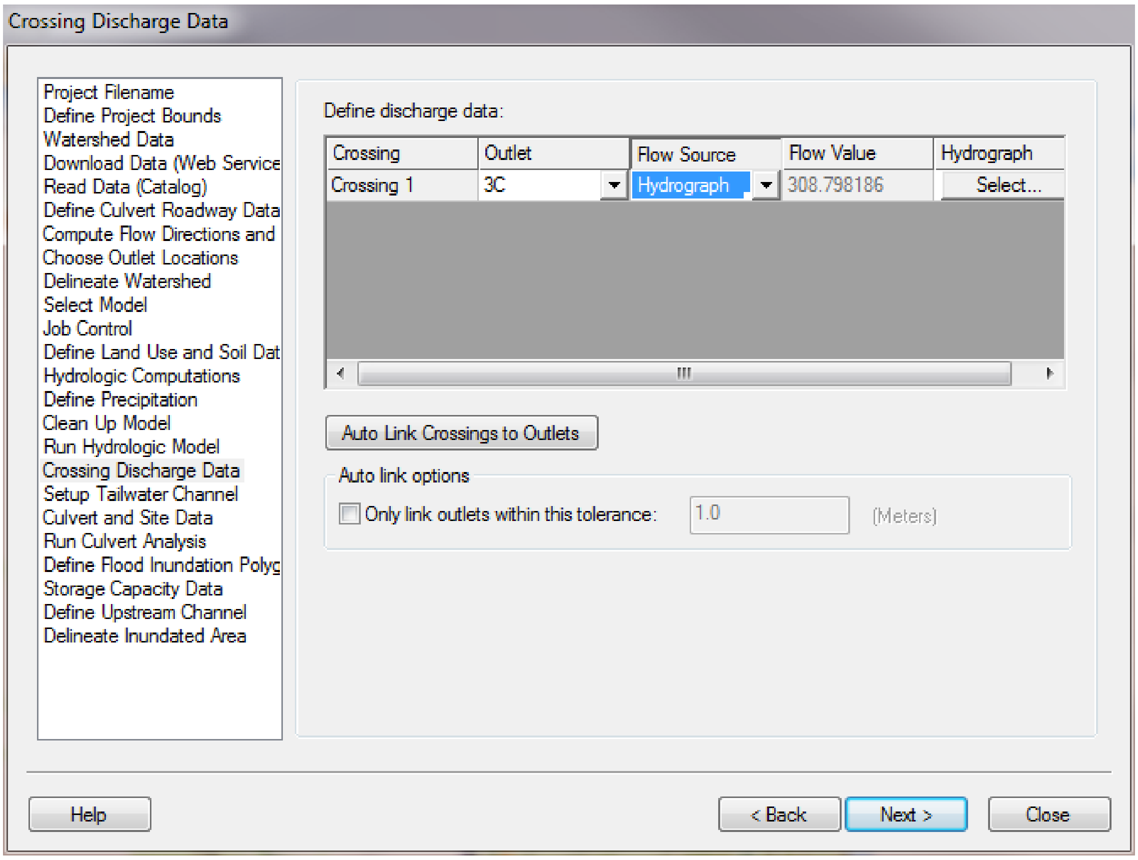
The Run Hydrologic Model step is used to run your hydrologic model and read the resulting hydrograph.

Help

If the current model is one of the models that is run from within WMS (TR-55, NSS, or Rational), this step only has a button to run the simulation. Since each model has different input and output files, WMS does not display all the input/output file edit fields for all of the models.

Crossing Discharge Data

Overview



The Crossing Discharge Data step allows users to pull a hydrograph from the hydrologic simulation, use the peak discharge from the hydrograph, or enter a hydrograph or peak discharge.

Help

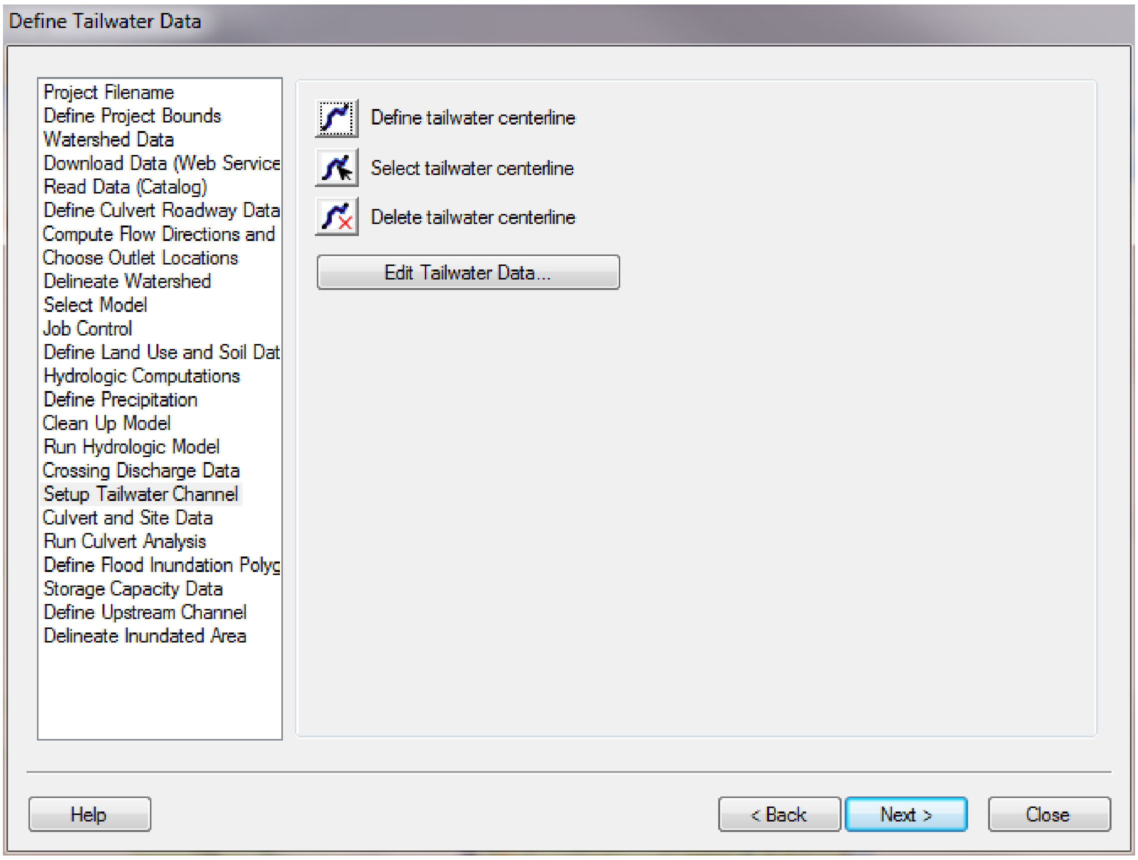
We list all the crossings in this dialog. If the “Auto link” option is selected, WMS links the outlets to the culvert crossings if they are within the specified tolerance (the same way outlets are connected to nodes in the storm drain interface).

For each crossing, the flow value and hydrograph are copied to the appropriate fields. Only the flow values on the rising limb of the hydrograph are copied to the XY series--we’re not concerned about the falling limb of the hydrograph since we will only be using the discharge-elevation curve computed by HY-8 for routing the original hydrograph. When WMS runs HY-8, it runs the analysis using the hydrograph peak flow value as the design and maximum flow.

For more information on how WMS uses this data to set the HY8 Roadway Data, please see: Path of Data from WMS to HY8

Setup Tailwater Channel

Overview



The Setup Tailwater Channel step is used to allow users to define a tailwater channel in much the same way as defining a roadway centerline or a culvert crossing centerline in a previous wizard step.

Help

When clicking on the define tailwater centerline button, WMS either gets the HY8 coverage and makes it the active coverage or creates a new HY8 coverage if one does not exist and make this new coverage the active coverage. You should create tailwater centerline arcs in the HY8 coverage. When creating the tailwater centerline arcs, start at the downstream end of the culvert crossing arc and create an arc downstream from that location.

If no tailwater arcs are selected, clicking on the **Edit Tailwater Data...** button selects all the tailwater arcs in the HY8 coverage and brings up the arc attributes for these arcs. If tailwater arcs are selected, only the selected tailwater arc attributes are shown. These attributes are shown in the arc attribute spreadsheet format.

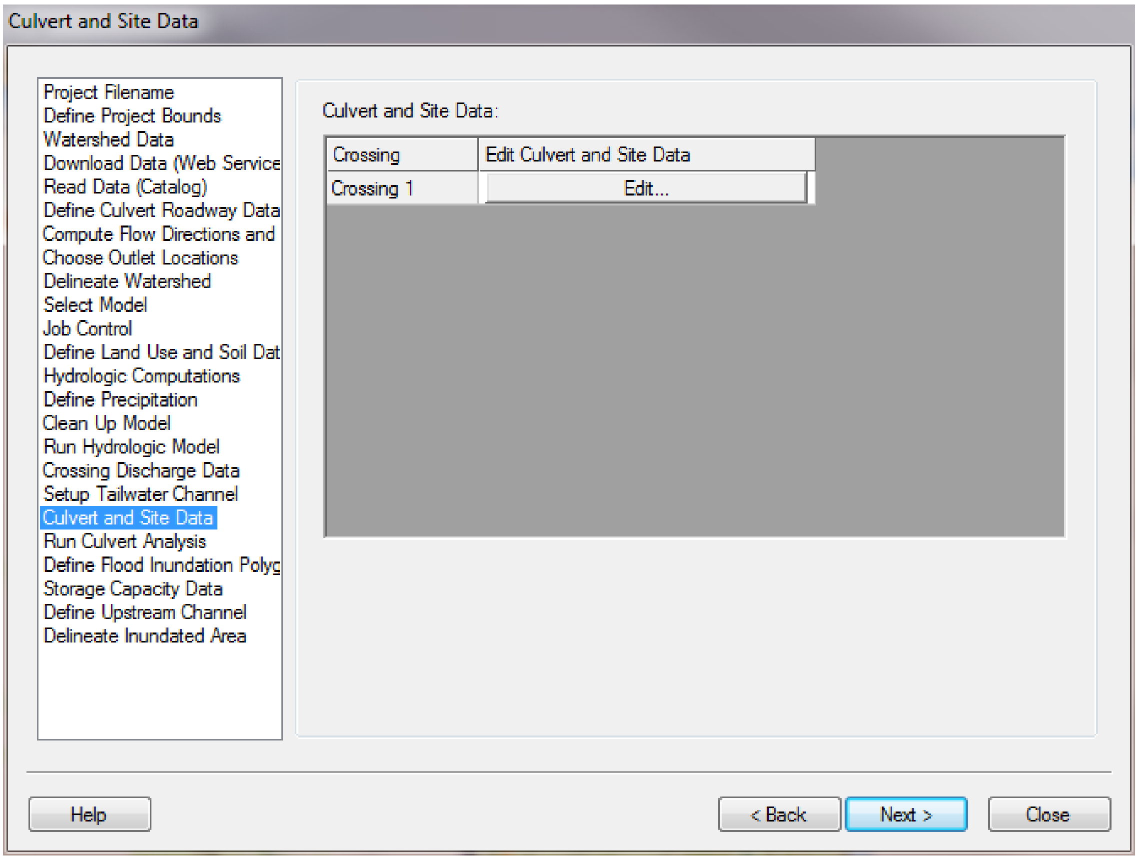
After creating tailwater arcs, values for channel slope and channel invert elevation are defaulted from the background DEM data, using the elevation at the most upstream end of the tailwater arc. Users need to enter values for other parameters such as bottom width, side slopes, and roughness.

If the irregular channel type is selected, a button that allows you to enter a cross section using the HY-8 irregular tailwater channel editor and then assigns this cross section to the tailwater channel.

For more information on how WMS uses this data to set the HY8 Roadway Data, please see: Path of Data from WMS to HY8

Culvert and Site Data

Overview



The Culvert and Site Data step is used to finish setting up your culvert data by defining the data in the HY-8 Crossing Data dialog. This step shows a spreadsheet with a listing of each culvert crossing. You can edit the culvert and site data associated with each crossing by clicking on the **Edit** button in the dialog.

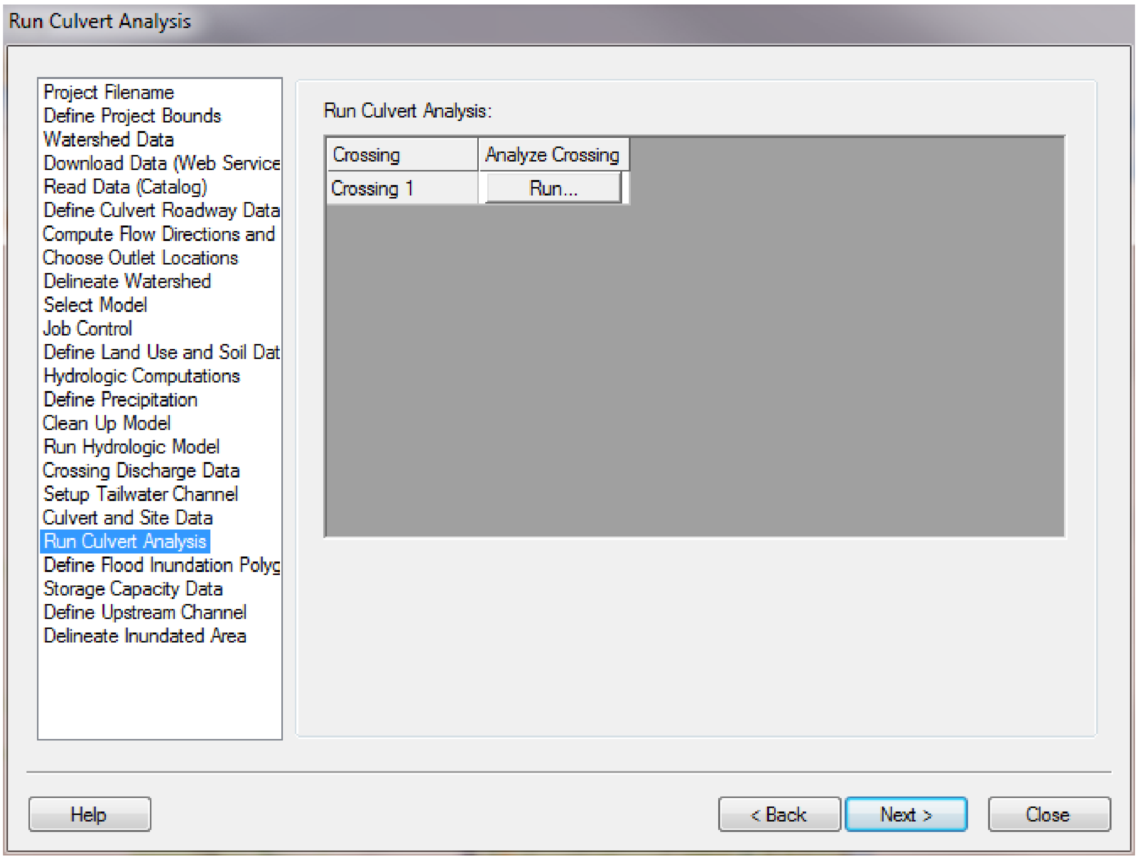
Help

Clicking the **edit** button saves an HY-8 input file to the *temp* directory, starts HY-8, and bring up the crossing data dialog for HY-8, where the site data parameters are defaulted based on the culvert arc locations entered earlier in the analysis. The discharge data, tailwater data, and roadway data are collected from previous steps in the wizard and from tailwater and roadway arc attributes. The culvert shape, size, and inlet condition still needs to be entered in the crossing data dialog.

Clicking on OK from the crossing data dialog saves an HY-8 file to the *temp* directory, closes HY-8, and reads the HY-8 file back into WMS so it is associated with the culvert crossing arc.

Run Culvert Analysis

Overview



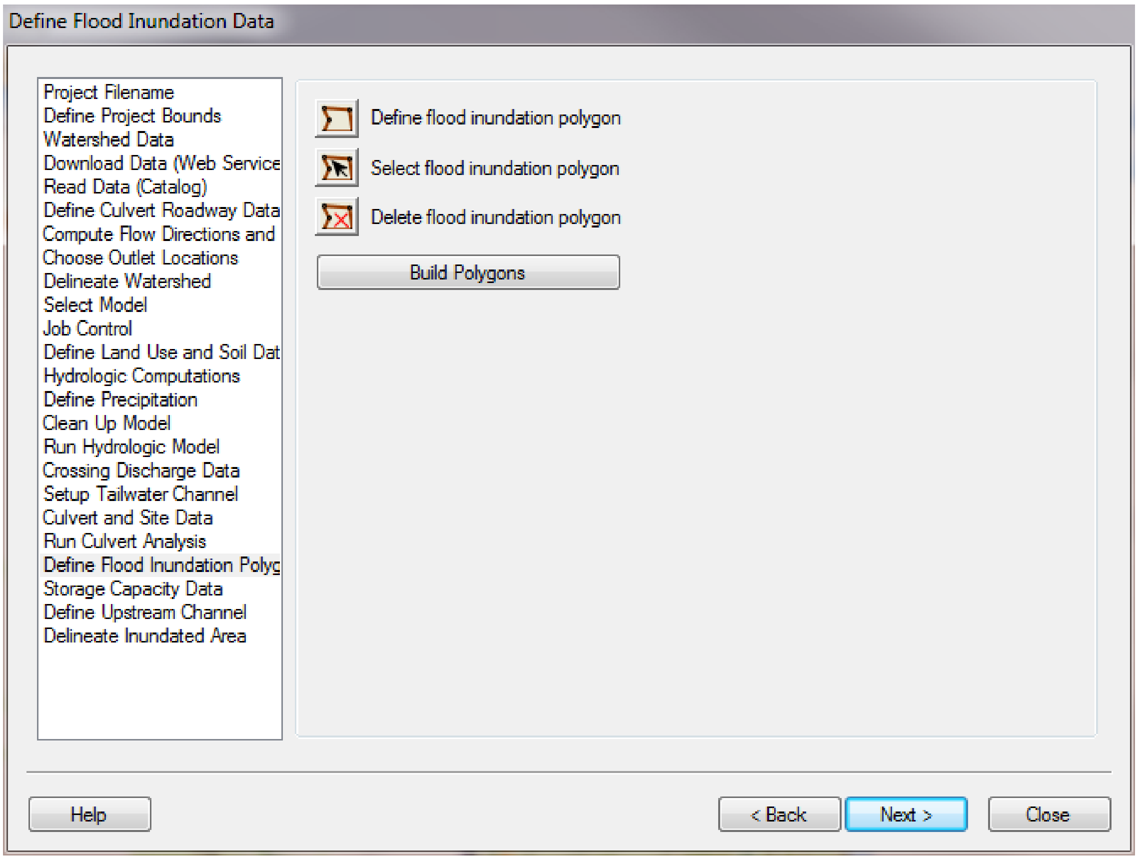
The Run Culvert Analysis step is used to run your culvert crossing and view the results of your culvert analysis.

Help

Clicking on the **run** button saves the HY-8 data to the *temp* directory and brings up the analysis in the HY-8 program itself. If all the data for a crossing is not setup, an error dialog appears with any errors that need to be fixed and a button is available in this dialog to go back to the crossing parameters dialog and fix any errors. Unless all errors are fixed, the analysis cannot be run.

Define Flood Inundation Polygon

Overview



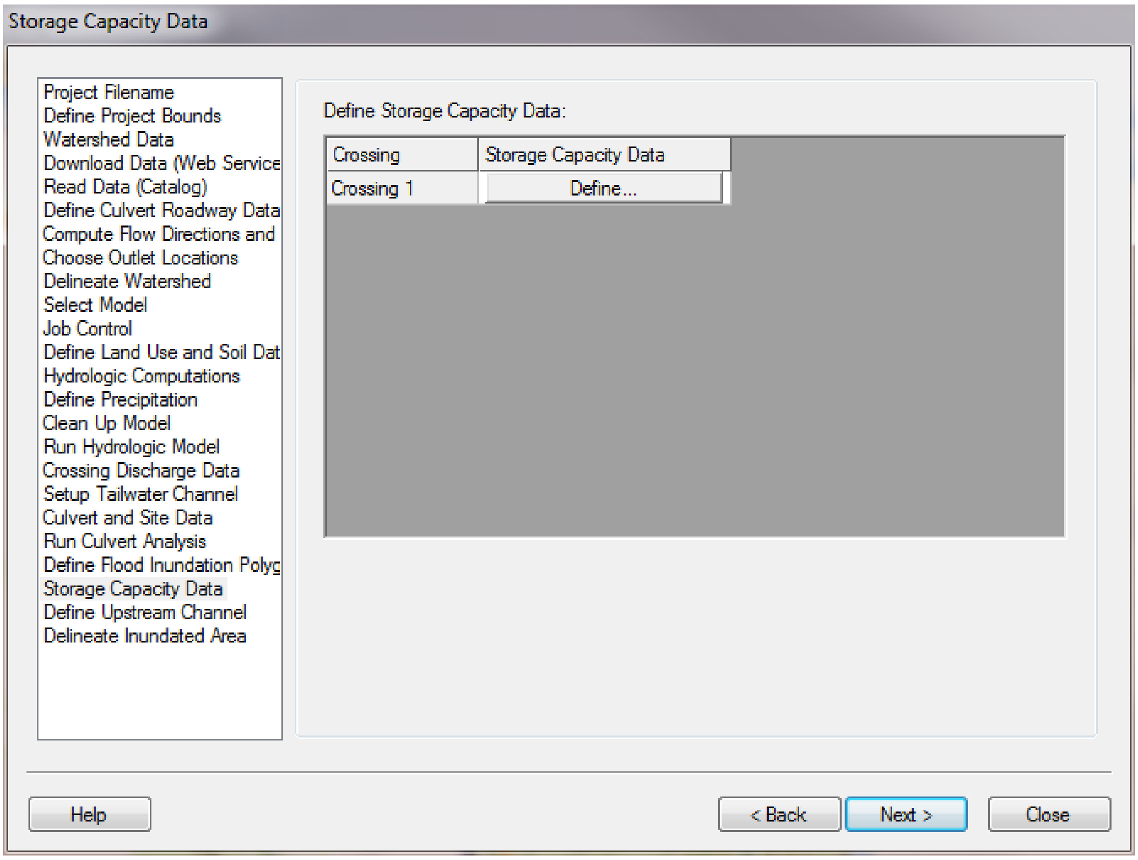
The Define Flood Inundation Polygon step is used to define a flood inundation polygon if a drainage coverage with a polygon defining the inundation area does not already exist. You need to define a flood inundation polygon for any culvert crossings with no outlets (and no basin polygons) assigned but with a hydrograph. The purpose of this inundation polygon is to compute a storage capacity curve and to define a flood barrier when running the floodplain delineation.

Help

Selecting the **define flood inundation polygon** button allows you to create arcs that represent your flood inundation boundary. Normally, you should use the roadway as one of the arcs and build arcs for the rest of the inundation boundary. If you select **delete flood inundation polygon** and then click on an arc or inside that polygon, the polygon is deleted and all the generic arcs (not the roadway centerline arcs) making up that polygon are deleted from memory. Selecting **build polygons** builds all the polygons in the HY-8 coverage and assigns them as flood inundation polygons without prompting whether you want to use all the arcs to build polygons.

Storage Capacity Data

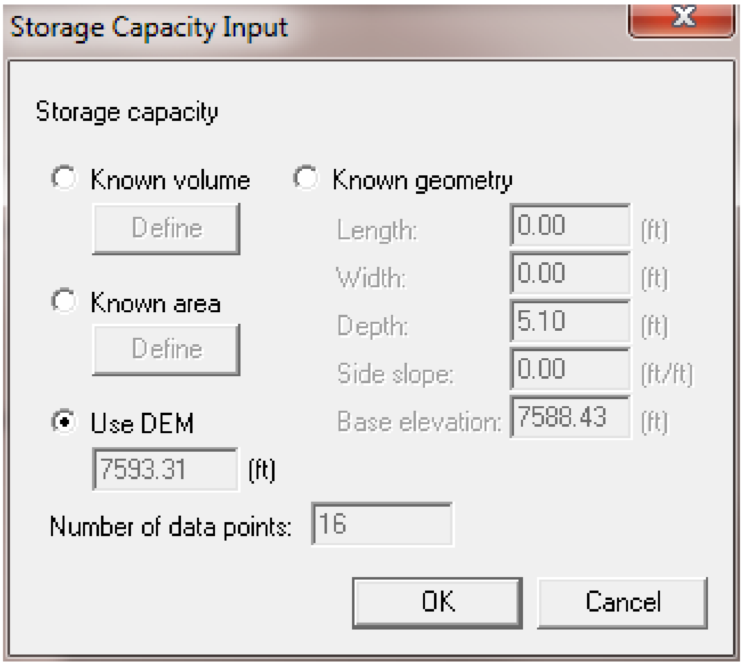
Overview



The Storage Capacity Data step is used to define storage capacity data and to route the hydrograph at the outlet through the culvert.

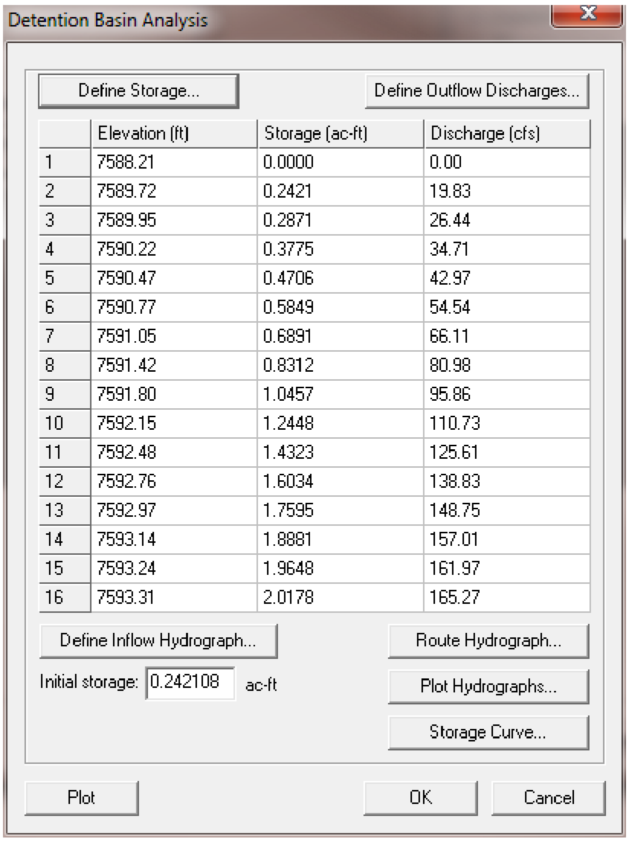
Help

This step in the wizard lists all the crossings in the HY-8 coverage and allows you to define storage capacity data for each of the crossings. The elevation-volume curve is defined from the DEM and the outlet/watershed boundary or the flood inundation polygon associated with the crossing. The elevations are defined from the headwater elevations in the HY-8 analysis. Alternatively, geometry of the area can be defined and the elevation-volume curve can be computed from this geometry or an elevation-area curve can be defined and the volume can be computed from this curve (a fourth option is that the storage can be entered directly into the storage capacity data dialog). When clicking on the Define button from the wizard, the *storage capacity input* dialog appears, shown below:



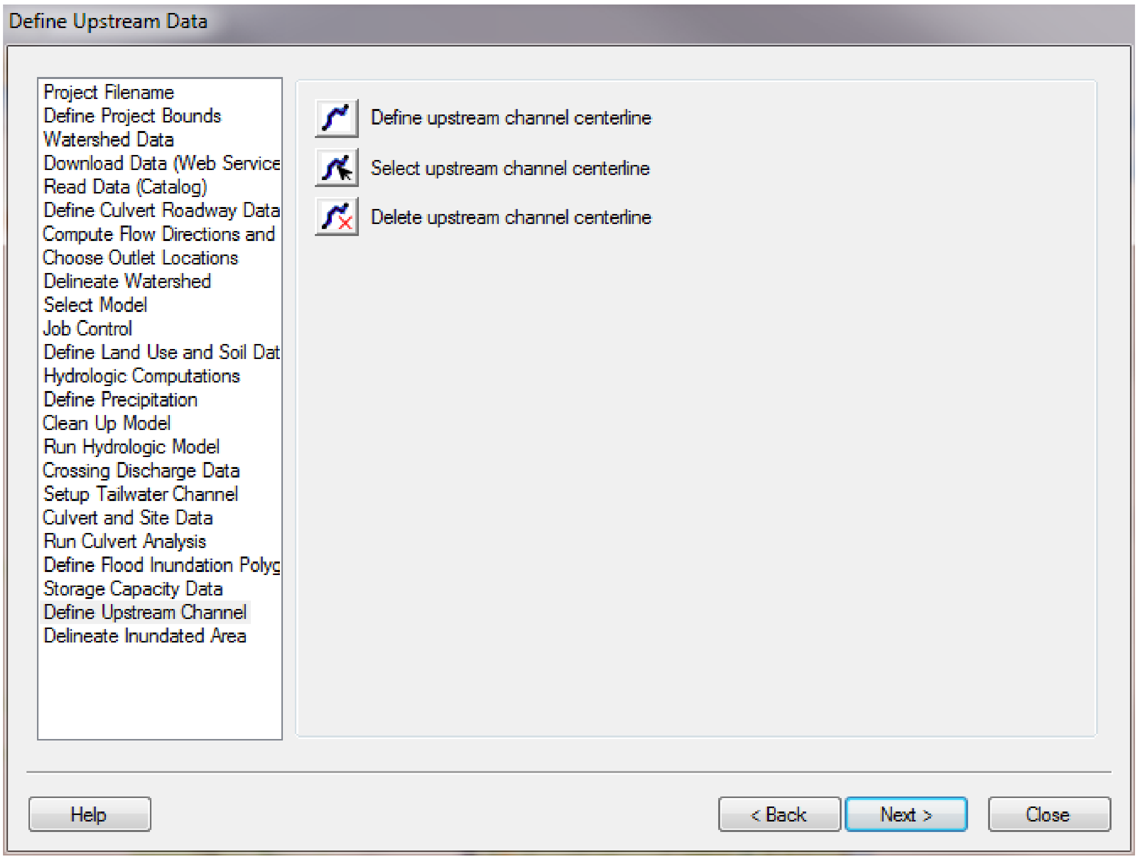
In this dialog, you will be able to select the option to use your DEM (this is the default if a DEM exists) to compute your volumes at each water surface elevation. Alternatively, you can enter the geometry of the basin upstream of the culvert or an elevation-area curve. If the *known area* option is selected, you will be allowed to manually enter the area at each of the headwater elevations computed by HY-8.

After exiting the define storage dialog, the *detention basin analysis* dialog shows the elevation-storage-discharge chart in an FHWA Hydraulic Toolbox dialog and allows you to view and edit the routed hydrograph and its associated data:



Define Upstream Channel

Overview



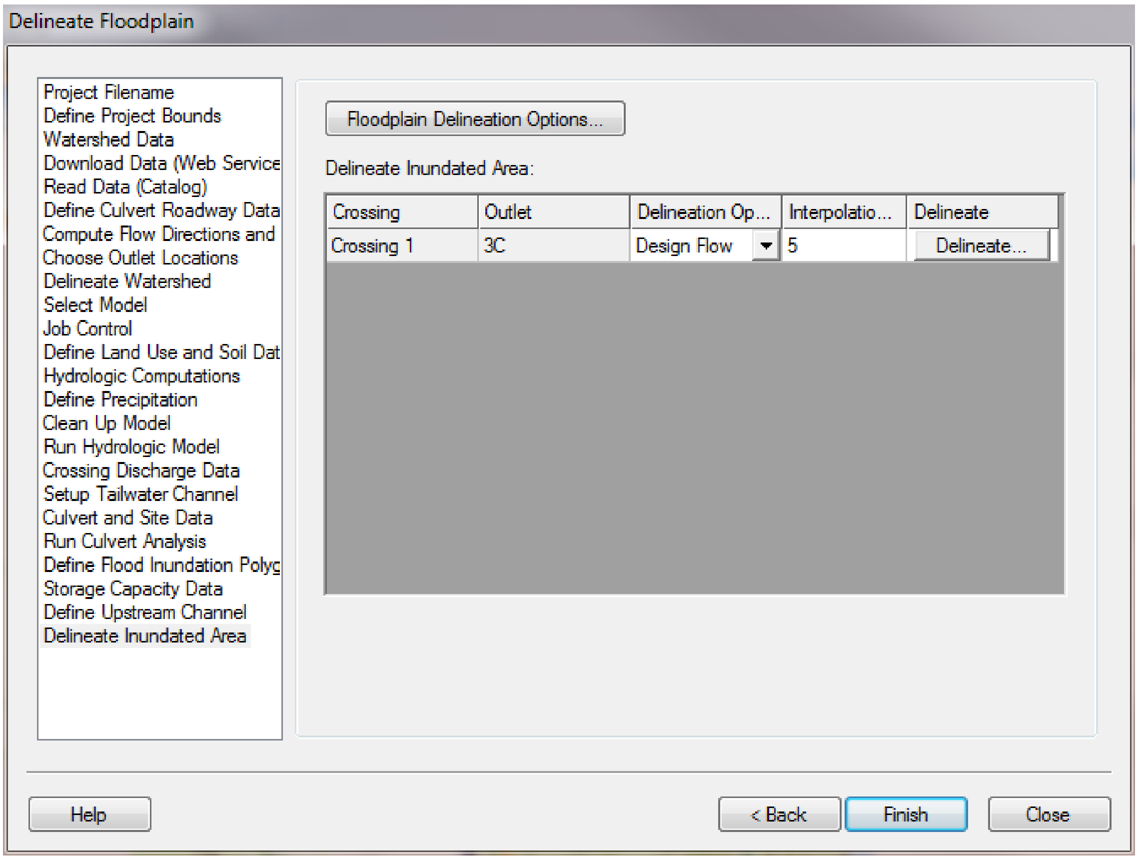
The Define Upstream Channel step is used to define an arc representing the upstream channel.

Help

This dialog functions similar to the step in which the tailwater channel was defined. When an arc is defined (arcs must be created from upstream to downstream) from this step in the wizard, their attribute is set to the HY-8 upstream arc attribute type. This arc is used when delineating the floodplain from the culvert headwater data in the next step of the wizard.

Delineate Inundated Area

Overview



The Delineate Inundated Area step is used to delineate the floodplain corresponding to the culvert's design flow or hydrograph at each culvert crossing.

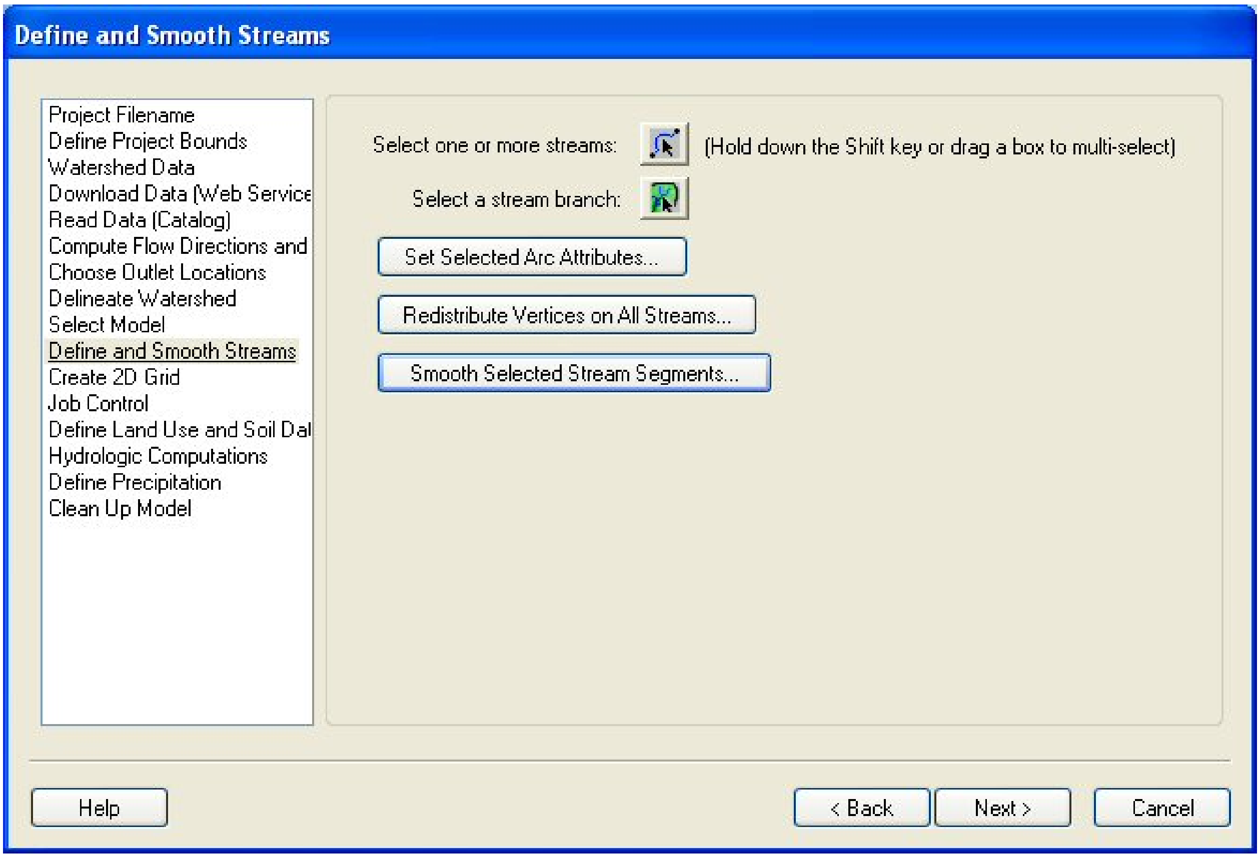
Help

The two delineation options are to delineate the entire hydrograph or to delineate only the design discharge. In either case, options such as the search radius set in the **floodplain delineation options** are used to delineate the floodplain.

When WMS delineates the floodplain for each culvert crossing, it creates a 2D scatter set with 1 water surface elevation point at the upstream end of the culvert and at the specified distance along the upstream arc. The water surface elevation point interpolation interval is defaulted such that 20 solution points show up along the length of the upstream arc, but you can edit this interpolation interval. When the **Delineate** button is pressed, WMS creates a new 2D scattered data set with elevation points at the upstream end of the culvert and at the specified interval along the upstream arc. WMS then populates a headwater elevation dataset in the 2D scattered dataset with the headwater values from the routed hydrograph or from the design discharge. Then WMS delineates the floodplain at each time step for a routed hydrograph or at a single time step for the design discharge.

Define and Smooth Streams

Overview



When you are building a GSSHA model, you need to define the parameters for and smooth all the streams in your model before building the 2D Grid. When the 2D grid is generated, the elevations on cells intersecting stream arcs will be made to match the stream arc channel depth elevations (the stream arc channel depth elevation along the stream = the stream elevation + the channel depth elevation entered in the GSSHA Feature Arc Type dialog).

Help

Select one or more streams – This button selects the select feature arc tool. After this tool is selected, you can select one or more stream arcs and edit the attributes or smooth the selected stream arcs.

Select a stream branch – This button selects the select stream branch tool. This is a specialized tool that selects a stream arc and any arcs upstream from the selected arc. This tool is useful for selecting an entire stream network or a branch of a stream network.

Set Selected Arc Attributes – This button brings up the GSSHA Feature Arc Type dialog. This dialog allows you to set the attributes for all the selected arcs.

Redistribute Vertices on All Streams – Selecting this button will first switch your active coverage to be the GSSHA coverage. If no arcs are selected, all the stream arcs will be selected in the GSSHA coverage. If arcs are selected, no additional arcs will be selected. The Redistribute Vertices dialog will appear, allowing you to set a new spacing for the vertices on the selected arcs.

Smooth Selected Stream Segments – This button will bring up the Smooth GSSHA Streams dialog for the selected arc(s). Before selecting this button, you should select a set of non-branched stream arcs. The smooth streams dialog has several options for modifying and smoothing the streams in the selected branch.

Article Sources and Contributors

Introduction to WMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=28299 *Contributors*: Eshaw

WMS Tutorials *Source*: http://www.xmswiki.com/xms/index.php?oldid=43240 *Contributors*: Cmsmemoe

What's new in WMS version 9.1 *Source*: http://www.xmswiki.com/xms/index.php?oldid=38248 *Contributors*: Cmsmemoe

FAQ *Source*: http://www.xmswiki.com/xms/index.php?oldid=43468 *Contributors*: Jcreer, Techbd

References *Source*: http://www.xmswiki.com/xms/index.php?oldid=33112 *Contributors*: Cmsmemoe

Readme *Source*: http://www.xmswiki.com/xms/index.php?oldid=30365 *Contributors*: Cbarlow

Setting Up WMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=28711 *Contributors*: Cmsmemoe

License Agreement *Source*: http://www.xmswiki.com/xms/index.php?oldid=39837 *Contributors*: Jcreer, Ksmith

Registering WMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=43469 *Contributors*: Cmsmemoe, Jcreer

Graphics Card Troubleshooting *Source*: http://www.xmswiki.com/xms/index.php?oldid=39838 *Contributors*: Jcreer, Ksmith

Running WMS for the first time *Source*: http://www.xmswiki.com/xms/index.php?oldid=25487 *Contributors*: Eshaw

Command Line *Source*: http://www.xmswiki.com/xms/index.php?oldid=27990 *Contributors*: Cmsmemoe

Bugfixes WMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=46026 *Contributors*: Cmsmemoe, Jcreer

WMS Basic Modeling Concepts *Source*: http://www.xmswiki.com/xms/index.php?oldid=28265 *Contributors*: Eshaw

WMS Downloads *Source*: http://www.xmswiki.com/xms/index.php?oldid=36600 *Contributors*: Cmsmemoe

Where Can I Get Data? *Source*: http://www.xmswiki.com/xms/index.php?oldid=25728 *Contributors*: Eshaw

Graphical User Interface (GUI) *Source*: http://www.xmswiki.com/xms/index.php?oldid=44903 *Contributors*: Eshaw, Jcreer

WMS Screen *Source*: http://www.xmswiki.com/xms/index.php?oldid=45007 *Contributors*: Eshaw, Jcreer

Overview *Source*: http://www.xmswiki.com/xms/index.php?oldid=45304 *Contributors*: Eshaw, Jcreer

WMS Dialog Help *Source*: http://www.xmswiki.com/xms/index.php?oldid=46537 *Contributors*: Cmsmemoe, Jcreer

Plot Windows *Source*: http://www.xmswiki.com/xms/index.php?oldid=46487 *Contributors*: Eshaw, Jcreer

Plot Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=25695 *Contributors*: Eshaw

Menu Bar *Source*: http://www.xmswiki.com/xms/index.php?oldid=25808 *Contributors*: Eshaw

Help Strip *Source*: http://www.xmswiki.com/xms/index.php?oldid=25869 *Contributors*: Eshaw

Toolbars *Source*: http://www.xmswiki.com/xms/index.php?oldid=26741 *Contributors*: Eshaw

Tool Palettes *Source*: http://www.xmswiki.com/xms/index.php?oldid=46500 *Contributors*: Eshaw, Jcreer

Module Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=44113 *Contributors*: Eshaw, Jcreer

Macros *Source*: http://www.xmswiki.com/xms/index.php?oldid=44881 *Contributors*: Eshaw, Jcreer

Units Toolbar *Source*: http://www.xmswiki.com/xms/index.php?oldid=25880 *Contributors*: Eshaw

Digitize Toolbar *Source*: http://www.xmswiki.com/xms/index.php?oldid=45768 *Contributors*: Cmsmemoe, Jcreer

Static Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=44826 *Contributors*: Eshaw, Jcreer

Dynamic Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=44925 *Contributors*: Eshaw, Jcreer

Drawing Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=26715 *Contributors*: Eshaw

Get Data Toolbar *Source*: http://www.xmswiki.com/xms/index.php?oldid=44956 *Contributors*: Cmsmemoe, Jcreer

Get Data Tool *Source*: http://www.xmswiki.com/xms/index.php?oldid=44872 *Contributors*: Jcreer, Rkent

Setting up Film Loops *Source*: http://www.xmswiki.com/xms/index.php?oldid=42719 *Contributors*: Eshaw, Jcreer

File Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=45018 *Contributors*: Eshaw, Jcreer

Edit Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=44947 *Contributors*: Eshaw, Jcreer

Display Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=45173 *Contributors*: Eshaw, Jcreer

Properties Window *Source*: http://www.xmswiki.com/xms/index.php?oldid=46181 *Contributors*: Cmsmemoe, Eshaw, Jcreer

Coordinate Systems *Source*: http://www.xmswiki.com/xms/index.php?oldid=41675 *Contributors*: Jcreer, Ksmith

Coordinate Conversions *Source*: http://www.xmswiki.com/xms/index.php?oldid=37513 *Contributors*: Ksmith

Projections *Source*: http://www.xmswiki.com/xms/index.php?oldid=37514 *Contributors*: Ksmith, Unjedai

CPP Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=37717 *Contributors*: Ksmith

Geographic Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=36074 *Contributors*: Ksmith, Unjedai

Local Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=38290 *Contributors*: Jcreer, Ksmith

Converting Coordinate Systems *Source*: http://www.xmswiki.com/xms/index.php?oldid=25896 *Contributors*: Eshaw

Coordinate Tracking *Source*: http://www.xmswiki.com/xms/index.php?oldid=25874 *Contributors*: -

Coordinate Systems of GIS Layers *Source*: http://www.xmswiki.com/xms/index.php?oldid=25899 *Contributors*: Eshaw

Editing XYZ Coordinates *Source*: http://www.xmswiki.com/xms/index.php?oldid=28286 *Contributors*: Eshaw

Local Origin Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=9273 *Contributors*: Eshaw

Single Point Conversion *Source*: http://www.xmswiki.com/xms/index.php?oldid=21007 *Contributors*: Eshaw

Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=46371 *Contributors*: Cmsmemoe, Eshaw, Jcreer

Color Ramp *Source*: http://www.xmswiki.com/xms/index.php?oldid=44987 *Contributors*: Eshaw, Jcreer

Contour Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=44974 *Contributors*: Eshaw, Jcreer

Drawing Objects *Source*: http://www.xmswiki.com/xms/index.php?oldid=43115 *Contributors*: Eshaw, Jcreer

Display Order *Source*: http://www.xmswiki.com/xms/index.php?oldid=46416 *Contributors*: Cmsmemoe, Jcreer

CAD Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=45356 *Contributors*: Jcreer

Hydrologic/Hydraulic Calculators *Source*: http://www.xmswiki.com/xms/index.php?oldid=45097 *Contributors*: Jcreer, Mshapiro

Lag Time and Time of Concentration *Source*: http://www.xmswiki.com/xms/index.php?oldid=25993 *Contributors*: Eshaw

Compute GIS Attributes *Source*: http://www.xmswiki.com/xms/index.php?oldid=45093 *Contributors*: Jcreer, Mshapiro

Detention Basin Calculator *Source*: http://www.xmswiki.com/xms/index.php?oldid=45034 *Contributors*: EJones, Eshaw, Jcreer

Channel Calculator *Source*: http://www.xmswiki.com/xms/index.php?oldid=45053 *Contributors*: EJones, Eshaw, Jcreer

Weir Calculator *Source*: http://www.xmswiki.com/xms/index.php?oldid=45038 *Contributors*: EJones, Eshaw, Jcreer

Datasets *Source*: http://www.xmswiki.com/xms/index.php?oldid=46486 *Contributors*: Cmsmemoe, Jcreer

Deleting Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44943 *Contributors*: Eshaw, Jcreer

Gages *Source*: http://www.xmswiki.com/xms/index.php?oldid=45774 *Contributors*: Cmsmemoe, Jcreer, Jschultze

Hydrologic Modeling Wizard Overview *Source*: http://www.xmswiki.com/xms/index.php?oldid=44960 *Contributors*: Jcreer

XY Series Editor *Source*: http://www.xmswiki.com/xms/index.php?oldid=45107 *Contributors*: Eshaw, Jcreer

Land Use *Source*: http://www.xmswiki.com/xms/index.php?oldid=46413 *Contributors*: Cmsmemoe, Jcreer, Mshapiro

Edit File *Source*: http://www.xmswiki.com/xms/index.php?oldid=45100 *Contributors*: Eshaw, Jcreer

Annotations *Source*: http://www.xmswiki.com/xms/index.php?oldid=41245 *Contributors*: Jcreer, Ksmith

Topologic Trees *Source*: http://www.xmswiki.com/xms/index.php?oldid=45182 *Contributors*: Eshaw, Jcreer

Hydrographs *Source*: http://www.xmswiki.com/xms/index.php?oldid=45167 *Contributors*: Eshaw, Jcreer

Hydrograph Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=23676 *Contributors*: Eshaw

Images *Source*: http://www.xmswiki.com/xms/index.php?oldid=45307 *Contributors*: Jcreer, Wood

Image Preferences *Source*: http://www.xmswiki.com/xms/index.php?oldid=45279 *Contributors*: Eshaw, Jcreer

Image Gallery *Source*: http://www.xmswiki.com/xms/index.php?oldid=36943 *Contributors*: Mmedina, Unjedai

Registering an Image *Source*: http://www.xmswiki.com/xms/index.php?oldid=39779 *Contributors*: Jcreer, Ksmith

Text Import Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=45024 *Contributors*: Eshaw, Jcreer

ARC/INFO ASCII Grid Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=45370 *Contributors*: Eshaw, Jcreer

ASCII Dataset Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=43057 *Contributors*: Eshaw, Jcreer

Binary Dataset Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=43060 *Contributors*: Eshaw, Jcreer

DSS Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=23673 *Contributors*: Eshaw

DSS File I/O *Source*: http://www.xmswiki.com/xms/index.php?oldid=23672 *Contributors*: Eshaw

DSS Files-WMS Interface *Source*: http://www.xmswiki.com/xms/index.php?oldid=23674 *Contributors*: Eshaw

File Extension *Source*: http://www.xmswiki.com/xms/index.php?oldid=32126 *Contributors*: Jschultze

File I/O *Source*: http://www.xmswiki.com/xms/index.php?oldid=45079 *Contributors*: Eshaw, Jcreer

Image Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=23677 *Contributors*: Eshaw

Import GIS File *Source*: http://www.xmswiki.com/xms/index.php?oldid=25912 *Contributors*: Eshaw

Export GIS File *Source*: http://www.xmswiki.com/xms/index.php?oldid=32558 *Contributors*: Wood

GRASS Grid (GSSHA Maps) *Source*: http://www.xmswiki.com/xms/index.php?oldid=43296 *Contributors*: Jcreer, Mshapiro

Land Use Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=23678 *Contributors*: Eshaw

Read Stage File *Source*: http://www.xmswiki.com/xms/index.php?oldid=28520 *Contributors*: Eshaw

Read a Stage File *Source*: http://www.xmswiki.com/xms/index.php?oldid=25840 *Contributors*: Eshaw

Shapefiles *Source*: http://www.xmswiki.com/xms/index.php?oldid=46179 *Contributors*: Eshaw, Jcreer

WMS Native Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=44904 *Contributors*: Eshaw, Jcreer

WMS Non-native Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=45336 *Contributors*: Cmsmemoe, Jcreer

Soil Type Runoff Coefficient Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=23679 *Contributors*: Eshaw

WMS Supported File Formats *Source*: http://www.xmswiki.com/xms/index.php?oldid=44914 *Contributors*: Eshaw, Jcreer

WMS Project Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=44895 *Contributors*: Eshaw, Jcreer

XY Series Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=23682 *Contributors*: Eshaw

Interpolation Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=44734 *Contributors*: Eshaw, Jcreer

Linear Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44634 *Contributors*: Eshaw, Jcreer

Inverse Distance Weighted Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44655 *Contributors*: Eshaw, Jcreer

Clogh-Tocher Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44657 *Contributors*: Eshaw, Jcreer

Natural Neighbor Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44668 *Contributors*: Eshaw, Jcreer

Interpolating Hydraulic Model Results *Source*: http://www.xmswiki.com/xms/index.php?oldid=44670 *Contributors*: Eshaw, Jcreer

Interpolate Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=44671 *Contributors*: Eshaw, Jcreer

Computation of Interpolation Weights *Source*: http://www.xmswiki.com/xms/index.php?oldid=44695 *Contributors*: Eshaw, Jcreer

Subset Definition *Source*: http://www.xmswiki.com/xms/index.php?oldid=44673 *Contributors*: Eshaw, Jcreer

Assigning an Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44627 *Contributors*: Eshaw, Jcreer

Customizing Equations *Source*: http://www.xmswiki.com/xms/index.php?oldid=44626 *Contributors*: Eshaw, Jcreer

Repeated Use of an Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44625 *Contributors*: Eshaw, Jcreer

Variables Computed by WMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=44621 *Contributors*: Eshaw, Jcreer

Overview of Basin Data Equations *Source*: http://www.xmswiki.com/xms/index.php?oldid=44570 *Contributors*: Eshaw, Jcreer

Travel Times from Basin Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=43213 *Contributors*: Eshaw, Jcreer

Colorado State Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44580 *Contributors*: Eshaw, Jcreer

Customized Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44579 *Contributors*: Eshaw, Jcreer

Denver Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44584 *Contributors*: Eshaw, Jcreer

Eagleson Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44586 *Contributors*: Eshaw, Jcreer

Espey Lag Time Equations *Source*: http://www.xmswiki.com/xms/index.php?oldid=44589 *Contributors*: Eshaw, Jcreer

Putnam Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44592 *Contributors*: Eshaw, Jcreer

Riverside County Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44596 *Contributors*: Eshaw, Jcreer

SCS Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44600 *Contributors*: Eshaw, Jcreer

Taylor Schwartz Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44603 *Contributors*: Eshaw, Jcreer

Tulsa District Lag Time Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44607 *Contributors*: Eshaw, Jcreer

Fort Bend Tc Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44624 *Contributors*: Eshaw, Jcreer

Kerby Tc Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44623 *Contributors*: Eshaw, Jcreer

Kirpich Tc Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44372 *Contributors*: Eshaw, Jcreer

Ramser Tc Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44367 *Contributors*: Eshaw, Jcreer

Modules *Source*: http://www.xmswiki.com/xms/index.php?oldid=26738 *Contributors*: Eshaw

DEM Guidelines *Source*: http://www.xmswiki.com/xms/index.php?oldid=42812 *Contributors*: Eshaw, Jcreer

Importing USGS DEMs *Source*: http://www.xmswiki.com/xms/index.php?oldid=42790 *Contributors*: Eshaw, Jcreer

Importing Flow Directions and Accumulations *Source*: http://www.xmswiki.com/xms/index.php?oldid=21904 *Contributors*: Eshaw

DEM to Stream Arcs *Source*: http://www.xmswiki.com/xms/index.php?oldid=43103 *Contributors*: Eshaw, Jcreer

Interpolation of DEMs elevations to TINs and 2D Grids *Source*: http://www.xmswiki.com/xms/index.php?oldid=42792 *Contributors*: Eshaw, Jcreer

Editing DEMs *Source*: http://www.xmswiki.com/xms/index.php?oldid=42988 *Contributors*: Eshaw, Jcreer

Editing DEM Elevations *Source*: http://www.xmswiki.com/xms/index.php?oldid=32249 *Contributors*: Jschultze

Delineation with DEMs *Source*: http://www.xmswiki.com/xms/index.php?oldid=42814 *Contributors*: Jcreer, Jdye

Smoothing DEMs *Source*: http://www.xmswiki.com/xms/index.php?oldid=28267 *Contributors*: Eshaw

DEM Point Attributes *Source*: http://www.xmswiki.com/xms/index.php?oldid=42819 *Contributors*: Eshaw, Jcreer

DEM Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=42797 *Contributors*: Eshaw, Jcreer

Converting DEMs *Source*: http://www.xmswiki.com/xms/index.php?oldid=46493 *Contributors*: Eshaw, Jcreer

Create Reservoir in DEMs *Source*: http://www.xmswiki.com/xms/index.php?oldid=43849 *Contributors*: Jcreer, Mshapiro

Draw Flow Patterns in DEMs *Source*: http://www.xmswiki.com/xms/index.php?oldid=43031 *Contributors*: Eshaw, Jcreer

Nodes and Vertices to Outlets *Source*: http://www.xmswiki.com/xms/index.php?oldid=44225 *Contributors*: Eshaw, Jcreer

DEM Basins *Source*: http://www.xmswiki.com/xms/index.php?oldid=43113 *Contributors*: Jcreer

Compute Basin Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44210 *Contributors*: Eshaw, Jcreer

TOPAZ *Source*: http://www.xmswiki.com/xms/index.php?oldid=42813 *Contributors*: Jcreer, Twood

TIN Guidelines *Source*: http://www.xmswiki.com/xms/index.php?oldid=45833 *Contributors*: Cbarlow, Cmsmemoe, Jcreer

Creating TINs *Source*: http://www.xmswiki.com/xms/index.php?oldid=45769 *Contributors*: Cmsmemoe, Eshaw, Jcreer

TIN Breaklines *Source*: http://www.xmswiki.com/xms/index.php?oldid=42927 *Contributors*: Eshaw, Jcreer

Streams *Source*: http://www.xmswiki.com/xms/index.php?oldid=42983 *Contributors*: Jcreer, Mshapiro

TIN Basins *Source*: http://www.xmswiki.com/xms/index.php?oldid=45832 *Contributors*: Cmsmemoe, Jcreer

TIN Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=42994 *Contributors*: Jcreer

Convert TINs *Source*: http://www.xmswiki.com/xms/index.php?oldid=44900 *Contributors*: Jcreer

Triangulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44924 *Contributors*: Eshaw, Jcreer

Reservoirs *Source*: http://www.xmswiki.com/xms/index.php?oldid=42972 *Contributors*: Jcreer

Watershed Delineation with TINs *Source*: http://www.xmswiki.com/xms/index.php?oldid=42966 *Contributors*: Eshaw, Jcreer

TIN Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=42992 *Contributors*: Eshaw, Jcreer

Outlets *Source*: http://www.xmswiki.com/xms/index.php?oldid=42982 *Contributors*: Eshaw, Jcreer

Drawing Flow Patterns *Source*: http://www.xmswiki.com/xms/index.php?oldid=42973 *Contributors*: Eshaw, Jcreer

Vertices *Source*: http://www.xmswiki.com/xms/index.php?oldid=42989 *Contributors*: Jcreer

Smooth Dataset *Source*: http://www.xmswiki.com/xms/index.php?oldid=45829 *Contributors*: Cmsmemoe

Terrain Data Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=44219 *Contributors*: Eshaw, Jcreer

Terrain Data Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=44214 *Contributors*: Eshaw, Jcreer

Project Explorer Contents for Terrain Data Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=46426 *Contributors*: Cmsmemoe, Eshaw, Jcreer

Drainage Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=42890 *Contributors*: Jcreer, Ndye

Drainage *Source*: http://www.xmswiki.com/xms/index.php?oldid=29142 *Contributors*: Rkent

Drainage Data to Feature Objects *Source*: http://www.xmswiki.com/xms/index.php?oldid=44221 *Contributors*: Eshaw, Jcreer

Drainage Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=44222 *Contributors*: Eshaw, Jcreer

Project Explorer Contents for Drainage Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=46427 *Contributors*: Cmsmemoe, Eshaw, Jcreer

Arcs to Outlets *Source*: http://www.xmswiki.com/xms/index.php?oldid=44224 *Contributors*: Jcreer, Mshapiro

Map Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=45330 *Contributors*: Eshaw, Jcreer

Map Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=10963 *Contributors*: Eshaw

Map Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=42209 *Contributors*: Eshaw, Jcreer

Mapping Tables *Source*: http://www.xmswiki.com/xms/index.php?oldid=43010 *Contributors*: Jcreer, Mshapiro

Conceptual Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=46120 *Contributors*: Cmsmemoe, Jcreer

Mapping to Feature Objects *Source*: http://www.xmswiki.com/xms/index.php?oldid=25737 *Contributors*: Eshaw

Project Explorer Contents for Map Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=25699 *Contributors*: Eshaw

Coverages *Source*: http://www.xmswiki.com/xms/index.php?oldid=46522 *Contributors*: Cmsmemoe, Eshaw, Jcreer

Coverage Overlays *Source*: http://www.xmswiki.com/xms/index.php?oldid=28723 *Contributors*: Mshapiro

1D-HYD Centerline *Source*: http://www.xmswiki.com/xms/index.php?oldid=44762 *Contributors*: Eshaw, Jcreer

1D-HYD Cross-section *Source*: http://www.xmswiki.com/xms/index.php?oldid=44763 *Contributors*: Eshaw, Jcreer

Area Property Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=44996 *Contributors*: Eshaw, Jcreer

Flood Barrier *Source*: http://www.xmswiki.com/xms/index.php?oldid=43160 *Contributors*: Eshaw, Jcreer

Flood Extent *Source*: http://www.xmswiki.com/xms/index.php?oldid=44760 *Contributors*: Eshaw, Jcreer

Flow Paths and Barrier Coverages *Source*: http://www.xmswiki.com/xms/index.php?oldid=44742 *Contributors*: Eshaw, Jcreer

General *Source*: http://www.xmswiki.com/xms/index.php?oldid=43155 *Contributors*: Eshaw, Jcreer

Land Use Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=44765 *Contributors*: Jcreer, Mshapiro

MODRAT Tc Coverage Type *Source*: http://www.xmswiki.com/xms/index.php?oldid=44766 *Contributors*: Eshaw, Jcreer

NSS Region Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=44767 *Contributors*: Cmsmemoe, Jcreer

Rain Gage *Source*: http://www.xmswiki.com/xms/index.php?oldid=44770 *Contributors*: Jcreer, Wood

Soil *Source*: http://www.xmswiki.com/xms/index.php?oldid=44771 *Contributors*: Jcreer, Mshapiro

Time Computation *Source*: http://www.xmswiki.com/xms/index.php?oldid=45099 *Contributors*: Eshaw, Jcreer

Feature Objects *Source*: http://www.xmswiki.com/xms/index.php?oldid=25801 *Contributors*: Eshaw

Feature Object Guidelines *Source*: http://www.xmswiki.com/xms/index.php?oldid=43099 *Contributors*: Eshaw, Jcreer

Feature Objects Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=43098 *Contributors*: Jcreer

Creating Grids *Source*: http://www.xmswiki.com/xms/index.php?oldid=25743 *Contributors*: Eshaw

Hydrologic Modeling Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=28172 *Contributors*: Eshaw

Hydrologic Modeling Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=25654 *Contributors*: Eshaw

Project Explorer Contents for Hydrologic Modeling Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=25802 *Contributors*: Eshaw

Combining Arc Travel Times *Source*: http://www.xmswiki.com/xms/index.php?oldid=43131 *Contributors*: Eshaw, Jcreer

Elevation Discharge Relationship *Source*: http://www.xmswiki.com/xms/index.php?oldid=42627 *Contributors*: Eshaw, Jcreer

Storage Capacity Curves *Source*: http://www.xmswiki.com/xms/index.php?oldid=46201 *Contributors*: Eshaw, Jcreer

Computing Area Between Elevations *Source*: http://www.xmswiki.com/xms/index.php?oldid=43901 *Contributors*: Jcreer, Mshapiro

Hydraulic Modeling Introduction *Source*: http://www.xmswiki.com/xms/index.php?oldid=44248 *Contributors*: Eshaw, Jcreer

Hydraulic Modeling Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=44229 *Contributors*: Eshaw, Jcreer

Hydraulic Schematic *Source*: http://www.xmswiki.com/xms/index.php?oldid=44231 *Contributors*: Cmsmemoe, Jcreer

Hydraulic Toolbox 1.0 *Source*: http://www.xmswiki.com/xms/index.php?oldid=44232 *Contributors*: Cmsmemoe, Jcreer

Mapping the Conceptual Model to a River Schematic *Source*: http://www.xmswiki.com/xms/index.php?oldid=44230 *Contributors*: Eshaw, Jcreer

Project Explorer Contents for River Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=46121 *Contributors*: Eshaw, Jcreer

Editing Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=44913 *Contributors*: Eshaw, Jcreer

Extract Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=44239 *Contributors*: Eshaw, Jcreer

Managing Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=44243 *Contributors*: Eshaw, Jcreer

River Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=44257 *Contributors*: Eshaw, Jcreer

Overview of Flood Plain Delineation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44754 *Contributors*: Eshaw, Jcreer

Differences From Earlier Versions (Version 6.0 and earlier) *Source*: http://www.xmswiki.com/xms/index.php?oldid=44755 *Contributors*: Eshaw, Jcreer

Delineate Flood Plain *Source*: http://www.xmswiki.com/xms/index.php?oldid=44748 *Contributors*: Eshaw, Jcreer

Preparing Stage Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44116 *Contributors*: Eshaw, Jcreer

GIS Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=45937 *Contributors*: Cmsmemoe, Eshaw, Jcreer

GIS Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=43003 *Contributors*: Eshaw, Jcreer

Project Explorer Contents for GIS Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=45938 *Contributors*: Cmsmemoe, Eshaw, Jcreer

Deleting GIS Data Layers *Source*: http://www.xmswiki.com/xms/index.php?oldid=25739 *Contributors*: Eshaw

Computation Step *Source*: http://www.xmswiki.com/xms/index.php?oldid=28728 *Contributors*: Mshapiro

GIS Tables *Source*: http://www.xmswiki.com/xms/index.php?oldid=43005 *Contributors*: Jcreer

Soil Type *Source*: http://www.xmswiki.com/xms/index.php?oldid=46423 *Contributors*: Cmsmemoe, Jcreer, Mshapiro

Creating Watershed Models *Source*: http://www.xmswiki.com/xms/index.php?oldid=25744 *Contributors*: Eshaw

Introduction to ArcObjects *Source*: http://www.xmswiki.com/xms/index.php?oldid=43014 *Contributors*: Eshaw, Jcreer

ArcObjects Add Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=43016 *Contributors*: Eshaw, Jcreer

ArcView Data Guidelines *Source*: http://www.xmswiki.com/xms/index.php?oldid=43100 *Contributors*: Eshaw, Jcreer

Feature Selection *Source*: http://www.xmswiki.com/xms/index.php?oldid=46519 *Contributors*: Eshaw, Jcreer

Importing Shapefiles *Source*: http://www.xmswiki.com/xms/index.php?oldid=43101 *Contributors*: Eshaw, Jcreer

Add Shapefile Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=43015 *Contributors*: Eshaw, Jcreer

Feature Objects to ArcHydro Geodatabase *Source*: http://www.xmswiki.com/xms/index.php?oldid=46520 *Contributors*: Eshaw, Jcreer

Feature Objects to Geodatabase *Source*: http://www.xmswiki.com/xms/index.php?oldid=43040 *Contributors*: Eshaw, Jcreer

2D Grid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=25443 *Contributors*: Eshaw

Grid Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=44865 *Contributors*: Eshaw, Jcreer

2D Grid Module Project Explorer *Source*: http://www.xmswiki.com/xms/index.php?oldid=43563 *Contributors*: Eshaw, Jcreer

2D Grid Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=46508 *Contributors*: Eshaw, Jcreer

2D Grid Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=46499 *Contributors*: Eshaw, Jcreer

Tree Contents for 2D Grid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=46510 *Contributors*: Eshaw, Jcreer

Active/Inactive Cells *Source*: http://www.xmswiki.com/xms/index.php?oldid=46512 *Contributors*: Eshaw, Jcreer

Data Type Conversion *Source*: http://www.xmswiki.com/xms/index.php?oldid=44131 *Contributors*: Eshaw, Jcreer

2D Scatter Point Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=46498 *Contributors*: Eshaw, Jcreer

Project Explorer Contents for 2D Scatter Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=44133 *Contributors*: Eshaw, Jcreer

Scatter Point Sets *Source*: http://www.xmswiki.com/xms/index.php?oldid=44129 *Contributors*: Eshaw, Jcreer

Models *Source*: http://www.xmswiki.com/xms/index.php?oldid=42215 *Contributors*: Jcreer, Wood

Model Selection *Source*: http://www.xmswiki.com/xms/index.php?oldid=25878 *Contributors*: Eshaw

How to select a model *Source*: http://www.xmswiki.com/xms/index.php?oldid=28643 *Contributors*: Jdye

Louisiana DOTD Modeling Overview *Source*: http://www.xmswiki.com/xms/index.php?oldid=45301 *Contributors*: Cmsmemoe, Jcreer

Maricopa County Modeling *Source*: http://www.xmswiki.com/xms/index.php?oldid=44189 *Contributors*: Cmsmemoe, Jcreer

SWMM *Source*: http://www.xmswiki.com/xms/index.php?oldid=44186 *Contributors*: Cmsmemoe, Jcreer

Stochastic Modeling *Source*: http://www.xmswiki.com/xms/index.php?oldid=44999 *Contributors*: Eshaw, Jcreer

CE-QUAL-W2 *Source*: http://www.xmswiki.com/xms/index.php?oldid=46477 *Contributors*: Cmsmemoe, Jcreer, Ndye

CE-QUAL-W2 Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=46202 *Contributors*: Cmsmemoe, Jcreer

CE-QUAL-W2 Branches *Source*: http://www.xmswiki.com/xms/index.php?oldid=46497 *Contributors*: Eshaw, Jcreer

CE-QUAL-W2 Layer Editor *Source*: http://www.xmswiki.com/xms/index.php?oldid=43493 *Contributors*: Eshaw, Jcreer

CE-QUAL-W2 Bathymetry *Source*: http://www.xmswiki.com/xms/index.php?oldid=46495 *Contributors*: Cmsmemoe, Jcreer

GSSHA *Source*: http://www.xmswiki.com/xms/index.php?oldid=46016 *Contributors*: Cbarlow, Jcreer

GSSHA Calibration *Source*: http://www.xmswiki.com/xms/index.php?oldid=43534 *Contributors*: Jcreer

GSSHA Channel Routing *Source*: http://www.xmswiki.com/xms/index.php?oldid=43257 *Contributors*: Jcreer, Mshapiro

GSSHA Contaminants *Source*: http://www.xmswiki.com/xms/index.php?oldid=44704 *Contributors*: Jcreer, Mshapiro

GSSHA Digital Dams *Source*: http://www.xmswiki.com/xms/index.php?oldid=44773 *Contributors*: Cmsmemoe, Jcreer, Mshapiro

GSSHA Embankment Arcs *Source*: http://www.xmswiki.com/xms/index.php?oldid=44774 *Contributors*: Jcreer, Ladamson

GSSHA Feature Arcs *Source*: http://www.xmswiki.com/xms/index.php?oldid=44282 *Contributors*: Jcreer, Mshapiro

GSSHA Feature Nodes *Source*: http://www.xmswiki.com/xms/index.php?oldid=46034 *Contributors*: Jcreer, Mshapiro

GSSHA Feature Polygons *Source*: http://www.xmswiki.com/xms/index.php?oldid=44285 *Contributors*: Cmsmemoe, Jcreer

GSSHA Groundwater *Source*: http://www.xmswiki.com/xms/index.php?oldid=44287 *Contributors*: Jcreer, Mshapiro

GSSHA Job Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=44261 *Contributors*: Jcreer, Mshapiro

GSSHA Join SSURGO Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44288 *Contributors*: Jcreer, Ladamson

GSSHA Lakes and Wetlands *Source*: http://www.xmswiki.com/xms/index.php?oldid=43571 *Contributors*: Cmsmemoe, Jcreer

GSSHA Manage Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=46010 *Contributors*: Cmsmemoe, Eshaw, Jcreer

GSSHA Mapping Tables *Source*: http://www.xmswiki.com/xms/index.php?oldid=44268 *Contributors*: Eshaw, Jcreer

GSSHA Maps *Source*: http://www.xmswiki.com/xms/index.php?oldid=46044 *Contributors*: Cmsmemoe, Jcreer

GSSHA Multiple Scenarios *Source*: http://www.xmswiki.com/xms/index.php?oldid=43575 *Contributors*: Jcreer, Ladamson

GSSHA Nutrients *Source*: http://www.xmswiki.com/xms/index.php?oldid=46025 *Contributors*: Cbarlow, Cmsmemoe, Jcreer

GSSHA Observation Points *Source*: http://www.xmswiki.com/xms/index.php?oldid=44299 *Contributors*: Jcreer, Mshapiro

GSSHA Output Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=44263 *Contributors*: Jcreer, Mshapiro

GSSHA Overland Soil Erosion *Source*: http://www.xmswiki.com/xms/index.php?oldid=44293 *Contributors*: Cmsmemoe, Jcreer

GSSHA Precipitation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44295 *Contributors*: Jcreer, Mshapiro

GSSHA Read Solution *Source*: http://www.xmswiki.com/xms/index.php?oldid=44560 *Contributors*: Eshaw, Jcreer

GSSHA Solution Analysis *Source*: http://www.xmswiki.com/xms/index.php?oldid=46021 *Contributors*: Jcreer, Mshapiro

GSSHA Solution Results *Source*: http://www.xmswiki.com/xms/index.php?oldid=46031 *Contributors*: Eshaw, Jcreer

GSSHA Stream Arcs *Source*: http://www.xmswiki.com/xms/index.php?oldid=43572 *Contributors*: Eshaw, Jcreer

Pipe and Node Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=44701 *Contributors*: Cmsmemoe, Jcreer

Radar Rainfall *Source*: http://www.xmswiki.com/xms/index.php?oldid=44703 *Contributors*: Cmsmemoe, Jcreer

HEC-1 Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=43843 *Contributors*: Eshaw, Jcreer

Basin HEC-1 Cards *Source*: http://www.xmswiki.com/xms/index.php?oldid=46478 *Contributors*: Jcreer

Routing HEC-1 Cards *Source*: http://www.xmswiki.com/xms/index.php?oldid=46480 *Contributors*: Cmsmemoe, Jcreer

Diversion HEC-1 Cards *Source*: http://www.xmswiki.com/xms/index.php?oldid=43873 *Contributors*: Jcreer

HEC-1 Job Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=46170 *Contributors*: Cmsmemoe, Eshaw, Jcreer

HEC-1 Input Hydrograph (QI) *Source*: http://www.xmswiki.com/xms/index.php?oldid=43874 *Contributors*: Eshaw, Jcreer

HEC-1 Model Check *Source*: http://www.xmswiki.com/xms/index.php?oldid=43977 *Contributors*: Eshaw, Jcreer

HEC-1 Observed Hydrograph (QO) *Source*: http://www.xmswiki.com/xms/index.php?oldid=43902 *Contributors*: Eshaw, Jcreer

HEC-1 Reading Existing Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=26167 *Contributors*: Eshaw

HEC-1 Run Analysis *Source*: http://www.xmswiki.com/xms/index.php?oldid=46449 *Contributors*: Eshaw, Jcreer

HEC-1 Saving Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=43328 *Contributors*: Eshaw, Jcreer

Computing Area Between Elevations *Source*: http://www.xmswiki.com/xms/index.php?oldid=43901 *Contributors*: Jcreer, Mshapiro

Weighted Average Precipitation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44097 *Contributors*: Jcreer, Wood

HEC-HMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=46226 *Contributors*: Cmsmemoe, Eshaw, Jcreer

HEC-HMS Base Flow *Source*: http://www.xmswiki.com/xms/index.php?oldid=26420 *Contributors*: Eshaw

HEC-HMS Convert Grids *Source*: http://www.xmswiki.com/xms/index.php?oldid=43592 *Contributors*: Eshaw, Jcreer

HEC-HMS Defining the Meteorological Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=44472 *Contributors*: Eshaw, Jcreer

HEC-HMS Diversions *Source*: http://www.xmswiki.com/xms/index.php?oldid=43589 *Contributors*: Eshaw, Jcreer

HEC-HMS Frequency Storm *Source*: http://www.xmswiki.com/xms/index.php?oldid=26497 *Contributors*: Eshaw

HEC-HMS Gages *Source*: http://www.xmswiki.com/xms/index.php?oldid=43593 *Contributors*: Eshaw, Jcreer

HEC-HMS Job Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=44567 *Contributors*: Eshaw, Jcreer

HEC-HMS Junctions *Source*: http://www.xmswiki.com/xms/index.php?oldid=26505 *Contributors*: Eshaw

HEC-HMS Loss Methods *Source*: http://www.xmswiki.com/xms/index.php?oldid=28897 *Contributors*: Mshapiro

HEC-HMS Post Processing *Source*: http://www.xmswiki.com/xms/index.php?oldid=46242 *Contributors*: Cmsmemoe, Eshaw, Jcreer

HEC-HMS Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=43596 *Contributors*: Eshaw, Jcreer

HEC-HMS Reach Routing *Source*: http://www.xmswiki.com/xms/index.php?oldid=44518 *Contributors*: Eshaw, Jcreer

HEC-HMS Reservoirs *Source*: http://www.xmswiki.com/xms/index.php?oldid=26407 *Contributors*: Eshaw

HEC-HMS Run Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=26371 *Contributors*: Eshaw

HEC-HMS SCS Hypothetical Storm *Source*: http://www.xmswiki.com/xms/index.php?oldid=26484 *Contributors*: Eshaw

HEC-HMS Saving an HMS File *Source*: http://www.xmswiki.com/xms/index.php?oldid=43600 *Contributors*: Eshaw, Jcreer

HEC-HMS Sinks *Source*: http://www.xmswiki.com/xms/index.php?oldid=45202 *Contributors*: Eshaw, Jcreer

HEC-HMS Sources *Source*: http://www.xmswiki.com/xms/index.php?oldid=45199 *Contributors*: Eshaw, Jcreer

HEC-HMS Standard Project Storm *Source*: http://www.xmswiki.com/xms/index.php?oldid=43982 *Contributors*: Eshaw, Jcreer

HEC-HMS Sub-Basins *Source*: http://www.xmswiki.com/xms/index.php?oldid=26422 *Contributors*: Eshaw

HEC-HMS Transform *Source*: http://www.xmswiki.com/xms/index.php?oldid=43980 *Contributors*: Eshaw, Jcreer

HEC-HMS User Hyetograph *Source*: http://www.xmswiki.com/xms/index.php?oldid=43609 *Contributors*: Eshaw, Jcreer

HEC-RAS *Source*: http://www.xmswiki.com/xms/index.php?oldid=44998 *Contributors*: Jcreer, Mshapiro

HEC-RAS Bridges/Culverts *Source*: http://www.xmswiki.com/xms/index.php?oldid=43425 *Contributors*: Cmsmemoe, Jcreer

HEC-RAS Model Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=44997 *Contributors*: Eshaw, Jcreer

HEC-RAS Stochastic Modeling *Source*: http://www.xmswiki.com/xms/index.php?oldid=44196 *Contributors*: Eshaw, Jcreer

HEC-RAS Unsteady Modeling *Source*: http://www.xmswiki.com/xms/index.php?oldid=43404 *Contributors*: Eshaw, Jcreer

Recompute All Stations *Source*: http://www.xmswiki.com/xms/index.php?oldid=45001 *Contributors*: Eshaw, Jcreer

HSPF *Source*: http://www.xmswiki.com/xms/index.php?oldid=44031 *Contributors*: Cmsmemoe, Eshaw, Jcreer

HSPF Automatic Segmentation of Land Segments *Source*: http://www.xmswiki.com/xms/index.php?oldid=44027 *Contributors*: Eshaw, Jcreer

HSPF Edit Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=44070 *Contributors*: Eshaw, Jcreer

HSPF External Sources *Source*: http://www.xmswiki.com/xms/index.php?oldid=44023 *Contributors*: Eshaw, Jcreer

HSPF External Targets *Source*: http://www.xmswiki.com/xms/index.php?oldid=44022 *Contributors*: Eshaw, Jcreer

HSPF Getting Started *Source*: http://www.xmswiki.com/xms/index.php?oldid=44064 *Contributors*: Eshaw, Jcreer

HSPF Global Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=44063 *Contributors*: Eshaw, Jcreer

HSPF Mass Links *Source*: http://www.xmswiki.com/xms/index.php?oldid=44065 *Contributors*: Eshaw, Jcreer

HSPF Modules *Source*: http://www.xmswiki.com/xms/index.php?oldid=44069 *Contributors*: Eshaw, Jcreer

HSPF Reading Existing Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=26516 *Contributors*: Eshaw

HSPF Run Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=36965 *Contributors*: Cmsmemoe, Eshaw

HSPF Saving and Reading HSPF UCI Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=43988 *Contributors*: Eshaw, Jcreer

HSPF Viewing WDM Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=43984 *Contributors*: Eshaw, Jcreer

HSPF WDM File Interaction *Source*: http://www.xmswiki.com/xms/index.php?oldid=26549 *Contributors*: Eshaw

MODRAT *Source*: http://www.xmswiki.com/xms/index.php?oldid=43664 *Contributors*: Disleman, Jcreer

MODRAT WMS Interface *Source*: http://www.xmswiki.com/xms/index.php?oldid=43662 *Contributors*: Eshaw, Jcreer

MODRAT Conveyance Type *Source*: http://www.xmswiki.com/xms/index.php?oldid=43694 *Contributors*: Eshaw, Jcreer

MODRAT Course Overview *Source*: http://www.xmswiki.com/xms/index.php?oldid=43672 *Contributors*: Cbarlow, Jcreer

MODRAT Creating Burned Simulations *Source*: http://www.xmswiki.com/xms/index.php?oldid=43746 *Contributors*: Eshaw, Jcreer

MODRAT Edit Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=43732 *Contributors*: Eshaw, Jcreer

MODRAT Input/Output Hydrographs *Source*: http://www.xmswiki.com/xms/index.php?oldid=43747 *Contributors*: Eshaw, Jcreer

MODRAT Job Control Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=43758 *Contributors*: Eshaw, Jcreer

MODRAT Length *Source*: http://www.xmswiki.com/xms/index.php?oldid=26992 *Contributors*: Eshaw

MODRAT Manning's Roughness Coefficient, n *Source*: http://www.xmswiki.com/xms/index.php?oldid=26989 *Contributors*: Eshaw

MODRAT Numbering the Tree *Source*: http://www.xmswiki.com/xms/index.php?oldid=43741 *Contributors*: Jcreer, Mshapiro

MODRAT Relief Drains *Source*: http://www.xmswiki.com/xms/index.php?oldid=43679 *Contributors*: Eshaw, Jcreer

MODRAT Reservoirs *Source*: http://www.xmswiki.com/xms/index.php?oldid=43748 *Contributors*: Eshaw, Jcreer

MODRAT Run Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=43749 *Contributors*: Eshaw, Jcreer

MODRAT Slope *Source*: http://www.xmswiki.com/xms/index.php?oldid=26991 *Contributors*: Eshaw

Using GIS Data to Map Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=43755 *Contributors*: Eshaw, Jcreer

Saving and Reading MODRAT Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=43673 *Contributors*: Eshaw, Jcreer

NSS *Source*: http://www.xmswiki.com/xms/index.php?oldid=33801 *Contributors*: Wood

NSS Assigning Regression Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=43381 *Contributors*: Cmsmemoe, Jcreer

NSS Computing Peak Discharges *Source*: http://www.xmswiki.com/xms/index.php?oldid=44091 *Contributors*: Cmsmemoe, Jcreer

NSS Hydrographs *Source*: http://www.xmswiki.com/xms/index.php?oldid=28149 *Contributors*: Cmsmemoe

NSS Maximum Flood Region *Source*: http://www.xmswiki.com/xms/index.php?oldid=28137 *Contributors*: Cmsmemoe

NSS Model Troubleshooting *Source*: http://www.xmswiki.com/xms/index.php?oldid=43378 *Contributors*: Cbarlow, Jcreer

NSS Overlapping Regions *Source*: http://www.xmswiki.com/xms/index.php?oldid=43385 *Contributors*: Cmsmemoe, Jcreer

NSS State *Source*: http://www.xmswiki.com/xms/index.php?oldid=43383 *Contributors*: Cmsmemoe, Jcreer

NSS State and National Urban Equations *Source*: http://www.xmswiki.com/xms/index.php?oldid=43393 *Contributors*: Jcreer

NSS Total Basin Area *Source*: http://www.xmswiki.com/xms/index.php?oldid=43377 *Contributors*: Cmsmemoe, Jcreer

Saving and Restoring a Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=43384 *Contributors*: Cmsmemoe, Jcreer

Orange County Unit Hydrograph *Source*: http://www.xmswiki.com/xms/index.php?oldid=44080 *Contributors*: Eshaw, Jcreer

OC Unit Hydro Design Storms *Source*: http://www.xmswiki.com/xms/index.php?oldid=44079 *Contributors*: Eshaw, Jcreer

OC Unit Hydro Effective Precipitation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44108 *Contributors*: Eshaw, Jcreer

OC Unit Hydro Job Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=44078 *Contributors*: Eshaw, Jcreer

OC Unit Hydro Post Processing *Source*: http://www.xmswiki.com/xms/index.php?oldid=45165 *Contributors*: Eshaw, Jcreer

OC Unit Hydro Saving a Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44074 *Contributors*: Eshaw, Jcreer

OC Unit Hydro Small Area Hydrographs *Source*: http://www.xmswiki.com/xms/index.php?oldid=44109 *Contributors*: Eshaw, Jcreer

OC Unit Hydro Unit Hydrograph Method *Source*: http://www.xmswiki.com/xms/index.php?oldid=45226 *Contributors*: Eshaw, Jcreer

OC Unit Hydro Using GIS Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44071 *Contributors*: Jcreer, Mshapiro

Orage County Hydrology *Source*: http://www.xmswiki.com/xms/index.php?oldid=27058 *Contributors*: Eshaw

Orange County Rational Method *Source*: http://www.xmswiki.com/xms/index.php?oldid=27107 *Contributors*: Eshaw

OC Rational Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=43925 *Contributors*: Eshaw, Jcreer

OC Losses *Source*: http://www.xmswiki.com/xms/index.php?oldid=43927 *Contributors*: Eshaw, Jcreer

OC Rational Initial Sub-area Time of Concentration *Source*: http://www.xmswiki.com/xms/index.php?oldid=43932 *Contributors*: Eshaw, Jcreer

OC Rational Job Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=43959 *Contributors*: Cmsmemoe, Jcreer

OC Rational Post Processing *Source*: http://www.xmswiki.com/xms/index.php?oldid=43921 *Contributors*: Eshaw, Jcreer

OC Rational Reach Routing *Source*: http://www.xmswiki.com/xms/index.php?oldid=43958 *Contributors*: Eshaw, Jcreer

OC Rational Reports *Source*: http://www.xmswiki.com/xms/index.php?oldid=43915 *Contributors*: Eshaw, Jcreer

OC Rational Run Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=43913 *Contributors*: Eshaw, Jcreer

OC Rational Saving a Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=43934 *Contributors*: Eshaw, Jcreer

OC Rational Sub-area Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=45225 *Contributors*: Eshaw, Jcreer

OC Rational Tree Mapping *Source*: http://www.xmswiki.com/xms/index.php?oldid=43933 *Contributors*: Eshaw, Jcreer

Rational Method *Source*: http://www.xmswiki.com/xms/index.php?oldid=45220 *Contributors*: Eshaw, Jcreer

Rational Method Basin Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=45210 *Contributors*: Jcreer, Mshapiro

Rational Method Computing Hydrographs *Source*: http://www.xmswiki.com/xms/index.php?oldid=44084 *Contributors*: Eshaw, Jcreer

Rational Method Edit Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=45211 *Contributors*: Jcreer, Mshapiro

Rational Method Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=45206 *Contributors*: Eshaw, Jcreer

Rational Method Outlet Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44100 *Contributors*: Eshaw, Jcreer

Rational Method Rainfall Intensity *Source*: http://www.xmswiki.com/xms/index.php?oldid=45212 *Contributors*: Cmsmemoe, Jcreer, Ndye

Rational Method Traditional vs Route by Summing *Source*: http://www.xmswiki.com/xms/index.php?oldid=45214 *Contributors*: Eshaw, Jcreer

Dekalb Rational Hydrograph *Source*: http://www.xmswiki.com/xms/index.php?oldid=45208 *Contributors*: Eshaw, Jcreer

Important Limitations *Source*: http://www.xmswiki.com/xms/index.php?oldid=27122 *Contributors*: Eshaw

Modified Rational Hydrograph *Source*: http://www.xmswiki.com/xms/index.php?oldid=45215 *Contributors*: Eshaw, Jcreer

Rational Hydrograph *Source*: http://www.xmswiki.com/xms/index.php?oldid=45216 *Contributors*: Eshaw, Jcreer

Runoff Coefficient Table *Source*: http://www.xmswiki.com/xms/index.php?oldid=27123 *Contributors*: Eshaw

Universal Rational Hydrograph *Source*: http://www.xmswiki.com/xms/index.php?oldid=45218 *Contributors*: Jcreer, Mshapiro

User Defined Rational Hydrograph *Source*: http://www.xmswiki.com/xms/index.php?oldid=45219 *Contributors*: Eshaw, Jcreer

Storm Drain *Source*: http://www.xmswiki.com/xms/index.php?oldid=46133 *Contributors*: Cbarlow, Jcreer

Storm Drain Job Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=43390 *Contributors*: Jcreer, Mshapiro

SMPDBK *Source*: http://www.xmswiki.com/xms/index.php?oldid=46573 *Contributors*: Eshaw, Jcreer

Defining Roughness *Source*: http://www.xmswiki.com/xms/index.php?oldid=46568 *Contributors*: Eshaw, Jcreer

SMPDBK Post Processing *Source*: http://www.xmswiki.com/xms/index.php?oldid=46564 *Contributors*: Eshaw, Jcreer

TR-20 *Source*: http://www.xmswiki.com/xms/index.php?oldid=45201 *Contributors*: Eshaw, Jcreer

TR-20 Edit Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=44382 *Contributors*: Eshaw, Jcreer

TR-20 Basin Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44434 *Contributors*: Eshaw, Jcreer

TR-20 Diversion Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44499 *Contributors*: Eshaw, Jcreer

TR-20 Reservoir Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44502 *Contributors*: Eshaw, Jcreer

TR-20 Routing Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=44497 *Contributors*: Eshaw, Jcreer

TR-20 Job Control Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=44423 *Contributors*: Eshaw, Jcreer

TR-20 Output Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=27214 *Contributors*: Eshaw

TR-55 *Source*: http://www.xmswiki.com/xms/index.php?oldid=46424 *Contributors*: Cmsmemoe, Jcreer

TR-55 Basin Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=46369 *Contributors*: Cmsmemoe, Eshaw, Jcreer

TR-55 Outlet Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=28180 *Contributors*: Eshaw

TR-55 Hydrographs *Source*: http://www.xmswiki.com/xms/index.php?oldid=46422 *Contributors*: Cmsmemoe, Jcreer

TR-55 Channel Travel Time *Source*: http://www.xmswiki.com/xms/index.php?oldid=46358 *Contributors*: Cmsmemoe, Eshaw

TR-55 Edit Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=44339 *Contributors*: Eshaw, Jcreer

Overview of Map Data Equations *Source*: http://www.xmswiki.com/xms/index.php?oldid=46363 *Contributors*: Cmsmemoe, Eshaw, Jcreer

Travel Times from Map Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=46362 *Contributors*: Cmsmemoe, Eshaw, Jcreer

User Defined Time of Concentration *Source*: http://www.xmswiki.com/xms/index.php?oldid=46488 *Contributors*: Eshaw, Jcreer

FHWA Sheet Flow Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=46554 *Contributors*: Eshaw, Jcreer

FHWA Shallow Concentrated Flow *Source*: http://www.xmswiki.com/xms/index.php?oldid=44353 *Contributors*: Eshaw, Jcreer

FHWA Channel Flow *Source*: http://www.xmswiki.com/xms/index.php?oldid=44357 *Contributors*: Eshaw, Jcreer

TR-55 Sheet Flow Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=46349 *Contributors*: Cmsmemoe, Eshaw, Jcreer

TR-55 Shallow Concentrated Flow Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=46420 *Contributors*: Cmsmemoe, Eshaw, Jcreer

TR-55 Channel Flow Equation *Source*: http://www.xmswiki.com/xms/index.php?oldid=46419 *Contributors*: Cmsmemoe, Eshaw, Jcreer

Maricopa County Time of Concentration *Source*: http://www.xmswiki.com/xms/index.php?oldid=46553 *Contributors*: Eshaw, Jcreer

What's new in WMS version 8.1 *Source*: http://www.xmswiki.com/xms/index.php?oldid=32454 *Contributors*: Cmsmemoe

What's new in WMS version 8.2 *Source*: http://www.xmswiki.com/xms/index.php?oldid=29341 *Contributors*: Cmsmemoe

What's new in WMS version 8.3 *Source*: http://www.xmswiki.com/xms/index.php?oldid=31595 *Contributors*: Cmsmemoe

What's new in WMS version 8.4 *Source*: http://www.xmswiki.com/xms/index.php?oldid=32997 *Contributors*: Cmsmemoe

What's new in WMS version 8.5 *Source*: http://www.xmswiki.com/xms/index.php?oldid=36540 *Contributors*: Cmsmemoe

What's new in WMS version 9.0 *Source*: http://www.xmswiki.com/xms/index.php?oldid=36947 *Contributors*: Cmsmemoe, Mmedina

Path of Data from WMS To HY8 *Source*: http://www.xmswiki.com/xms/index.php?oldid=34659 *Contributors*: EJones

HY8 Modeling Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=34645 *Contributors*: Cmsmemoe, EJones

Project Filename *Source*: http://www.xmswiki.com/xms/index.php?oldid=44952 *Contributors*: Cmsmemoe, Jcreer

Define Project Bounds *Source*: http://www.xmswiki.com/xms/index.php?oldid=33194 *Contributors*: Cmsmemoe

Watershed Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=33195 *Contributors*: Cmsmemoe

Define Culvert Roadway Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=34663 *Contributors*: Cmsmemoe, EJones

Compute Flow Directions and Flow Accumulations *Source*: http://www.xmswiki.com/xms/index.php?oldid=33198 *Contributors*: Cmsmemoe

Choose Outlet Locations *Source*: http://www.xmswiki.com/xms/index.php?oldid=33233 *Contributors*: Cmsmemoe

Delineate Watershed *Source*: http://www.xmswiki.com/xms/index.php?oldid=42821 *Contributors*: Cmsmemoe, Jcreer

Select Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=33200 *Contributors*: Cmsmemoe

HMW Job Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=33201 *Contributors*: Cmsmemoe

Define Land Use and Soil Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=33202 *Contributors*: Cmsmemoe

Hydrologic Computations *Source*: http://www.xmswiki.com/xms/index.php?oldid=33203 *Contributors*: Cmsmemoe

Define Precipitation *Source*: http://www.xmswiki.com/xms/index.php?oldid=33204 *Contributors*: Cmsmemoe

Clean Up Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=33205 *Contributors*: Cmsmemoe

Run Hydrologic Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=33235 *Contributors*: Cmsmemoe

Crossing Discharge Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=34665 *Contributors*: Cmsmemoe, EJones

Setup Tailwater Channel *Source*: http://www.xmswiki.com/xms/index.php?oldid=34664 *Contributors*: Cmsmemoe, EJones

Culvert and Site Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=33250 *Contributors*: Cmsmemoe

Run Culvert Analysis *Source*: http://www.xmswiki.com/xms/index.php?oldid=33251 *Contributors*: Cmsmemoe

Define Flood Inundation Polygon *Source*: http://www.xmswiki.com/xms/index.php?oldid=33252 *Contributors*: Cmsmemoe

Storage Capacity Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=33256 *Contributors*: Cmsmemoe

Define Upstream Channel *Source*: http://www.xmswiki.com/xms/index.php?oldid=33257 *Contributors*: Cmsmemoe

Delineate Inundated Area *Source*: http://www.xmswiki.com/xms/index.php?oldid=33258 *Contributors*: Cmsmemoe

Define and Smooth Streams *Source*: http://www.xmswiki.com/xms/index.php?oldid=43105 *Contributors*: Cmsmemoe, Jcreer

Image Sources, Licenses and Contributors

Image:AddDataCommand.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:AddDataCommand.png *License*: unknown *Contributors*: Cmsmemoe
Image:RasterDisplayAndElevationMenu.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:RasterDisplayAndElevationMenu.png *License*: unknown *Contributors*: Cmsmemoe
Image:VectorDataConvert.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:VectorDataConvert.png *License*: unknown *Contributors*: Cmsmemoe
Image:RasterDataExport.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:RasterDataExport.png *License*: unknown *Contributors*: Cmsmemoe
Image:VectorDataExport.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:VectorDataExport.png *License*: unknown *Contributors*: Cmsmemoe
Image:LandXMLFile.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:LandXMLFile.png *License*: unknown *Contributors*: Cmsmemoe
Image:MultipleDEMs.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:MultipleDEMs.png *License*: unknown *Contributors*: Cmsmemoe
Image:OnlineDataTools.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:OnlineDataTools.png *License*: unknown *Contributors*: Cmsmemoe
Image:VirtualEarth.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:VirtualEarth.png *License*: unknown *Contributors*: Cmsmemoe
Image:GetDataDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GetDataDialog.png *License*: unknown *Contributors*: Cmsmemoe
Image:TextureMapping.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:TextureMapping.png *License*: unknown *Contributors*: Cmsmemoe
Image:GSSHASnowmelt.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHASnowmelt.png *License*: unknown *Contributors*: Cmsmemoe
Image:SWMMAttributes.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SWMMAttributes.png *License*: unknown *Contributors*: Cmsmemoe
Image:WelcomeToWMS.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WelcomeToWMS.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:RegisterWMS.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:RegisterWMS.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:EnableWizard.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:EnableWizard.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:EnableWizard2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:EnableWizard2.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:EnableWizard1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:EnableWizard1.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:image181.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image181.jpg *License*: unknown *Contributors*: Cballard
Image:image356.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image356.gif *License*: unknown *Contributors*: Cballard
Image:image175.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image175.jpg *License*: unknown *Contributors*: Cballard
Image:image190.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image190.gif *License*: unknown *Contributors*: Cballard
Image:image192.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image192.gif *License*: unknown *Contributors*: Cballard
Image:image194.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image194.gif *License*: unknown *Contributors*: Cballard
Image:image195.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image195.gif *License*: unknown *Contributors*: Cballard
Image:image57.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image57.jpg *License*: unknown *Contributors*: Eshaw
Image:image66.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image66.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Menu_Bar.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Menu_Bar.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage176.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage176.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSImage178.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage178.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Terrain_Data_Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Terrain_Data_Module.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Drainage_Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Drainage_Module.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Terrain_Data_Palette.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Terrain_Data_Palette.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Drainage_Palette.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Drainage_Palette.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Map_Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Map_Module.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Hydrologic_Modeling_Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Hydrologic_Modeling_Module.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Map_Palette.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Map_Palette.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Hydrologic_Modeling_Palette.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Hydrologic_Modeling_Palette.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_River_Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_River_Module.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_GIS_Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_GIS_Module.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_River_Palette.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_River_Palette.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_GIS_Palette.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_GIS_Palette.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_2D_Grid_Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_2D_Grid_Module.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_2D_Scatter_Point_Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_2D_Scatter_Point_Module.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_2D_Grid_Palette.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_2D_Grid_Palette.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_2D_Scatter_Point_Palette.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_2D_Scatter_Point_Palette.png *License*: unknown *Contributors*: Eshaw
Image:image180.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image180.jpg *License*: unknown *Contributors*: Cballard
Image:WMSIcon_Macros1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Macros1.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Macros2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Macros2.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage012.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage012.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage009.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage009.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage010.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage010.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage011.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage011.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Delete.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Delete.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage373.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage373.png *License*: unknown *Contributors*: Eshaw
Image:WMSDigitize.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSDigitize.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSUseDigitizeToolbar.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSUseDigitizeToolbar.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSIcon_Pan.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Pan.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Zoom.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Zoom.png *License*: unknown *Contributors*: Eshaw
Image:WMSimage340.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage340.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage341.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage341.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Frame.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Frame.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Previous_View.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Previous_View.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Rotate.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Rotate.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Contour_Labels.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Contour_Labels.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Select_Drainage_Unit.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Drainage_Unit.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Move_Basin_Labels.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Move_Basin_Labels.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Flow_Path.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Flow_Path.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Create_Text.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Text.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Create Rectangle.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Rectangle.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Create Oval.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Oval.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Create Line.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Line.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Select Drawing Objects.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Drawing_Objects.png *License*: unknown *Contributors*: Eshaw

Image:GetDataToolbar1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GetDataToolbar1.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSIcon_Measure_Tool.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Measure_Tool.png *License*: unknown *Contributors*: Eshaw

Image:measureangle.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Measureangle.jpg *License*: unknown *Contributors*: Cballard

Image:HydrologicModelingWizardTool.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardTool.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:WMSImage273.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage273.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage161.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage161.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage270.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage270.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_New.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_New.png *License*: unknown *Contributors*: Eshaw

Image:image051.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image051.jpg *License*: unknown *Contributors*: Cballard

Image:image68.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image68.jpg *License*: unknown *Contributors*: Cballard

Image:image055.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image055.jpg *License*: unknown *Contributors*: Cballard

Image:image69.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image69.jpg *License*: unknown *Contributors*: Cballard

Image:image67.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image67.jpg *License*: unknown *Contributors*: Cballard

Image:image434.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image434.gif *License*: unknown *Contributors*: Cballard

Image:ellipsiodcoordsys.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ellipsiodcoordsys.jpg *License*: unknown *Contributors*: Ksmith

Image:cppcoordsys1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys1.jpg *License*: unknown *Contributors*: Ksmith

Image:cppcoordsys2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys2.jpg *License*: unknown *Contributors*: Ksmith

Image:cppcoordsys3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys3.jpg *License*: unknown *Contributors*: Ksmith

Image:cppcoordsys4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys4.jpg *License*: unknown *Contributors*: Ksmith

Image:WMSImage363.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage363.gif *License*: unknown *Contributors*: Eshaw

Image:WMSimage173.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage173.jpg *License*: unknown *Contributors*: Eshaw

Image:image267.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image267.gif *License*: unknown *Contributors*: Eshaw

Image:image283.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image283.jpg *License*: unknown *Contributors*: Eshaw

Image:image284.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image284.jpg *License*: unknown *Contributors*: Eshaw

Image:image285.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image285.jpg *License*: unknown *Contributors*: Eshaw

Image:image286.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image286.jpg *License*: unknown *Contributors*: Eshaw

Image:image298.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image298.jpg *License*: unknown *Contributors*: Eshaw

Image:image352.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image352.gif *License*: unknown *Contributors*: Eshaw

Image:image351.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image351.gif *License*: unknown *Contributors*: Eshaw

Image:image350.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image350.gif *License*: unknown *Contributors*: Eshaw

Image:image349.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image349.gif *License*: unknown *Contributors*: Eshaw

Image:image140.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image140.jpg *License*: unknown *Contributors*: Eshaw

Image:image141.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image141.jpg *License*: unknown *Contributors*: Eshaw

File:image172.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image172.jpg *License*: unknown *Contributors*: Eshaw

File:image344.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image344.gif *License*: unknown *Contributors*: Cballard

File:image345.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image345.gif *License*: unknown *Contributors*: Cballard

File:image346.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image346.gif *License*: unknown *Contributors*: Cballard

File:image169.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image169.jpg *License*: unknown *Contributors*: Eshaw

File:image348.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image348.gif *License*: unknown *Contributors*: Cballard

File:WMSImage354.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage354.gif *License*: unknown *Contributors*: Eshaw

Image:WMSImage_Icon_ContLabel.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage_Icon_ContLabel.png *License*: unknown *Contributors*: Eshaw

File:WMSImage353.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage353.gif *License*: unknown *Contributors*: Eshaw

File:image347.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image347.gif *License*: unknown *Contributors*: Cballard

Image:DisplayOrderOffPlanView.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:DisplayOrderOffPlanView.png *License*: unknown *Contributors*: Cmsmemoe

Image:DisplayOrderOnPlanView.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:DisplayOrderOnPlanView.png *License*: unknown *Contributors*: Cmsmemoe

Image:DisplayOrderOnObliqueView.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:DisplayOrderOnObliqueView.png *License*: unknown *Contributors*: Cmsmemoe

Image:DisplayOrderOffObliqueView.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:DisplayOrderOffObliqueView.png *License*: unknown *Contributors*: Cmsmemoe

Image:image124.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image124.jpg *License*: unknown *Contributors*: Cballard

Image:WMSChannelCalculations.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSChannelCalculations.png *License*: unknown *Contributors*: EJones

Image:WMSChannelCalculator.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSChannelCalculator.png *License*: unknown *Contributors*: EJones

File:Image330.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image330.jpg *License*: unknown *Contributors*: Jcreer

Image:Data_Calculator.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Data_Calculator.jpg *License*: unknown *Contributors*: Rjones

Image:DatasetZonalClassification1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DatasetZonalClassification1.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:DatasetZonalClassification2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DatasetZonalClassification2.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:DatasetZonalClassificationDisplay1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DatasetZonalClassificationDisplay1.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:WMSImage4.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage4.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage7.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage7.png *License*: unknown *Contributors*: Eshaw

Image:WMSimage147.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage147.gif *License*: unknown *Contributors*: Eshaw

Image:HydrologicModelingWizardProjectFilename.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardProjectFilename.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardDefineProjectBounds.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardDefineProjectBounds.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardWatershedData.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardWatershedData.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardDownloadDataWebService.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardDownloadDataWebService.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardReadDataCatalog.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardReadDataCatalog.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardProjectComputeFlowDirections.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardProjectComputeFlowDirections.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardChooseOutletLocations.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardChooseOutletLocations.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardDelineateWatershed.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardDelineateWatershed.png *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardSelectModel.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardSelectModel.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardDefineAndSmoothStreams.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardDefineAndSmoothStreams.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardCreate2DGrid.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardCreate2DGrid.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardJobControl.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardJobControl.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardDefineLandUseSoilData.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardDefineLandUseSoilData.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardHydrologicComputations.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardHydrologicComputations.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardDefinePrecipitation.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardDefinePrecipitation.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardCleanUpModel.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardCleanUpModel.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:image159.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image159.jpg *License*: unknown *Contributors*: Cballard

Image:AnnotationTree.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:AnnotationTree.jpg *License*: unknown *Contributors*: Djordan

Image:Scale.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scale.jpg *License*: unknown *Contributors*: Sramiah

Image:North-arrow.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:North-arrow.jpg *License*: unknown *Contributors*: Ksmith

Image:Dialog_Hydrograph_Table.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_Hydrograph_Table.png *License*: unknown *Contributors*: Eshaw

Image:GsshaGw_b.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaGw_b.png *License*: unknown *Contributors*: Cmsmemoe

Image:GsshaSmoothStreams_b.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaSmoothStreams_b.png *License*: unknown *Contributors*: Cmsmemoe

Image:Gssha14x9 1b.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gssha14x9_1b.PNG *License*: unknown *Contributors*: Mmedina

Image:Google1.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Google1.PNG *License*: unknown *Contributors*: Mmedina

Image:GsshaGw_w.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaGw_w.png *License*: unknown *Contributors*: Cmsmemoe

Image:GsshaSmoothStreams_w.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaSmoothStreams_w.png *License*: unknown *Contributors*: Cmsmemoe

Image:Gssha14x9 1w.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gssha14x9_1w.PNG *License*: unknown *Contributors*: Mmedina

Image:Hmswms1.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hmswms1.PNG *License*: unknown *Contributors*: Mmedina

Image:Hy8wms1.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hy8wms1.PNG *License*: unknown *Contributors*: Mmedina

Image:Wmsras1.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Wmsras1.PNG *License*: unknown *Contributors*: Mmedina

Image:Hydtoolbox1.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hydtoolbox1.PNG *License*: unknown *Contributors*: Mmedina

Image:Hydtoolbox2.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hydtoolbox2.PNG *License*: unknown *Contributors*: Mmedina

Image:Hydtoolbox3.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hydtoolbox3.PNG *License*: unknown *Contributors*: Mmedina

Image:Hydtoolbox4.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hydtoolbox4.PNG *License*: unknown *Contributors*: Mmedina

Image:Onlineimages14x9 1.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Onlineimages14x9_1.PNG *License*: unknown *Contributors*: Mmedina

Image:Onlineimages14x9 2.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Onlineimages14x9_2.PNG *License*: unknown *Contributors*: Mmedina

Image:SelectRegistrationPoint.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SelectRegistrationPoint.jpg *License*: unknown *Contributors*: Abarney

Image:Zoom.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Zoom.JPG *License*: unknown *Contributors*: Jdhowlett

Image:Pan.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Pan.JPG *License*: unknown *Contributors*: Jdhowlett

Image:Frame.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Frame.JPG *License*: unknown *Contributors*: Jdhowlett

Image:TI_1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TI_1.jpg *License*: unknown *Contributors*: Cballard

Image:TI_2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TI_2.jpg *License*: unknown *Contributors*: Cballard

Image:image145.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image145.jpg *License*: unknown *Contributors*: Cballard

Image:image147.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image147.jpg *License*: unknown *Contributors*: Cballard

Image:image158.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image158.jpg *License*: unknown *Contributors*: Cballard

Image:image93.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image93.jpg *License*: unknown *Contributors*: Cballard

Image:image117.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image117.jpg *License*: unknown *Contributors*: Cballard

Image:image122.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image122.jpg *License*: unknown *Contributors*: Cballard

Image:image267.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image267.jpg *License*: unknown *Contributors*: Eshaw

Image:linear01.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Linear01.gif *License*: unknown *Contributors*: Cballard

Image:WMScttriangle.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMScttriangle.gif *License*: unknown *Contributors*: Eshaw

File:Thiessen2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Thiessen2.jpg *License*: unknown *Contributors*: Jcreer

File:Localcoord2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Localcoord2.jpg *License*: unknown *Contributors*: Jcreer

Image:image296.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image296.gif *License*: unknown *Contributors*: Cballard

Image:image108.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image108.jpg *License*: unknown *Contributors*: Cballard

Image:image297.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image297.gif *License*: unknown *Contributors*: Cballard

Image:image107.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image107.jpg *License*: unknown *Contributors*: Eshaw

Image:WMSidw_fig2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSidw_fig2.jpg *License*: unknown *Contributors*: Jcreer

Image:WMSidw_fig3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSidw_fig3.jpg *License*: unknown *Contributors*: Jcreer

Image:WMSidw_fig1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSidw_fig1.gif *License*: unknown *Contributors*: Eshaw

Image:image278.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image278.jpg *License*: unknown *Contributors*: Cballard

Image:image40.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image40.jpg *License*: unknown *Contributors*: Cballard
Image:WMSIcon_Map Module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Map_Module.png *License*: unknown *Contributors*: Eshaw
Image:DEM_Guidelines_1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DEM_Guidelines_1.jpg *License*: unknown *Contributors*: Eshaw
Image:DEM_Guidelines_2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:DEM_Guidelines_2.gif *License*: unknown *Contributors*: Eshaw
Image:DEM_Guidelines_3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DEM_Guidelines_3.jpg *License*: unknown *Contributors*: Eshaw
Image:DEM_Guidelines_4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DEM_Guidelines_4.jpg *License*: unknown *Contributors*: Eshaw
Image:DEM_Guidelines_5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DEM_Guidelines_5.jpg *License*: unknown *Contributors*: Eshaw
Image:DEM_Guidelines_6.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DEM_Guidelines_6.jpg *License*: unknown *Contributors*: Eshaw
Image:DEM_Guidelines_7.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DEM_Guidelines_7.jpg *License*: unknown *Contributors*: Eshaw
Image:DEM_Guidelines_8.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DEM_Guidelines_8.jpg *License*: unknown *Contributors*: Eshaw
Image:image166.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image166.jpg *License*: unknown *Contributors*: Cballard
Image:image52.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image52.gif *License*: unknown *Contributors*: Cballard
Image:WMSImageSetActiveRgn.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImageSetActiveRgn.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage272.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage272.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage0274.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage0274.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_DEM_Points.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_DEM_Points.png *License*: unknown *Contributors*: Eshaw
Image:image77.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image77.jpg *License*: unknown *Contributors*: Cballard
Image:WMSIcon_Select_DEM_Point.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_DEM_Point.png *License*: unknown *Contributors*: Eshaw
Image:image75.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image75.jpg *License*: unknown *Contributors*: Cballard
Image:image78.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image78.jpg *License*: unknown *Contributors*: Cballard
Image:image79.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image79.jpg *License*: unknown *Contributors*: Cballard
Image:WMSImage75.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage75.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSImage80.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage80.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage8.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage8.jpg *License*: unknown *Contributors*: Eshaw
Image:image335.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image335.gif *License*: unknown *Contributors*: Cballard
Image:image337.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image337.gif *License*: unknown *Contributors*: Cballard
Image:image336.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image336.gif *License*: unknown *Contributors*: Cballard
Image:image339.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image339.gif *License*: unknown *Contributors*: Cballard
Image:image338.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image338.gif *License*: unknown *Contributors*: Eshaw
Image:image11.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image11.jpg *License*: unknown *Contributors*: Cballard
Image:image12.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image12.jpg *License*: unknown *Contributors*: Cballard
Image:image15.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image15.jpg *License*: unknown *Contributors*: Cballard
Image:image16.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image16.jpg *License*: unknown *Contributors*: Cballard
Image:image44.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image44.gif *License*: unknown *Contributors*: Cballard
Image:image33.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image33.jpg *License*: unknown *Contributors*: Cballard
Image:image270.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image270.gif *License*: unknown *Contributors*: Cballard
Image:image31.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image31.jpg *License*: unknown *Contributors*: Eshaw
Image:image32.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image32.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSimage74.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage74.jpg *License*: unknown *Contributors*: Eshaw
Image:image64.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image64.gif *License*: unknown *Contributors*: Cballard
Image:image66.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image66.gif *License*: unknown *Contributors*: Cballard
Image:image72.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image72.jpg *License*: unknown *Contributors*: Cballard
Image:WMSIcon_Select_Verticies.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Verticies.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Triangles.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Triangles.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Create_Breakline.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Breakline.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Swap_Triangle_Edges.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Swap_Triangle_Edges.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Add_Vertices.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Add_Vertices.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Create_Triangle.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Triangle.png *License*: unknown *Contributors*: Eshaw
Image:image200.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image200.gif *License*: unknown *Contributors*: Cballard
Image:image325.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image325.png *License*: unknown *Contributors*: Cmsmemoe
Image:image323.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image323.png *License*: unknown *Contributors*: Cmsmemoe
Image:image203.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image203.png *License*: unknown *Contributors*: Cmsmemoe
Image:image322.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image322.png *License*: unknown *Contributors*: Cmsmemoe
Image:image30.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image30.gif *License*: unknown *Contributors*: Cballard
Image:image31.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image31.gif *License*: unknown *Contributors*: Cballard
Image:image334.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image334.gif *License*: unknown *Contributors*: Cballard
Image:WMSIcon_Select_Feature_Objects.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Feature_Objects.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Feature_Point.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Feature_Point.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Feature_Vertex.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Feature_Vertex.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Feature_Arc.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Feature_Arc.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Create_Feature_Point.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Feature_Point.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Create_Feature_Vertex.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Feature_Vertex.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Create_Feature_Arc.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Feature_Arc.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Feature_Polygon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Feature_Polygon.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Feature_Line_Branch.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Feature_Line_Branch.png *License*: unknown *Contributors*: Eshaw
Image:SWMMHydraulicSchematicFromStormDrainConceptualModel.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SWMMHydraulicSchematicFromStormDrainConceptualModel.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:WMSImage196.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage196.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage328.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage328.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage327.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage327.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage326.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage326.gif *License*: unknown *Contributors*: Eshaw

Image:image333.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image333.gif *License*: unknown *Contributors*: Cballard
Image:image332.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image332.gif *License*: unknown *Contributors*: Cballard
Image:image331.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image331.gif *License*: unknown *Contributors*: Cballard
Image:overlay1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Overlay1.jpg *License*: unknown *Contributors*: Eshaw
Image:overlay2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Overlay2.jpg *License*: unknown *Contributors*: Eshaw
Image:overlay3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Overlay3.jpg *License*: unknown *Contributors*: Eshaw
Image:image300.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image300.gif *License*: unknown *Contributors*: Cballard
Image:image104.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image104.jpg *License*: unknown *Contributors*: Cballard
Image:image293.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image293.gif *License*: unknown *Contributors*: Cballard
Image:image110.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image110.jpg *License*: unknown *Contributors*: Cballard
Image:image111.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image111.jpg *License*: unknown *Contributors*: Cballard
Image:image268.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image268.jpg *License*: unknown *Contributors*: Cballard
Image:image423.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image423.gif *License*: unknown *Contributors*: Cballard
Image:image424.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image424.gif *License*: unknown *Contributors*: Cballard
Image:image425.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image425.gif *License*: unknown *Contributors*: Cballard
Image:image426.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image426.gif *License*: unknown *Contributors*: Cballard
Image:image427.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image427.gif *License*: unknown *Contributors*: Cballard
Image:image428.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image428.gif *License*: unknown *Contributors*: Cballard
Image:LandUseAtts.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:LandUseAtts.jpg *License*: unknown *Contributors*: Mshapiro
Image:SoilAtts.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SoilAtts.jpg *License*: unknown *Contributors*: Mshapiro
Image:image204.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image204.jpg *License*: unknown *Contributors*: Cballard
Image:image183.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image183.jpg *License*: unknown *Contributors*: Cballard
Image:feature_object_guidelines_1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Feature_object_guidelines_1.jpg *License*: unknown *Contributors*: Cballard
Image:feature_object_guidelines_2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Feature_object_guidelines_2.jpg *License*: unknown *Contributors*: Cballard
Image:feature_object_guidelines_3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Feature_object_guidelines_3.jpg *License*: unknown *Contributors*: Cballard
Image:image40.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image40.gif *License*: unknown *Contributors*: Cballard
Image:image41.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image41.gif *License*: unknown *Contributors*: Cballard
Image:image42.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image42.gif *License*: unknown *Contributors*: Cballard
Image:image43.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image43.gif *License*: unknown *Contributors*: Cballard
Image:image45.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image45.gif *License*: unknown *Contributors*: Cballard
Image:WMSIcon_Select_Outlet.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Outlet.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Basin.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Basin.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Diversion.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Diversion.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_Hydrograph.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Hydrograph.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage205.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage205.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage206.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage206.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage207.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage207.gif *License*: unknown *Contributors*: Eshaw
Image:image277.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image277.jpg *License*: unknown *Contributors*: Cballard
Image:image279.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image279.jpg *License*: unknown *Contributors*: Cballard
Image:image2760.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image2760.jpg *License*: unknown *Contributors*: Cballard
Image:image144.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image144.gif *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_River_Cross_Section.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_River_Cross_Section.png *License*: unknown *Contributors*: Eshaw
Image:WMSIcon_Select_River_Reach.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_River_Reach.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage295.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage295.png *License*: unknown *Contributors*: Eshaw
Image:SWMMHydraulicSchematicMenuOptions.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SWMMHydraulicSchematicMenuOptions.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:SWMMLinkMenuOptions.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SWMMLinkMenuOptions.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:image109.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image109.jpg *License*: unknown *Contributors*: Cballard
Image:WMSImage112.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage112.jpg *License*: unknown *Contributors*: Eshaw
Image:image126.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image126.jpg *License*: unknown *Contributors*: Cballard
Image:image127.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image127.jpg *License*: unknown *Contributors*: Cballard
Image:WMSImage294.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage294.jpg *License*: unknown *Contributors*: Jcreer
Image:image430.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image430.gif *License*: unknown *Contributors*: Cballard
Image:image431.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image431.gif *License*: unknown *Contributors*: Cballard
Image:image432.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image432.gif *License*: unknown *Contributors*: Cballard
Image:image82.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image82.jpg *License*: unknown *Contributors*: Cballard
Image:image429.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image429.jpg *License*: unknown *Contributors*: Jcreer
Image:WMSIcon_Select_Features.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Features.png *License*: unknown *Contributors*: Eshaw
Image:image59.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image59.jpg *License*: unknown *Contributors*: Cballard
Image:WMSIcon_Identify.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Identify.png *License*: unknown *Contributors*: Eshaw
Image:image58.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image58.jpg *License*: unknown *Contributors*: Cballard
Image:image1740.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image1740.jpg *License*: unknown *Contributors*: Cballard
Image:WMSIcon_Select_Shapes.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Shapes.png *License*: unknown *Contributors*: Eshaw
Image:WMSPointShapefileTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSPointShapefileTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSLineShapefileTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSLineShapefileTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSPolygonShapefileTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSPolygonShapefileTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSGISVectorDataTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSGISVectorDataTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSGISRasterImageTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSGISRasterImageTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSGISPaletteImageTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSGISPaletteImageTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSGISElevationImageTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSGISElevationImageTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSOnlineGISRasterImageTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSOnlineGISRasterImageTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSOnlineGISPaletteImageTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSOnlineGISPaletteImageTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSOnlineGISElevationImageTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSOnlineGISElevationImageTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSArcObjectsPointTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSArcObjectsPointTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSArcObjectsArcTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSArcObjectsArcTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSArcObjectsPolygonTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSArcObjectsPolygonTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSArcObjectsRasterTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSArcObjectsRasterTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMSArcObjectsCadTinTreeIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSArcObjectsCadTinTreeIcon.png *License*: unknown *Contributors*: Cmsmemoe

Image:image214.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image214.gif *License*: unknown *Contributors*: Cballard

Image:image213.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image213.gif *License*: unknown *Contributors*: Cballard

Image:image216.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image216.gif *License*: unknown *Contributors*: Cballard

Image:WMSImage131.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage131.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage132.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage132.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage133.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage133.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage098.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage098.png *License*: unknown *Contributors*: Eshaw

Image:WMSArcHydro001.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSArcHydro001.png *License*: unknown *Contributors*: Eshaw

Image:WMSArcHydro002.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSArcHydro002.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage274.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage274.png *License*: unknown *Contributors*: Eshaw

Image:WMSGeodatabase001.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSGeodatabase001.png *License*: unknown *Contributors*: Eshaw

Image:WMSimage142.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage142.jpg *License*: unknown *Contributors*: Eshaw

Image:GSSHA_Smooth_grid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHA_Smooth_grid.gif *License*: unknown *Contributors*: Cballard

Image:GSSHA_Block_grid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHA_Block_grid.gif *License*: unknown *Contributors*: Cballard

Image:GSSHA_flow_vectors.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHA_flow_vectors.gif *License*: unknown *Contributors*: Cballard

Image:WMSIcon_Select_Cell.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Cell.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Select_i.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_i.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Select_j.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_j.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Create_Gages.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Gages.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Select_Gages.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Gages.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage219.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage219.gif *License*: unknown *Contributors*: Eshaw

Image:WMSImage329.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage329.gif *License*: unknown *Contributors*: Eshaw

Image:WMSImage221.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage221.gif *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Select_Scatter_Point.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Select_Scatter_Point.png *License*: unknown *Contributors*: Eshaw

Image:WMSIcon_Create_Scatter_Data_Point.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSIcon_Create_Scatter_Data_Point.png *License*: unknown *Contributors*: Eshaw

Image:WMSImage209.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage209.gif *License*: unknown *Contributors*: Eshaw

Image:WMSImage321.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage321.gif *License*: unknown *Contributors*: Eshaw

Image:WMSImage324.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage324.gif *License*: unknown *Contributors*: Eshaw

Image:WMSImage212.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage212.gif *License*: unknown *Contributors*: Eshaw

Image:WMSImage374.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage374.png *License*: unknown *Contributors*: Eshaw

File:WMSImage277.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage277.jpg *License*: unknown *Contributors*: Jcreer

Image:image85.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image85.jpg *License*: unknown *Contributors*: Eshaw

Image:WMSImage310.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage310.jpg *License*: unknown *Contributors*: Jcreer

Image:image84.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image84.jpg *License*: unknown *Contributors*: Eshaw

Image:WMSImage311.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage311.jpg *License*: unknown *Contributors*: Jcreer

Image:image86.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image86.jpg *License*: unknown *Contributors*: Eshaw

Image:image144.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image144.jpg *License*: unknown *Contributors*: Eshaw

Image:image83.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image83.jpg *License*: unknown *Contributors*: Eshaw

Image:image91.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image91.jpg *License*: unknown *Contributors*: Eshaw

Image:image330.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image330.gif *License*: unknown *Contributors*: Cballard

Image:image137.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image137.jpg *License*: unknown *Contributors*: Eshaw

Image:image135.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image135.jpg *License*: unknown *Contributors*: Eshaw

Image:image136.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image136.jpg *License*: unknown *Contributors*: Eshaw

Image:AEPMAP.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:AEPMAP.jpg *License*: unknown *Contributors*: Eshaw

Image:WMSImage312.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage312.jpg *License*: unknown *Contributors*: Jcreer

Image:image212.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image212.jpg *License*: unknown *Contributors*: Eshaw

Image:image214.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image214.jpg *License*: unknown *Contributors*: Eshaw

Image:image205.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image205.jpg *License*: unknown *Contributors*: Eshaw

Image:image206.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image206.jpg *License*: unknown *Contributors*: Eshaw

Image:image207.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image207.jpg *License*: unknown *Contributors*: Eshaw

Image:image208.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image208.jpg *License*: unknown *Contributors*: Eshaw

Image:W2Image210.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:W2Image210.jpg *License*: unknown *Contributors*: Eshaw

Image:CEQUALW2BathymetryFig1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig1.png *License*: unknown *Contributors*: Cmsmemoe

Image:CEQUALW2BathymetryFig2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig2.png *License*: unknown *Contributors*: Cmsmemoe

Image:CEQUALW2BathymetryFig3.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig3.png *License*: unknown *Contributors*: Cmsmemoe

Image:CEQUALW2BathymetryFig4.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig4.png *License*: unknown *Contributors*: Cmsmemoe

Image:CEQUALW2BathymetryFig5.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig5.png *License*: unknown *Contributors*: Cmsmemoe

Image:CEQUALW2BathymetryFig6.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig6.png *License*: unknown *Contributors*: Cmsmemoe

Image:CEQUALW2BathymetryFig7.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig7.png *License*: unknown *Contributors*: Cmsmemoe

Image:CEQUALW2BathymetryFig8.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig8.png *License*: unknown *Contributors*: Cmsmemoe
Image:CEQUALW2BathymetryFig9.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig9.png *License*: unknown *Contributors*: Cmsmemoe
Image:CEQUALW2BathymetryFig10.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig10.png *License*: unknown *Contributors*: Cmsmemoe
Image:CEQUALW2BathymetryFig11.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:CEQUALW2BathymetryFig11.png *License*: unknown *Contributors*: Cmsmemoe
Image:Dialog_GSSHA_Model_Check.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Model_Check.png *License*: unknown *Contributors*: Eshaw
Image:Dialog_GSSHA_Run_Options.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Run_Options.png *License*: unknown *Contributors*: Eshaw
Image:CalibParams.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CalibParams.jpg *License*: unknown *Contributors*: Ladamson
Image:MappingCalibParams.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:MappingCalibParams.jpg *License*: unknown *Contributors*: Ladamson
Image:CalibrationObserved.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CalibrationObserved.jpg *License*: unknown *Contributors*: Ladamson
Image:CalibrationSCEParams.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CalibrationSCEParams.jpg *License*: unknown *Contributors*: Ladamson
Image:GSSHA_Contaminants.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHA_Contaminants.jpg *License*: unknown *Contributors*: Mshapiro
Image:GSSHA_Cell_Fill_Block_Style_Digital_Dams.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHA_Cell_Fill_Block_Style_Digital_Dams.jpg *License*: unknown *Contributors*: Cballard
Image:GSSHA_Embankment_Arc.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHA_Embankment_Arc.jpg *License*: unknown *Contributors*: Ladamson
Image:GsshaArcAtts.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaArcAtts.jpg *License*: unknown *Contributors*: Mshapiro
Image:GsshaNodeAtts.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaNodeAtts.jpg *License*: unknown *Contributors*: Mshapiro
Image:GSSHA Lakes and Wetlands.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHA_Lakes_and_Wetlands.jpg *License*: unknown *Contributors*: Eshaw
Image:wetlands1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Wetlands1.png *License*: unknown *Contributors*: Cmsmemoe
Image:GsshaGroundwater.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaGroundwater.jpg *License*: unknown *Contributors*: Mshapiro
Image:Dialog_GSSHA_Job_Control.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Job_Control.png *License*: unknown *Contributors*: Eshaw
Image:ssurgo.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ssurgo.jpg *License*: unknown *Contributors*: Ladamson
Image:Dialog_GSSHA_Save_Project_File.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Save_Project_File.png *License*: unknown *Contributors*: Eshaw
Image:Dialog_GSSHA_Map_Table_Editor.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Map_Table_Editor.png *License*: unknown *Contributors*: Eshaw
Image:Dialog_GSSHA_Maps.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Maps.png *License*: unknown *Contributors*: Eshaw
Image:Dialog GSSHA Maps3.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Maps3.png *License*: unknown *Contributors*: Eshaw
Image:Dialog GSSHA Maps2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Maps2.png *License*: unknown *Contributors*: Eshaw
Image:GSSHAMultipleModels.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHAMultipleModels.png *License*: unknown *Contributors*: Ladamson
File:NutrientsJobControl.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NutrientsJobControl.jpg *License*: unknown *Contributors*: Cbarlow
File:NutrientsDialogOther.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NutrientsDialogOther.jpg *License*: unknown *Contributors*: Cbarlow
File:NutrientsDialogUniformProperties.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NutrientsDialogUniformProperties.jpg *License*: unknown *Contributors*: Cbarlow
File:GSSHAStreamIndexMap.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHAStreamIndexMap.jpg *License*: unknown *Contributors*: Cbarlow
Image:GsshaObservations.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaObservations.jpg *License*: unknown *Contributors*: Mshapiro
Image:Dialog_GSSHA_Output_Control.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Output_Control.png *License*: unknown *Contributors*: Eshaw
Image:Dialog_GSSHA_Overland_Soil_Erosion.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Overland_Soil_Erosion.png *License*: unknown *Contributors*: Cmsmemoe
Image:Dialog_GSSHA_Precipitation.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dialog_GSSHA_Precipitation.png *License*: unknown *Contributors*: Eshaw
Image:GsshaSolutionAnalysis.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaSolutionAnalysis.jpg *License*: unknown *Contributors*: Ladamson
Image:GSSHA_Solution_Data.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHA_Solution_Data.jpg *License*: unknown *Contributors*: Cballard
Image:GsshaPipeNodeParameters.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaPipeNodeParameters.png *License*: unknown *Contributors*: Cmsmemoe
Image:GsshaPipeArcProperties.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaPipeArcProperties.png *License*: unknown *Contributors*: Cmsmemoe
Image:GsshaPipePointProperties.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GsshaPipePointProperties.png *License*: unknown *Contributors*: Cmsmemoe
Image:HMSRadarImage.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMSRadarImage.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:GSSHARadarImage.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GSSHARadarImage.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:image190.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image190.jpg *License*: unknown *Contributors*: Cballard
Image:HEC1OutputControlDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HEC1OutputControlDialog.png *License*: unknown *Contributors*: Cmsmemoe
Image:HEC1JobControlDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HEC1JobControlDialog.png *License*: unknown *Contributors*: Cmsmemoe
Image:WMSImage191.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage191.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSImage192.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage192.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSimage6.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage6.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSimage7.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage7.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSimage8.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage8.jpg *License*: unknown *Contributors*: Eshaw
Image:HMSConvertGrids1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMSConvertGrids1.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:image5.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image5.gif *License*: unknown *Contributors*: Eshaw
Image:WMSimage19.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage19.jpg *License*: unknown *Contributors*: Eshaw
Image:HMSFrequencyStorm1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMSFrequencyStorm1.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HMS_Gages1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMS_Gages1.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HMS_Gages2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMS_Gages2.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HMS_Gages3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMS_Gages3.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HMS_Gages4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMS_Gages4.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HMS_Gages5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMS_Gages5.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:WMSImage_HMS_Job_Control.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage_HMS_Job_Control.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage_HMS_Basin_Options.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage_HMS_Basin_Options.png *License*: unknown *Contributors*: Eshaw
Image:HMSMeteorologicalOptions1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMSMeteorologicalOptions1.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:WMSimage4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage4.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSimage5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSimage5.jpg *License*: unknown *Contributors*: Eshaw
Image:image198.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image198.jpg *License*: unknown *Contributors*: Eshaw
Image:image195.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image195.jpg *License*: unknown *Contributors*: Eshaw
Image:image3.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image3.gif *License*: unknown *Contributors*: Eshaw
Image:HMSSCSHypotheticalStorm1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMSSCSHypotheticalStorm1.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HMSSCSHypotheticalStorm2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMSSCSHypotheticalStorm2.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HMSStandardProjectStorm1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMSStandardProjectStorm1.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HMSUserHyetograph1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HMSUserHyetograph1.jpg *License*: unknown *Contributors*: Cmsmemoe

Image:WMSImage51.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage51.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSImage199.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage199.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSImage52.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage52.jpg *License*: unknown *Contributors*: Eshaw
Image:image202.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image202.jpg *License*: unknown *Contributors*: Cballard
Image:image201.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image201.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSMODRATRenumber.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSMODRATRenumber.jpg *License*: unknown *Contributors*: Eshaw
Image:MODRATMapAtts.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:MODRATMapAtts.jpg *License*: unknown *Contributors*: Cballard
Image:image125.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image125.jpg *License*: unknown *Contributors*: Eshaw
Image:image89.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image89.gif *License*: unknown *Contributors*: Cballard
Image:OCUnitHydroEditParam.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:OCUnitHydroEditParam.jpg *License*: unknown *Contributors*: Eshaw
Image:WMStc.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMStc.jpg *License*: unknown *Contributors*: Eshaw
Image:job_control.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Job_control.JPG *License*: unknown *Contributors*: Eshaw
Image:WMSmain.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSmain.jpg *License*: unknown *Contributors*: Eshaw
Image:WMStree_mapping.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMStree_mapping.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSImage73.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage73.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage203.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage203.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSImage396.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage396.jpg *License*: unknown *Contributors*: Jcreer
Image:WMSImage395.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage395.jpg *License*: unknown *Contributors*: Jcreer
Image:WMSImage397.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage397.jpg *License*: unknown *Contributors*: Jcreer
Image:WMSImage393.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage393.jpg *License*: unknown *Contributors*: Jcreer
Image:WMSImage392.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage392.jpg *License*: unknown *Contributors*: Jcreer
Image:WMSImage394.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage394.jpg *License*: unknown *Contributors*: Jcreer
Image:WMSImage78.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage78.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage77.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage77.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage76.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage76.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage82.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage82.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage80.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage80.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage79.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage79.gif *License*: unknown *Contributors*: Eshaw
Image:WMSImage81.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage81.gif *License*: unknown *Contributors*: Eshaw
Image:StormDrainMenu.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:StormDrainMenu.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:image115.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image115.jpg *License*: unknown *Contributors*: Eshaw
Image:image118.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image118.jpg *License*: unknown *Contributors*: Eshaw
Image:image120.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image120.jpg *License*: unknown *Contributors*: Eshaw
Image:image152.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image152.jpg *License*: unknown *Contributors*: Cballard
Image:image153.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image153.jpg *License*: unknown *Contributors*: Cballard
Image:smpdbk12.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Smpdbk12.jpg *License*: unknown *Contributors*: Eshaw
Image:WMSImage154.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage154.png *License*: unknown *Contributors*: Eshaw
Image:WMSImage155.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSImage155.png *License*: unknown *Contributors*: Eshaw
Image:image156.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image156.jpg *License*: unknown *Contributors*: Cballard
Image:image157.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image157.jpg *License*: unknown *Contributors*: Cballard
Image:image73.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image73.jpg *License*: unknown *Contributors*: Cballard
Image:image76.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image76.jpg *License*: unknown *Contributors*: Cballard
Image:image89.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image89.jpg *License*: unknown *Contributors*: Cballard
Image:TR55EditParametersDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:TR55EditParametersDialog.png *License*: unknown *Contributors*: Cmsmemoe
Image:SwmmInterface.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SwmmInterface.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:NexradRainfallGages.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NexradRainfallGages.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:ModClarkModel.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:ModClarkModel.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:WmsToXpswmm.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WmsToXpswmm.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HY8CulvertDesign.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8CulvertDesign.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:CurbGutterCalculator.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CurbGutterCalculator.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:GetDataToolbar.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GetDataToolbar.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:VirtualEarthLocator.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:VirtualEarthLocator.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:HydrologicModelingWizard.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizard.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:XmsWiki.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:XmsWiki.jpg *License*: unknown *Contributors*: Cmsmemoe
Image:Calibration.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Calibration.png *License*: unknown *Contributors*: Cmsmemoe
Image:SmoothStreams.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SmoothStreams.png *License*: unknown *Contributors*: Cmsmemoe
Image:MultipleScenarios.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:MultipleScenarios.png *License*: unknown *Contributors*: Cmsmemoe
Image:GroundwaterAndSediment.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GroundwaterAndSediment.png *License*: unknown *Contributors*: Cmsmemoe
Image:Embankments.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Embankments.png *License*: unknown *Contributors*: Cmsmemoe
Image:Sediment.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sediment.png *License*: unknown *Contributors*: Cmsmemoe
Image:WebServices.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WebServices.png *License*: unknown *Contributors*: Cmsmemoe
Image:GoogleEarthKmzAnimExport.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GoogleEarthKmzAnimExport.png *License*: unknown *Contributors*: Cmsmemoe
Image:RainfallToGoogleEarth.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:RainfallToGoogleEarth.png *License*: unknown *Contributors*: Cmsmemoe
Image:IndexMaps.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:IndexMaps.png *License*: unknown *Contributors*: Cmsmemoe
Image:ImportIndexMap.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ImportIndexMap.png *License*: unknown *Contributors*: Cmsmemoe
Image:ConstituentMassPlot.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ConstituentMassPlot.png *License*: unknown *Contributors*: Cmsmemoe
Image:ContaminantConcentration.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ContaminantConcentration.png *License*: unknown *Contributors*: Cmsmemoe
Image:OverlandFlowBcStormSurge.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:OverlandFlowBcStormSurge.png *License*: unknown *Contributors*: Cmsmemoe
Image:JoinSsurgoData.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:JoinSsurgoData.png *License*: unknown *Contributors*: Cmsmemoe
Image:ObservedAndCalibratedData.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ObservedAndCalibratedData.png *License*: unknown *Contributors*: Cmsmemoe
Image:Interface1.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Interface1.PNG *License*: unknown *Contributors*: Mmedina

Image:HY12Interface.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY12Interface.png *License*: unknown *Contributors*: Cmsmemoe

Image:OnlineImages.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:OnlineImages.png *License*: unknown *Contributors*: Cmsmemoe

Image:GetOnlineMaps.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GetOnlineMaps.png *License*: unknown *Contributors*: Cmsmemoe

Image:GetDataToolbar.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GetDataToolbar.png *License*: unknown *Contributors*: Cmsmemoe

Image:WMS85WebServices.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMS85WebServices.png *License*: unknown *Contributors*: Cmsmemoe

Image:GetDataToolbarWebServices.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GetDataToolbarWebServices.png *License*: unknown *Contributors*: Cmsmemoe

Image:VectorImages.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:VectorImages.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY873.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY873.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8EditCrossing.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8EditCrossing.png *License*: unknown *Contributors*: EJones

Image:HY8ModelingWizardCrossingDischargeData.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardCrossingDischargeData.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardWMSScreenshot.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardWMSScreenshot.png *License*: unknown *Contributors*: EJones

Image:HY8-WMSTailwaterAtts.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8-WMSTailwaterAtts.png *License*: unknown *Contributors*: EJones

Image:HY8-WMSRoadwayAtts.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8-WMSRoadwayAtts.png *License*: unknown *Contributors*: EJones

Image:HY8EditDemElevations.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8EditDemElevations.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardDialog.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardTool.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardTool.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardDefineCulvertRoadwayData.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardDefineCulvertRoadwayData.png *License*: unknown *Contributors*: Cmsmemoe

Image:HydrologicModelingWizardRunHydrologicModel.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HydrologicModelingWizardRunHydrologicModel.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardSetupTailwaterChannel.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardSetupTailwaterChannel.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardCulvertAndSiteData.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardCulvertAndSiteData.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardRunCulvertAnalysis.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardRunCulvertAnalysis.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardDefineFloodPolygon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardDefineFloodPolygon.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardStorageCapacityData.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardStorageCapacityData.png *License*: unknown *Contributors*: Cmsmemoe

Image:StorageCapacityInput.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:StorageCapacityInput.png *License*: unknown *Contributors*: Cmsmemoe

Image:DetentionBasinAnalysis.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:DetentionBasinAnalysis.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardDefineUpstreamChannel.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardDefineUpstreamChannel.png *License*: unknown *Contributors*: Cmsmemoe

Image:HY8ModelingWizardDelineateInundatedArea.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HY8ModelingWizardDelineateInundatedArea.png *License*: unknown *Contributors*: Cmsmemoe