WMS User Manual (v10.1)

The Watershed Modeling System

1. Introduction

Introduction to WMS

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

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.

New users may want to become oriented to WMS by following the WMS Quick Tour .

Related Topics

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

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 should be completed beforehand.

Many of the tutorials have files that are needed to run the tutorial. These files and PDF instructions can be found in the "tutorial" folder where WMS was installed or are linked on the WMS Tutorials page. Additionally a zip file containing all the SMS tutorial files and instructions can be downloaded here .

Solutions

The final output files for all the tutorials can be downloaded __ .

General WMS

Category	Tutorial (PDF)	Required Files
Introduction	•Introduction to WMS •Images •Basic Feature Objects •Advanced Feature Objects	•[0] •[1] •[2] •[3]
Editing Elevations	•DEM Basics •Using TINs	•[4] •[5]
Watershed Modeling	•DEM Delineation •Advanced DEM Delineation Techniques •Time of Concentration Calculations and Computing a Composite CN	•[6] •[7] •[8] , [9]

Hydrologic Models

Category	Tutorial (PDF)	Required Files
HEC-1	•HEC-1 Interface	•[10]
HEC-HMS	•HEC-HMS Interface •Using Online Spatial Data to Create an HEC-HMS Model	<None>
Rational Method	•Rational Method Interface	•[11]
NSS (National Streamflow Statistics)	•National Streamflow Statistics Program (NSS) Interface	•[12]
Maricopa County (Phoenix, AZ, USA) Hydrology	•Maricopa County: NSS and HEC-1 •Maricopa County: Master Plan – Creating a Predictive HEC-1 Model	•[13]
MODRAT	•MODRAT Interface – Schematic •MODRAT Interface (GIS-based) •MODRAT Interface (Map-based)	•[14]
Orange County Rational and Unit Hydrograph Methods	•Orange County Rational Method •Orange County Unit Hydrograph •Orange County Small Area Hydrograph •Orange County Hydrology – Using GIS Data	•[15]

	•Orange County Rational Method – GIS •Orange County Unit Hydrograph – GIS	
Spatial Hydrology- HMS ModClark and GSSHA	•HEC-HMS Distributed Parameter Modeling with the MODClark Transform •Developing a GSSHA Model using the Hydrologic Modeling Wizard •Using NEXRAD Rainfall Data in an HEC-HMS (MODClark) Model •Using NEXRAD Rainfall Data in GSSHA •Upgrading an HEC-HMS Model to a GSSHA Model	•[16] •[17]

Distributed Hydrologic Modeling using GSSHA

Category	**Tutorial (PDF)**	**Required Files**
WMS Basics	•Loading DEMs, Contour Options, Images, and Coordinate Systems •Watershed Delineation using DEMs and 2D Grid Generation •Creating Feature Objects and Mapping their Attributes to the 2D Grid	•Personal , GSSHA Images •Personal , Raw Data •Personal , GSSHA Tables , Raw Data , Watershed Delineation Files
GSSHA Modeling	•GSSHA Initial Overland Flow Model Setup •Correcting Overland Flow Problems •Infiltration •Stream Flow •Developing a GSSHA Model Using the Hydrologic Modeling Wizard in WMS •Post-Processing and Visualization of GSSHA Model Results	•Personal , Raw Data , GSSHA Basic Model •Personal , GSSHA Tables , GSSHA Digital Dams •Personal , Raw Data , GSSHA Tables , GSSHA Digital Dams , GSSHA Infiltration •Personal , GSSHA Tables , GSSHA Infiltration •Personal , GSSHA Tables , Raw Data •Personal , Raw Data , GSSHA Visualization
Applications	•Precipitation Methods in GSSHA •Analyzing the Effects of Land Use Change (Part - I) •Analyzing the Effects of Land Use Change (Part - II) •Long Term Simulations in GSSHA •Simulating Sediment Transport •Simulating Constituent Transport	•Personal , GSSHA Tables , Raw Data , GSSHA Precipitation •Personal , GSSHA Tables , GSSHA Scenarios •Personal , GSSHA Tables , GSSHA Scenarios •Personal , Raw Data , GSSHA Long Term

	•Overland Flow Boundary Conditions in GSSHA •Snowmelt Modeling in GSSHA	•Personal , GSSHA Sediment •Personal , GSSHA Contaminants •Personal , GSSHA Boundary Conditions •GSSHA Snowmelt
Calibration	•Manual Calibration of GSSHA models •Stochastic Simulations of GSSHA models •Automated Calibration of GSSHA models	•Personal , GSSHA Calibration •Personal , GSSHA Calibration •Personal , GSSHA Calibration
Groundwater Modeling	•Groundwater Modeling in GSSHA •Advanced Groundwater Modeling in GSSHA •Subsurface Tile and Storm Drains	•Personal , Raw Data , GSSHA Groundwater •Personal , GSSHA Groundwater •Personal , GSSHA Subsurface

Hydraulics and Floodplain Modeling

Category	**Tutorial (PDF)**	**Required Files**
HEC-RAS	•HEC-RAS Analysis •HEC-RAS – Managing Cross Sections	•[18] •[19]
Floodplain Delineation	•Floodplain Delineation •Stochastic Modeling Using HEC-1 and HEC-RAS	•[20] •[21]
SMPDBK (Simplified Dam Break)	•Simplified Dam Break	•[22]
HY-8 Culvert Modeling Wizard	•HY-8 Modeling Wizard	•[23]
Modeling with FHWA's Hydraulic Toolbox	•Modeling with the Hydraulic Toolbox	•[24]

Storm Drain Modeling

Category	**Tutorial (PDF)**	**Required Files**
EPA-SWMM and xpswmm	•SWMM Modeling	•[25]
FHWA Storm Drain	•Storm Drain: Rational Design •Storm Drain: Hydrographic Design	
FHWA HY-12	•HY-12: Rational Design	•[26]

Water Quality Modeling

Category	**Tutorial (PDF)**	**Required Files**
HSPF	•HSPF Interface	•[27]

CE-QUAL-W2	•CE-QUAL-W2 Interface	•[28]

Related Topics

•References

•WMS Tutorial Archive

What's new in WMS version 10.1

The WMS software development team is excited about the release of WMS 10.1! This page lists the exciting new features that have been added to WMS 10.1.

What's new in WMS 10.1

Community Edition

If WMS is unlicensed, it runs as a "Community Edition". This version of WMS is the full version of WMS with the following limitations:

1. Maximum number of sub-basins: 1
2. Maximum number of GIS layers: 2
3. Maximum number of map module coverages and attribute grids: 3
4. Maximum number of terrain data layers (DEMs and TINs): 1
5. Maximum number of storm drain pipes: 50

In addition, 2D scattered data, CAD data, 2D grid data, and 1D open-channel hydraulic data (HECRAS) are not allowed in the community edition. Purchase licenses that enable more layers in each of these modules at http://www.aquaveo.com.

The community edition has an interface to HY12 that is completely free of charge. This means HY12 models with more than 50 pipes are supported as long as the other conditions of the community edition are met.

The community edition allows reading and viewing Rational, NSS, SWMM, EPANET, and other models that exceed these limits, but WMS does not allow saving changes to models that exceed the limits. This edition also allows building storm drain, water distribution, and sanitary sewer models that meet the above criteria as long as the maximum number of pipes is not exceeded. More information about the features enabled in the community edition of WMS and how to purchase WMS packages can be found on the WMS pricing page .

EPANET Model Support

WMS 10.1 adds support for the EPANET model, a widely used water distribution model. Read GIS data files and map their attributes to the EPANET model attributes, or read and edit existing EPANET models.

Improved EPA-SWMM Model Support

WMS 10.1 fully supports both sanitary sewer options and storm drain options associated with EPA-SWMM models. Read GIS data files and map their attributes to the EPA-SWMM model attributes, or read and edit existing EPA-SWMM models.

Improved HY12 Model Support

HY12 is an Federal Highway Administration (FHWA)-sponsored storm drain and hydrology model developed by Aquaveo. The Community edition of WMS 10.1 includes a completely free interface to HY12. There are four new tutorials that describe how to import and use various types of data into WMS to build an HY12 model.

WMS 10.1 includes a simplified interface to HY12 that makes developing an HY12 model much easier than before. The spreadsheet-like interface is similar to the EPA-SWMM and EPANET interfaces and much of the data can be transferred between HY12 and EPA-SWMM so data can be shared between these models.

In addition, a tool exists that allows editing link/node elevations in an HY12 model using a profile editor.

Time Series Data Calculator

Use the time series calculator in the Time Series Editor program to perform simple mathematical operations using time series data.

GSSHA Model Improvements

Several GSSHA model improvements have been made related to better calibration and renumbering streams.

Mine Water Balance Model (MWBM) Wizard

The MWBM wizard steps through the process of editing and modifying a GSSHA model based on changes to the terrain, land use, pumping stations, embankments, and other parameters in a mine model that is tracking sediment output from the mine.

Bug fixes

To view the list of bugs fixed in WMS 10.1, visit the WMS bugfix page .

1.1 Set Up

Setting Up WMS

Installing

On the PC, the installation wizard will guide through the installation process. The installation wizard will give options to install different parts of the WMS program including the executable files, tutorial files, documentation files, etc. If 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.

Running WMS For The First Time

When running WMS the first time (and each time until WMS is registered), the registration wizard is the first dialog seen. If WMS is licensed and there is a hardware lock or need to get the security string to obtain a password, choose the **Enable** button.

Defaults File (ini)

When starting WMS, there are default values set for directories, display options, etc. Modify these settings and save the new defaults in an *.ini file by choosing the **Save Current Settings** command from the *File* menu. Each time starting 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

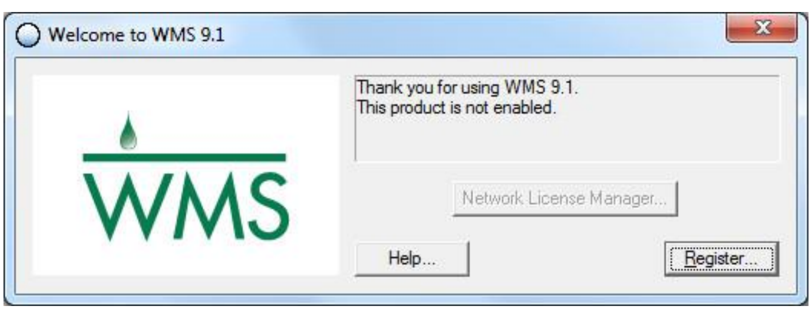
Find out about the available command line arguments that can be used to customize how WMS starts up. See the article Command Line .

Related Topics

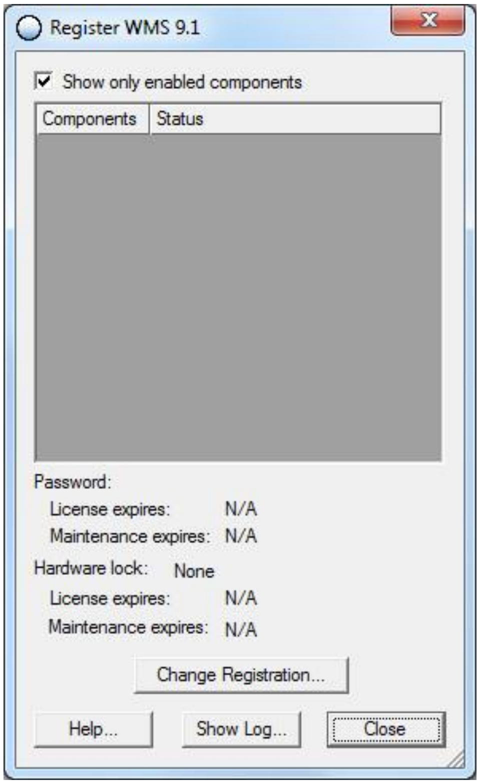
- Registering WMS
- WMS System Requirements
- WMS command line arguments

Registering WMS

After installing WMS, it will need to be registered. Registering online can be done whether or not there is a hardware lock . Either way, obtain a string of characters called a *registration code* . Whether evaluating WMS, have purchased a license without a hardware lock, or have a hardware lock, it's necessary to have a *registration code* to enable WMS. If evaluating WMS for the first time, obtain a free 30-day registration code by registering and providing 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:

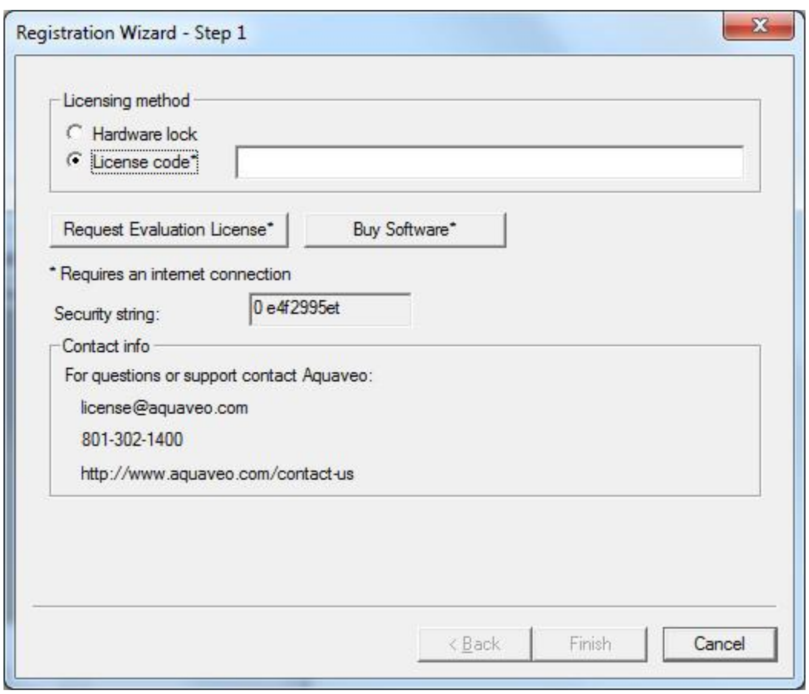


To get started using WMS, the software must be registered. Register WMS by choosing the **Register...** button. The following window will appear:



I

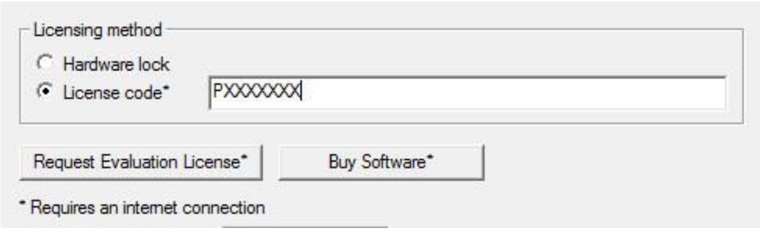
f having just installed a new version of WMS, it is likely that the copy of WMS will not be enabled. It's necessary to enable the copy of WMS. To enable WMS, press the **Change Registration...** button in the *Register* window. The *Registration Wizard* will appear:



In this window, there are two options. It's required to have an active internet connection to register the software. If already having a vendor-supplied code (from a paid license or a temporary demo license), enter the code in the field and select **Next** to register the software. (If there is a hardware lock, make sure the hardware lock drivers are installed and that the hardware lock is connected to the computer before entering the code and selecting **Next** .) If there is no registration code, having not purchased WMS, and having not evaluated WMS before, obtain a registration code for a one-month demo of the software. This will give full access to all the capabilities of WMS for one month. Select the *Request Evaluation License* button, select the **Next** button, and fill in appropriate contact information on the web page that appears in the web browser. A registration code will be sent to supplied e-mail. Return to this *Registration Wizard* to enter this code after receiving it. After the evaluation period expires, purchase a license to continue using WMS.

Paid Licenses

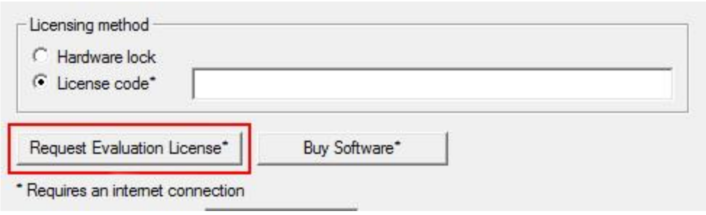
If having purchased a license of WMS, the WMS vendor should have provided a registration code, normally sent to the purchaser's e-mail address. Please enter this code in the *Registration Wizard* .



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

Evaluation Version

If not having a registration code, having not purchased WMS and having not evaluated WMS before, it is possible to 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 **Request Evaluation License** button in the *Registration Wizard* .



A web browser should appear and an internet connection started. Fill out the contact information in the web browser and a valid registration code will be sent via email. After receiving the registration code, enter it into the *Registration Wizard* , and select **Next** . After having evaluated WMS, please contact Aquaveo 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* .

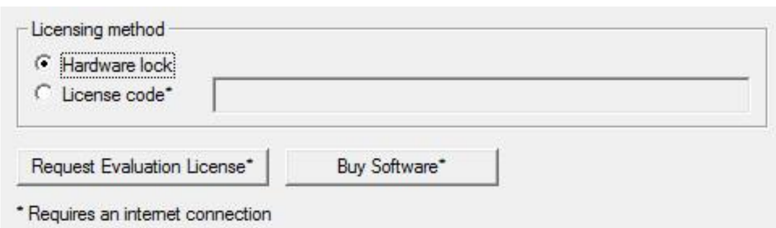
4. The *Register WMS* dialog displays the registered components, licensing method, and license expiration dates.

Hardware Locks

Follow the instructions received with the hardware lock to install the hardware lock and accompanying drivers. If a hardware lock instructions were not received, 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 a web browser.

Additional instructions on using hardware locks can be found in the article Hardware Locks . It's recommended to go 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 the hardware lock with the software will significantly reduce the time and trouble required to get the hardware lock working.

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



The WMS vendor should have provided a registration code, normally sent to the purchaser's e-mail address. Make sure the hardware lock is plugged into the computer, then enter this registration code in the *Enable Wizard* . Make the computer is connected to the internet, and the 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 Windows interface to build the files for HEC-1, TR-20, and the other basic hydrologic models. If wanting the public domain version, contact a WMS vendor and a valid registration code will be sent via email. After receiving the registration code, enter it into the *Registration Wizard* and select **Register** , the same as for paid licenses.

Changing the Registration

If a need arrives to change the what components of WMS are registered, the registration can be accessed through the *Help* menu and selecting the **Register WMS** command.

Related Topics

- 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 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 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. This trade off can be made 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:
    a. 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.

b. Use the system default library on this run (and on future runs if the "Do not ask again box" is checked).

c. 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 to determine your computer's hardware, operating system, and graphics card. To use the dxdiag 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 , HP or Compaq , Toshiba , Sony , 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
- ATI
- Diamond
- Elsa
- Intel
- Matrox

- nVidia
- S3 – Not all S3 card support OpenGL 1.5 which is required for all display options to be enabled.
- SIS – Not all SIS card support OpenGL 1.5 which is required for all display options to be enabled.
- VIA – 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 . 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 .

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 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 .

External Links

•Aquaveo Technical Support

Command Line

Several command line arguments can be used with WMS when it is launched. This 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>

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

Example -dm <module>:

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

•-ini <path>

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

Example -ini <path>:

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

•-tmp <path>

3. 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 -tmp <path>:

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

•-f <file file file...>

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

Example -f <file file file...>:

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

•-av <file>

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

Example -av <file>:

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

•**-about**

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

Example -about:

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

Related Topics

•Setting up WMS

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 the user through the WMS installation. If the pop-up screen does not appear when inserting 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 first installing WMS, it needs to be registered. The user can choose to register with a password, a hardware lock, or with an evaluation password, which allows evaluation of 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 the user through the registration process. If not using a hardware lock, a password will be sent to the registered email to allow the user to register the product.

Hardware Lock:

Follow the instructions received with the hardware lock to install the hardware lock and accompanying drivers. If a hardware lock instructions were not received, 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 a web browser. If wanting 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 received with 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 missing Adode Acrobat Reader, a user can install for free from the Adobe website. If wanting hardcopies of any documentation related to WMS, please contact Aquaveo. Pricing for documentation can be found on Aquaveo's website.

Subscribing to the WMS Mailing List

The WMS mailing list keeps users informed of the latest product news, webinars, training courses, announcements, and special offers. A user 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. A user 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 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 www.aquaveo.com/government-support .

License Agreement

XMS License Agreement

XMS 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. We agree that these programs may send information back to Aquaveo in order to enable a program on the computer in use. Additionally, parts of the software may be covered by one or more of the following agreements.

Boost Software License

Boost Software License - Version 1.0 - August 17th, 2003

Permission is hereby granted, free of charge, to any person or organization obtaining a copy of the software and accompanying documentation covered by this license (the "Software") to use, reproduce, display, distribute, execute, and transmit the Software, and to prepare derivative works of the Software, and to permit third-parties to whom the Software is furnished to do so, all subject to the following:

The copyright notices in the Software and this entire statement, including the above license grant, this restriction and the following disclaimer, must be included in all copies of the Software, in whole or in part, and all derivative works of the Software, unless such copies or derivative works are solely in the form of machine-executable object code generated by a source language processor.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, TITLE AND NON-INFRINGEMENT. IN NO EVENT SHALL THE COPYRIGHT HOLDERS OR ANYONE DISTRIBUTING THE SOFTWARE BE LIABLE FOR ANY DAMAGES OR OTHER LIABILITY, WHETHER IN CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

FTGL License Agreement

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:

Copyright (c) <year> <copyright holders> Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Bugtrap

http://www.codeproject.com/Articles/14618/Catch-All-Bugs-with-BugTrap

The above URL states that Bugtrap is licensed as follows:

Microsoft Public License (MS-PL)

[OSI Approved License]

This license governs use of the accompanying software. If you use the software, you accept this license. If you do not accept the license, do not use the software.

1. Definitions The terms "reproduce," "reproduction," "derivative works," and "distribution" have the same meaning here as under U.S. copyright law. A "contribution" is the original software, or any additions or changes to the software. A "contributor" is any person that distributes its contribution under this license. "Licensed patents" are a contributor's patent claims that read directly on its contribution.

2. Grant of Rights

(A) Copyright Grant- Subject to the terms of this license, including the license conditions and limitations in section 3, each contributor grants you a non-exclusive, worldwide, royalty-free copyright license to reproduce its contribution, prepare derivative works of its contribution, and distribute its contribution or any derivative works that you create.

(B) Patent Grant- Subject to the terms of this license, including the license conditions and limitations in section 3, each contributor grants you a non-exclusive, worldwide, royalty-free license under its licensed patents to make, have made, use, sell, offer for sale, import, and/or otherwise dispose of its contribution in the software or derivative works of the contribution in the software.

3. Conditions and Limitations

(A) No Trademark License- This license does not grant you rights to use any contributors' name, logo, or trademarks.

(B) If you bring a patent claim against any contributor over patents that you claim are infringed by the software, your patent license from such contributor to the software ends automatically.

(C) If you distribute any portion of the software, you must retain all copyright, patent, trademark, and attribution notices that are present in the software.

(D) If you distribute any portion of the software in source code form, you may do so only under this license by including a complete copy of this license with your distribution. If you distribute any portion of the software in compiled or object code form, you may only do so under a license that complies with this license.

(E) The software is licensed "as-is." You bear the risk of using it. The contributors give no express warranties, guarantees or conditions. You may have additional consumer rights under your local laws which this license cannot change. To the extent permitted under your local laws, the contributors exclude the implied warranties of merchantability, fitness for a particular purpose and non-infringement.

GNU Lesser General Public License

GNU LESSER GENERAL PUBLIC LICENSE

Version 2.1, February 1999

Copyright (C) 1991, 1999 Free Software Foundation, Inc. 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301 USA Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

[This is the first released version of the Lesser GPL. It also counts as the successor of the GNU Library Public License, version 2, hence the version number 2.1.]

Preamble

The licenses for most software are designed to take away your freedom to share and change it. By contrast, the GNU General Public Licenses are intended to guarantee your freedom to share and change free software--to make sure the software is free for all its users.

This license, the Lesser General Public License, applies to some specially designated software packages--typically libraries--of the Free Software Foundation and other authors who decide to use it. You can use it too, but we suggest you first think carefully about whether this license or the ordinary General Public License is the better strategy to use in any particular case, based on the explanations below.

When we speak of free software, we are referring to freedom of use, not price. Our General Public Licenses are designed to make sure that you have the freedom to distribute copies of free software (and charge for this service if you wish); that you receive source code or can get it if you want it; that you can change the software and use pieces of it in new free programs; and that you are informed that you can do these things.

To protect your rights, we need to make restrictions that forbid distributors to deny you these rights or to ask you to surrender these rights. These restrictions translate to certain responsibilities for you if you distribute copies of the library or if you modify it.

For example, if you distribute copies of the library, whether gratis or for a fee, you must give the recipients all the rights that we gave you. You must make sure that they, too, receive or can get the source code. If you link other code with the library, you must provide complete object files to the recipients, so that they can relink them with the library after making changes to the library and recompiling it. And you must show them these terms so they know their rights.

We protect your rights with a two-step method: (1) we copyright the library, and (2) we offer you this license, which gives you legal permission to copy, distribute and/or modify the library.

To protect each distributor, we want to make it very clear that there is no warranty for the free library. Also, if the library is modified by someone else and passed on, the recipients should know that what they have is not the original version, so that the original author's reputation will not be affected by problems that might be introduced by others.

Finally, software patents pose a constant threat to the existence of any free program. We wish to make sure that a company cannot effectively restrict the users of a free program by obtaining a restrictive license from a patent holder. Therefore, we insist that any patent license obtained for a version of the library must be consistent with the full freedom of use specified in this license.

Most GNU software, including some libraries, is covered by the ordinary GNU General Public License. This license, the GNU Lesser General Public License, applies to certain designated libraries, and is quite different from the ordinary General Public License. We use this license for certain libraries in order to permit linking those libraries into non-free programs.

When a program is linked with a library, whether statically or using a shared library, the combination of the two is legally speaking a combined work, a derivative of the original library. The ordinary General Public License therefore permits such linking only if the entire combination fits its criteria of freedom. The Lesser General Public License permits more lax criteria for linking other code with the library.

We call this license the "Lesser" General Public License because it does Less to protect the user's freedom than the ordinary General Public License. It also provides other free software developers Less of an advantage over competing non-free programs. These disadvantages are the reason we use the ordinary General Public License for many libraries. However, the Lesser license provides advantages in certain special circumstances.

For example, on rare occasions, there may be a special need to encourage the widest possible use of a certain library, so that it becomes a de-facto standard. To achieve this, non-free programs must be allowed to use the library. A more frequent case is that a free library does the same job as widely used non-free libraries. In this case, there is little to gain by limiting the free library to free software only, so we use the Lesser General Public License.

In other cases, permission to use a particular library in non-free programs enables a greater number of people to use a large body of free software. For example, permission to use the GNU C Library in non-free programs enables many more people to use the whole GNU operating system, as well as its variant, the GNU/Linux operating system.

Although the Lesser General Public License is Less protective of the users' freedom, it does ensure that the user of a program that is linked with the Library has the freedom and the wherewithal to run that program using a modified version of the Library.

The precise terms and conditions for copying, distribution and modification follow. Pay close attention to the difference between a "work based on the library" and a "work that uses the library". The former contains code derived from the library, whereas the latter must be combined with the library in order to run.

Terms and Conditions for Copying, Distribution and Modification

GNU LESSER GENERAL PUBLIC LICENSE

TERMS AND CONDITIONS FOR COPYING, DISTRIBUTION AND MODIFICATION

0. This License Agreement applies to any software library or other program which contains a notice placed by the copyright holder or other authorized party saying it may be distributed under the terms of this Lesser General Public License (also called "this License"). Each licensee is addressed as "you".

A "library" means a collection of software functions and/or data prepared so as to be conveniently linked with application programs (which use some of those functions and data) to form executables.

The "Library", below, refers to any such software library or work which has been distributed under these terms. A "work based on the Library" means either the Library or any derivative work under copyright law: that is to say, a work containing the Library or a portion of it, either verbatim or with modifications and/or translated straightforwardly into another language. (Hereinafter, translation is included without limitation in the term "modification".)

"Source code" for a work means the preferred form of the work for making modifications to it. For a library, complete source code means all the source code for all modules it contains, plus any associated interface definition files, plus the scripts used to control compilation and installation of the library.

Activities other than copying, distribution and modification are not covered by this License; they are outside its scope. The act of running a program using the Library is not restricted, and output from such a program is covered only if its contents constitute a work based on the Library (independent of the use of the Library in a tool for writing it). Whether that is true depends on what the Library does and what the program that uses the Library does.

1. You may copy and distribute verbatim copies of the Library's complete source code as you receive it, in any medium, provided that you conspicuously and appropriately publish on each copy an appropriate copyright notice and disclaimer of warranty; keep intact all the notices that refer to this License and to the absence of any warranty; and distribute a copy of this License along with the Library.

You may charge a fee for the physical act of transferring a copy, and you may at your option offer warranty protection in exchange for a fee.

2. You may modify your copy or copies of the Library or any portion of it, thus forming a work based on the Library, and copy and distribute such modifications or work under the terms of Section 1 above, provided that you also meet all of these conditions:

a) The modified work must itself be a software library.

b) You must cause the files modified to carry prominent notices stating that you changed the files and the date of any change.

c) You must cause the whole of the work to be licensed at no charge to all third parties under the terms of this License.

d) If a facility in the modified Library refers to a function or a table of data to be supplied by an application program that uses the facility, other than as an argument passed when the facility is invoked, then you must make a good faith effort to ensure that, in the event an application does not supply such function or table, the facility still operates, and performs whatever part of its purpose remains meaningful.

(For example, a function in a library to compute square roots has a purpose that is entirely well-defined independent of the application. Therefore, Subsection 2d requires that any application-supplied function or table used by this function must be optional: if the application does not supply it, the square root function must still compute square roots.)

These requirements apply to the modified work as a whole. If identifiable sections of that work are not derived from the Library, and can be reasonably considered independent and separate works in themselves, then this License, and its terms, do not apply to those sections when you distribute them as separate works. But when you distribute the same sections as part of a whole which is a work based on the Library, the distribution of the whole must be on the terms of this License, whose permissions for other licensees extend to the entire whole, and thus to each and every part regardless of who wrote it.

Thus, it is not the intent of this section to claim rights or contest your rights to work written entirely by you; rather, the intent is to exercise the right to control the distribution of derivative or collective works based on the Library.

In addition, mere aggregation of another work not based on the Library with the Library (or with a work based on the Library) on a volume of a storage or distribution medium does not bring the other work under the scope of this License.

3. You may opt to apply the terms of the ordinary GNU General Public License instead of this License to a given copy of the Library. To do this, you must alter all the notices that refer to this License, so that they refer to the ordinary GNU General Public License, version 2, instead of to this License. (If a newer version than version 2 of the ordinary GNU General Public License has appeared, then you can specify that version instead if you wish.) Do not make any other change in these notices.

Once this change is made in a given copy, it is irreversible for that copy, so the ordinary GNU General Public License applies to all subsequent copies and derivative works made from that copy.

This option is useful when you wish to copy part of the code of the Library into a program that is not a library.

4. You may copy and distribute the Library (or a portion or derivative of it, under Section 2) in object code or executable form under the terms of Sections 1 and 2 above provided that you accompany it with the complete corresponding machine-readable source code, which must be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange.

If distribution of object code is made by offering access to copy from a designated place, then offering equivalent access to copy the source code from the same place satisfies the requirement to distribute the source code, even though third parties are not compelled to copy the source along with the object code.

5. A program that contains no derivative of any portion of the Library, but is designed to work with the Library by being compiled or linked with it, is called a "work that uses the Library". Such a work, in isolation, is not a derivative work of the Library, and therefore falls outside the scope of this License.

However, linking a "work that uses the Library" with the Library creates an executable that is a derivative of the Library (because it contains portions of the Library), rather than a "work that uses the library". The executable is therefore covered by this License. Section 6 states terms for distribution of such executables.

When a "work that uses the Library" uses material from a header file that is part of the Library, the object code for the work may be a derivative work of the Library even though the source code is not. Whether this is true is especially significant if the work can be linked without the Library, or if the work is itself a library. The threshold for this to be true is not precisely defined by law.

If such an object file uses only numerical parameters, data structure layouts and accessors, and small macros and small inline functions (ten lines or less in length), then the use of the object file is unrestricted, regardless of whether it is legally a derivative work. (Executables containing this object code plus portions of the Library will still fall under Section 6.)

Otherwise, if the work is a derivative of the Library, you may distribute the object code for the work under the terms of Section 6. Any executables containing that work also fall under Section 6, whether or not they are linked directly with the Library itself.

6. As an exception to the Sections above, you may also combine or link a "work that uses the Library" with the Library to produce a work containing portions of the Library, and distribute that work under terms of your choice, provided that the terms permit modification of the work for the customer's own use and reverse engineering for debugging such modifications.

You must give prominent notice with each copy of the work that the Library is used in it and that the Library and its use are covered by this License. You must supply a copy of this License. If the work during execution displays copyright notices, you must include the copyright notice for the Library among them, as well as a reference directing the user to the copy of this License. Also, you must do one of these things:

a) Accompany the work with the complete corresponding machine-readable source code for the Library including whatever changes were used in the work (which must be distributed under Sections 1 and 2 above); and, if the work is an executable linked with the Library, with the complete machine-readable "work that uses the Library", as object code and/or source code, so that the user can modify the Library and then relink to produce a modified executable containing the modified Library. (It is understood that the user who changes the contents of definitions files in the Library will not necessarily be able to recompile the application to use the modified definitions.)

b) Use a suitable shared library mechanism for linking with the Library. A suitable mechanism is one that (1) uses at run time a copy of the library already present on the user's computer system, rather than copying library functions into the executable, and (2) will operate properly with a modified version of the library, if the user installs one, as long as the modified version is interface-compatible with the version that the work was made with.

c) Accompany the work with a written offer, valid for at least three years, to give the same user the materials specified in Subsection 6a, above, for a charge no more than the cost of performing this distribution.

d) If distribution of the work is made by offering access to copy from a designated place, offer equivalent access to copy the above specified materials from the same place.

e) Verify that the user has already received a copy of these materials or that you have already sent this user a copy.

For an executable, the required form of the "work that uses the Library" must include any data and utility programs needed for reproducing the executable from it. However, as a special exception, the materials to be distributed need not include anything that is normally distributed (in either source or binary form) with the major components (compiler, kernel, and so on) of the operating system on which the executable runs, unless that component itself accompanies the executable.

It may happen that this requirement contradicts the license restrictions of other proprietary libraries that do not normally accompany the operating system. Such a contradiction means you cannot use both them and the Library together in an executable that you distribute.

7. You may place library facilities that are a work based on the Library side-by-side in a single library together with other library facilities not covered by this License, and distribute such a combined library, provided that the separate distribution of the work based on the Library and of the other library facilities is otherwise permitted, and provided that you do these two things:

a) Accompany the combined library with a copy of the same work based on the Library, uncombined with any other library facilities. This must be distributed under the terms of the Sections above.

b) Give prominent notice with the combined library of the fact that part of it is a work based on the Library, and explaining where to find the accompanying uncombined form of the same work.

8. You may not copy, modify, sublicense, link with, or distribute the Library except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense, link with, or distribute the Library is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.

9. You are not required to accept this License, since you have not signed it. However, nothing else grants you permission to modify or distribute the Library or its derivative works. These actions are prohibited by law if you do not accept this License. Therefore, by modifying or distributing the Library (or any work based on the Library), you indicate your acceptance of this License to do so, and all its terms and conditions for copying, distributing or modifying the Library or works based on it.

10. Each time you redistribute the Library (or any work based on the Library), the recipient automatically receives a license from the original licensor to copy, distribute, link with or modify the Library subject to these terms and conditions. You may not impose any further restrictions on the recipients' exercise of the rights granted herein. You are not responsible for enforcing compliance by third parties with this License.

11. If, as a consequence of a court judgment or allegation of patent infringement or for any other reason (not limited to patent issues), conditions are imposed on you (whether by court order, agreement or otherwise) that contradict the conditions of this License, they do not excuse you from the conditions of this License. If you cannot distribute so as to satisfy simultaneously your obligations under this License and any other pertinent obligations, then as a consequence you may not distribute the Library at all. For example, if a patent license would not permit royalty-free redistribution of the Library by all those who receive copies directly or indirectly through you, then the only way you could satisfy both it and this License would be to refrain entirely from distribution of the Library.

If any portion of this section is held invalid or unenforceable under any particular circumstance, the balance of the section is intended to apply, and the section as a whole is intended to apply in other circumstances.

It is not the purpose of this section to induce you to infringe any patents or other property right claims or to contest validity of any such claims; this section has the sole purpose of protecting the integrity of the free software distribution system which is implemented by public license practices. Many people have made generous contributions to the wide range of software distributed through that system in reliance on consistent application of that system; it is up to the author/donor to decide if he or she is willing to distribute software through any other system and a licensee cannot impose that choice.

This section is intended to make thoroughly clear what is believed to be a consequence of the rest of this License.

12. If the distribution and/or use of the Library is restricted in certain countries either by patents or by copyrighted interfaces, the original copyright holder who places the Library under this License may add an explicit geographical distribution limitation excluding those countries, so that distribution is permitted only in or among countries not thus excluded. In such case, this License incorporates the limitation as if written in the body of this License.

13. The Free Software Foundation may publish revised and/or new versions of the Lesser General Public License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns.

Each version is given a distinguishing version number. If the Library specifies a version number of this License which applies to it and "any later version", you have the option of following the terms and conditions either of that version or of any later version published by the Free Software Foundation. If the Library does not specify a license version number, you may choose any version ever published by the Free Software Foundation.

14. If you wish to incorporate parts of the Library into other free programs whose distribution conditions are incompatible with these, write to the author to ask for permission. For software which is copyrighted by the Free Software Foundation, write to the Free Software Foundation; we sometimes make exceptions for this. Our decision will be guided by the two goals of preserving the free status of all derivatives of our free software and of promoting the sharing and reuse of software generally.

No Warranty

NO WARRANTY

15. BECAUSE THE LIBRARY IS LICENSED FREE OF CHARGE, THERE IS NO WARRANTY FOR THE LIBRARY, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE LIBRARY "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE LIBRARY IS WITH YOU. SHOULD THE LIBRARY PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

16. IN NO EVENT UNLESS REQUIRED BY APPLICABLE LAW OR AGREED TO IN WRITING WILL ANY COPYRIGHT HOLDER, OR ANY OTHER PARTY WHO MAY MODIFY AND/OR REDISTRIBUTE THE LIBRARY AS PERMITTED ABOVE, BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE LIBRARY (INCLUDING BUT NOT LIMITED TO LOSS OF DATA OR DATA BEING RENDERED INACCURATE OR LOSSES SUSTAINED BY YOU OR THIRD PARTIES OR A FAILURE OF THE LIBRARY TO OPERATE WITH ANY OTHER SOFTWARE), EVEN IF SUCH HOLDER OR OTHER PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

END OF TERMS AND CONDITIONS

How to Apply These Terms to Your New Libraries

If you develop a new library, and you want it to be of the greatest possible use to the public, we recommend making it free software that everyone can redistribute and change. You can do so by permitting redistribution under these terms (or, alternatively, under the terms of the ordinary General Public License).

To apply these terms, attach the following notices to the library. It is safest to attach them to the start of each source file to most effectively convey the exclusion of warranty; and each file should have at least the "copyright" line and a pointer to where the full notice is found.

```
<one line to give the library's name and a brief idea of what it
does.>    Copyright (C) <year>  <name of author>    This library is
free software; you can redistribute it and/or   modify it under the
terms of the GNU Lesser General Public   License as published by
the Free Software Foundation; either   version 2.1 of the License,
or (at your option) any later version.   This library is
distributed in the hope that it will be useful,   but WITHOUT ANY
WARRANTY; without even the implied warranty of   MERCHANTABILITY or
FITNESS FOR A PARTICULAR PURPOSE.  See the GNU   Lesser General
Public License for more details.   You should have received a copy
of the GNU Lesser General Public   License along with this library;
if not, write to the Free Software   Foundation, Inc., 51 Franklin
Street, Fifth Floor, Boston, MA  02110-1301  USA
```

Also add information on how to contact you by electronic and paper mail.

You should also get your employer (if you work as a programmer) or your school, if any, to sign a "copyright disclaimer" for the library, if necessary. Here is a sample; alter the names:

Yoyodyne, Inc., hereby disclaims all copyright interest in the library `Frob' (a library for tweaking knobs) written by James Random Hacker.

<signature of Ty Coon>, 1 April 1990

Ty Coon, President of Vice

That's all there is to it!

Qt

Other Licenses Used in Qt

Qt contains some code that is not provided under the GNU General Public License (GPL), GNU Lesser General Public License (LGPL) or the Qt Commercial License Agreement, but rather under specific licenses from the original authors. Some pieces of code were developed by Nokia and others originated from third parties. This page lists the licenses used, names the authors, and links to the places where it is used.

Nokia gratefully acknowledges these and other contributions to Qt. We recommend that programs that use Qt also acknowledge these contributions, and quote these license statements in an appendix to the documentation.

See also: Licenses for Fonts Used in Qt for Embedded Linux

Copyright (c) 2001 Robert Penner All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
- Neither the name of the author nor the names of contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES(INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Easing Equations by Robert Penner

OpenGL is a trademark of Silicon Graphics, Inc. in the United States and other countries.

QGLFormat

QGLFormat

QPlatformWindowFormat

QtScript is licensed under the GNU Library General Public License. Individual contributor names and copyright dates can be found inline in the code.

This library is free software; you can redistribute it and/or modify it under the terms of the GNU Library General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version.

This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Library General Public License for more details.

You should have received a copy of the GNU Library General Public License along with this library; see the file COPYING.LIB. If not, write to the Free Software Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301, USA.

QtScript Module

QtScript Module

WebKit is licensed under the GNU Library General Public License. Individual contributor names and copyright dates can be found inline in the code.

This library is free software; you can redistribute it and/or modify it under the terms of the GNU Library General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version.

This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Library General Public License for more details.

You should have received a copy of the GNU Library General Public License along with this library; see the file COPYING.LIB. If not, write to the Free Software Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301, USA.

WebKit in Qt

Copyright (C) 1999 Serika Kurusugawa. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR AND CONTRIBUTORS "AS IS". ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Shift-JIS Text Codec

ISO 2022-JP (JIS) Text Codec

EUC-JP Text Codec

Copyright (C) 1999-2000 Mizi Research Inc. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

EUC-KR Text Codec

GBK Text Codec

Big5-HKSCS Text Codec

Big5 Text Codec

This file is written to contribute to Nokia Corporation and/or its subsidiary(-ies) under their own license. You may use this file under your Qt license. Following description is copied from their original file headers. Contact immodule-qt@freedesktop.org if any conditions of this licensing are not clear to you.

QInputContextPlugin, QInputContext, QInputContextFactory

- QInputContextPlugin
- QInputContext
- QInputContextFactory

QtHelp Module

QImage

Contributions to the Following QtGui Files: qapplication_cocoa_p.h, qapplication_mac.mm, qdesktopwidget_mac.mm qeventdispatcher_mac.mm qeventdispatcher_mac_p.h qmacincludes_mac.h qt_cocoa_helpers.mm qt_cocoa_helpers_p.h qwidget_mac.mm qsystemtrayicon_mac.mm

QAxServer Module, QAxContainer Module

QAxServer Module

QAxContainer Module

The qtmain Library

Implementation of the Recursive Shadow Casting Algorithm in Qt Designer

Contributions to the Following qt3to4 Files

Contributions to the Following qt3to4 Files: treewalker.h, treedump.cpp, treedump.h, treewalker.cpp

Torrent Example

QRegion

QDate::weekNumber()

QLocale

TSCII Text Codec

QTestLib Manual

Permission to use, copy, modify, distribute, and sell this software for any purpose is hereby granted without fee, provided that the above copyright notice appear in all copies and that both that copyright notice and this permission notice appear in supporting documentation, and that the name of Daniel Dardailler not be used in advertising or publicity pertaining to distribution of the software without specific, written prior permission. Daniel Dardailler makes no representations about the suitability of this software for any purpose. It is provided "as is" without express or implied warranty.

Drag and Drop

Copyright 2002 USC/Information Sciences Institute

Permission to use, copy, modify, distribute, and sell this software and its documentation for any purpose is hereby granted without fee, provided that the above copyright notice appear in all copies and that both that copyright notice and this permission notice appear in supporting documentation, and that the name of Information Sciences Institute not be used in advertising or publicity pertaining to distribution of the software without specific, written prior permission. Information Sciences Institute makes no representations about the suitability of this software for any purpose. It is provided "as is" without express or implied warranty.

INFORMATION SCIENCES INSTITUTE DISCLAIMS ALL WARRANTIES WITH REGARD TO THIS SOFTWARE, INCLUDING ALL IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS, IN NO EVENT SHALL INFORMATION SCIENCES INSTITUTE BE LIABLE FOR ANY SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGES OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THIS SOFTWARE.

QtSvg Module

This file is part of the KDE project

Copyright (C) 2004-2009 Matthias Kretz <kretz@kde.org>
Copyright (C) 2008 Ian Monroe <ian@monroe.nu>
Copyright (C) 2007-2008 Trolltech ASA
Copyright (C) 2012 Nokia Corporation and/or its subsidiary(-ies).
Contact: Nokia Corporation (qt-info@nokia.com)

This library is free software; you can redistribute it and/or modify it under the terms of the GNU Library General Public License version 2 as published by the Free Software Foundation.

This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Library General Public License for more details.

You should have received a copy of the GNU Library General Public License along with this library; see the file COPYING.LIB. If not, write to the Free Software Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301, USA.

Phonon Module

W3C© SOFTWARE NOTICE AND LICENSE

This license came from: http://www.w3.org/Consortium/Legal/2002/copyright-software-20021231

This work (and included software, documentation such as READMEs, or other related items) is being provided by the copyright holders under the following license. By obtaining, using and/or copying this work, you (the licensee) agree that you have read, understood, and will comply with the following terms and conditions.

Permission to copy, modify, and distribute this software and its documentation, with or without modification, for any purpose and without fee or royalty is hereby granted, provided that you include the following on ALL copies of the software and documentation or portions thereof, including modifications:

1. The full text of this NOTICE in a location viewable to users of the redistributed or derivative work. 2. Any pre-existing intellectual property disclaimers, notices, or terms and conditions. If none exist, the W3C Software Short Notice should be included (hypertext is preferred, text is permitted) within the body of any redistributed or derivative code. 3. Notice of any changes or modifications to the files, including the date changes were made. (We recommend you provide URIs to the location from which the code is derived.)

THIS SOFTWARE AND DOCUMENTATION IS PROVIDED "AS IS," AND COPYRIGHT HOLDERS MAKE NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO, WARRANTIES OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE OR THAT THE USE OF THE SOFTWARE OR DOCUMENTATION WILL NOT INFRINGE ANY THIRD PARTY PATENTS, COPYRIGHTS, TRADEMARKS OR OTHER RIGHTS.

COPYRIGHT HOLDERS WILL NOT BE LIABLE FOR ANY DIRECT, INDIRECT, SPECIAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF ANY USE OF THE SOFTWARE OR DOCUMENTATION.

The name and trademarks of copyright holders may NOT be used in advertising or publicity pertaining to the software without specific, written prior permission. Title to copyright in this software and any associated documentation will at all times remain with copyright holders.

QtXmlPatterns Module

Copyright (C) 2000-2004, International Business Machines Corporation and others. All Rights Reserved. Copyright (C) 2007 Apple Inc. All rights reserved.

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, provided that the above copyright notice(s) and this permission notice appear in all copies of the Software and that both the above copyright notice(s) and this permission notice appear in supporting documentation.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT OF THIRD PARTY RIGHTS. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR HOLDERS INCLUDED IN THIS NOTICE BE LIABLE FOR ANY CLAIM, OR ANY SPECIAL INDIRECT OR CONSEQUENTIAL DAMAGES, OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THIS SOFTWARE.

Except as contained in this notice, the name of a copyright holder shall not be used in advertising or otherwise to promote the sale, use or other dealings in this Software without prior written authorization of the copyright holder.

Parts of WebKit used by the QtWebKit module

Copyright (c) 1998 by Bjorn Reese <breese@imada.ou.dk>

Permission to use, copy, modify, and distribute this software for any purpose with or without fee is hereby granted, provided that the above copyright notice and this permission notice appear in all copies.

THIS SOFTWARE IS PROVIDED ``AS IS *AND WITHOUT ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, WITHOUT LIMITATION, THE IMPLIED WARRANTIES OF MERCHANTIBILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE AUTHORS AND CONTRIBUTORS ACCEPT NO RESPONSIBILITY IN ANY CONCEIVABLE MANNER.*

Parts of the QCrashHandler class

Parts of the FreeType projects have been modified and put into Qt for use in the painting subsystem. These files are ftraster.h, ftraster.c, ftgrays.h and ftgrays.c. The following modifications has been made to these files:

```
Renamed FT_ and ft_ symbols to QT_FT_ and qt_ft_ to avoid name
conflicts in qrasterdefs_p.h. Removed parts of code not relevant
when compiled with _STANDALONE_ defined. Changed behavior in
ftraster.c to follow X polygon filling rules. Implemented support
in ftraster.c for winding / odd even polygon fill rules. Replaced
bitmap generation with span generation in ftraster.c. Renamed
ftraster.h as qblackraster_p.h. Renamed ftraster.c as qblackraster.c.
Renamed ftgrays.h as qgrayraster_p.h. Renamed ftgrays.c as
qgrayraster.c.
```

See src/3rdparty/freetype/docs/FTL.txt and src/3rdparty/freetype/docs/GPL.txt for license details.

Copyright (c) 1985, 1986, 1987 X Consortium

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE X CONSORTIUM BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Except as contained in this notice, the name of the X Consortium shall not be used in advertising or otherwise to promote the sale, use or other dealings in this Software without prior written authorization from the X Consortium.

Parts of the Q3PolygonScanner class used in Qt for Embedded Linux

Copyright 1987 by Digital Equipment Corporation, Maynard, Massachusetts.

All Rights Reserved

Permission to use, copy, modify, and distribute this software and its documentation for any purpose and without fee is hereby granted, provided that the above copyright notice appear in all copies and that both that copyright notice and this permission notice appear in supporting documentation, and that the name of Digital not be used in advertising or publicity pertaining to distribution of the software without specific, written prior permission.

Parts of the Q3PolygonScanner class used in Qt for Embedded Linux

Parts of the internal QKeyMapper class on X11 platforms

pnmscale.c - read a portable anymap and scale it

Parts of the internal QImageSmoothScaler

Parts of the internal QImageSmoothScaler::scale() function use code based on pnmscale.c by Jef Poskanzer.

jQuery JavaScript Library v1.3.2 http://jquery.com/

•Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

•Neither the name of Research In Motion Limited nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY Research In Motion Limited *AS IS* AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL Research In Motion Limited BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

```
src/corelib/io/qurl.cpp
```

Copyright (c) 2007-2008, Apple, Inc.
Copyright (C) 2008 Cameron Zwarich <cwzwarich@uwaterloo.ca>
Copyright (C) 2009 Google Inc. All rights reserved.
Copyright (C) 2008, 2009 Paul Pedriana <ppedriana@ea.com>. All rights reserved.
Copyright (C) 2007 Justin Haygood (jhaygood@reaktix.com)
Copyright (C) 2009 Jian Li <jianli@chromium.org>
Copyright (C) 2007 Staikos Computing Services Inc.
Copyright (C) 2008 Nokia Corporation and/or its subsidiary(-ies)
Copyright (C) 2008 Nuanti Ltd.
Copyright (C) 2007 David Smith (catfish.man@gmail.com)
Copyright (C) 2008 Tony Chang <idealisms@gmail.com>
Copyright (C) 2007 Graham Dennis (graham.dennis@gmail.com)

All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

•Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

•Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

•Neither the name of Apple, Inc. nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

Parts of WebKit used by the QtWebKit module

Parts of the codecs implemented by Qt

```
src/corelib/tools/qlocale.cpp
```


Parts of WebKit used by the QtWebKit module

Parts of the FreeType library

Included in the build system of the FreeType library

See CatharonLicense.txt for more information

Copyright (C) 2006 Apple Computer, Inc. All rights reserved.
Copyright (C) 2007 Eric Seidel <eric@webkit.org>
Copyright (C) 2008 Kelvin W Sherlock (ksherlock@gmail.com)
Copyright (C) 2008 Alp Toker <alp@atoker.com>
Copyright (C) 2009 University of Szeged
Copyright (C) 2007 Alexey Proskuryakov (ap@nypop.com)
Copyright (C) 2009 Daniel Bates (dbates@intudata.com)
Copyright (C) 2008 Nikolas Zimmermann <zimmermann@kde.org>
Copyright (C) 2006 Michael Emmel mike.emmel@gmail.com
Copyright (C) 2007 Holger Hans Peter Freyther
Copyright (C) 2008 Collabora Ltd. All rights reserved.
Copyright (C) 2006 Dirk Mueller <mueller@kde.org>
Copyright (C) 2006 Zack Rusin <zack@kde.org>
Copyright (C) 2006 George Staikos <staikos@kde.org>
Copyright (C) 2006 Simon Hausmann <hausmann@kde.org>
Copyright (C) 2006 Rob Buis <buis@kde.org>
Copyright (C) 2008 Julien Chaffraix <jchaffraix@webkit.org>
Copyright (C) 2007 Henry Mason (hmason@mac.com)
Copyright (C) 1999 Lars Knoll (knoll@kde.org)
Copyright (C) 1999 Antti Koivisto (koivisto@kde.org)
Copyright (c) 2009 The Android Open Source Project
Copyright (C) 2008 Dirk Schulze <krit@webkit.org>
Copyright (C) 2008 Nokia Corporation and/or its subsidiary(-ies)
Copyright (C) 2008 Matt Lilek <webkit@mattlilek.com>
Copyright (C) 2009 280 North Inc. All Rights Reserved.
Copyright (C) 2009 Joseph Pecoraro
Copyright (C) 2008 Anthony Ricaud (rik24d@gmail.com)
Copyright (C) 2006 Samuel Weinig <sam.weinig@gmail.com>
Copyright (C) 2008 Christian Dywan <christian@imendio.com>
Copyright (C) 2006 Michael Emmel mike.emmel@gmail.com
Copyright (C) 2009 Holger Hans Peter Freyther
Copyright (C) 2008 Google Inc. All rights reserved.
Copyright (C) 2006 Friedemann Kleint <fkleint@trolltech.com>
Copyright (C) 2007 Nicholas Shanks <webkit@nickshanks.com>
Copyright (C) 2008 Collin Jackson <collinj@webkit.org>
Copyright (C) 2007 Staikos Computing Services Inc. <info@staikos.net>
Copyright (C) 2008 Kevin Ollivier <kevino@theolliviers.com> All Rights Reserved.
Copyright (C) 2005 Frerich Raabe <raabe@kde.org>
Copyright (C) 2005 Maksim Orlovich <maksim@kde.org>
Copyright (C) 2005, 2006 Kimmo Kinnunen <kimmo.t.kinnunen@nokia.com>.
Copyright (C) 2007-2009 Torch Mobile, Inc.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY APPLE COMPUTER, INC. ``AS IS *AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL APPLE COMPUTER, INC. OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.*

Parts of WebKit used by the QtWebKit module

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY UNIVERSITY OF SZEGED ``AS IS *AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL UNIVERSITY OF SZEGED OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.*

Parts of WebKit used by the QtWebKit module

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. 3. Neither the name of the project nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

```
src/3rdparty/ce-compat/ce_time.c
```


Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

•Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

•Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

•Neither the name of the University of Cambridge nor the name of Apple Inc. nor the names of their contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Parts of WebKit used by the QtWebKit module

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

•Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

•Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

•Neither the name of the University of Cambridge nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

Parts of WebKit used by the QtWebKit module

Parts of WebKit used by the QtWebKit module

Included in util/unicode/data, tests/auto/qtextboundaryfinder/data and tests/auto/qchar Parts of the makeqpf tool

HDF5

NETCDF

NETCDF

http://www.unidata.ucar.edu/software/netcdf/copyright.html

Ticpp

Ticpp

VTK

VTK is an open-source toolkit licensed under the BSD license.

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 the same information can be viewed 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.

Related Topics

- FAQ
- Setting Up WMS

Hardware Locks

XMS Hardware Locks

There are different types of hardware locks that the XMS software supports. Select the type that you use to find out installation and troubleshooting information for your lock.

Single User USB Hardware Lock

1. Install the hardware lock drivers. If hardware lock drivers have already been installed, skip to the next step. The drivers can be installed by running the **Sentinel System Driver Installer.exe** program found in the WMS installation directory.
2. After installing the hardware lock drivers, plug the Aquaveo hardware lock into an available USB port.
3. Start XMS and the program should automatically detect the hardware lock. If the Welcome dialog appears, click the **Register…** button.
4. Select **Hardware lock** for the *Licensing method* and click the **Next** > button.
5. In the *Hardware lock* options, select **Get license from a single user lock** and click the **Next >** button.
6. If the registration is successful, click **Finish** to exit the *Registration Wizard* .
7. The *Register XMS* dialog displays the registered components, licensing method, and license expiration dates.

Update a Single User USB Hardware Lock

1. Plug the Aquaveo hardware lock into a computer with hardware lock drivers and XMS installed.
2. Start XMS 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 XMS.

3. Select **Hardware lock** for the *Licensing method* and click the **Next >** button.

4. In the *Hardware lock* options, select **Modify lock on this computer with the following code** and click the **Next >** button.

5. In the *Hardware Lock* dialog, click the **Next >** button to burn the hardware lock.

6. If the registration is successful, click **Finish** to exit the *Registration Wizard* .

7. The *Register XMS* dialog indicates the registered components, the licensing method, and the license expiration dates.

Setup a Network License Server

1. Install the Sentinel installation program that includes hardware lock drivers and the Sentinel Protection Server software. The installation program can be downloaded at www.aquaveo.com/downloads .

2. In the Sentinel installation wizard, select "Complete" for the setup type.

3. After installing the Sentinel lock drivers and server software, plug the Aquaveo hardware lock into an available USB port.

4. Ensure that the computer with the network hardware lock can be seen by other computers on the local network. Client machines can connect to the server by hostname or IP address.

5. The License Server is now ready to provide WMS licenses to client machines. Refer to the instructions for registering XMS with a Network USB Hardware Lock for more information.

Network USB Hardware Lock

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 XMS.

2. Select **Hardware lock** for the *Licensing method* and click the **Next >** button.

3. In the *Hardware lock* options, select **Get license from a network lock** and click the **Next >** button.

4. Enter the **IP address** or **Host name** of the server hosting the network hardware lock.

5. Click the **Browse Lock Setting...** button. This opens a web browser and tests the a connection to the hardware lock over a local network.

6. Click the **Apply Lock Setting...** button.

7. Once the "Lock license acquired" message appears, click the **Finish** button.

8. The *Register XMS* dialog displays the registered components, licensing method, hardware lock serial number, and license expiration dates.

Update a Network USB Hardware Lock

1. Plug the Aquaveo hardware lock into a computer with hardware lock drivers and WMS installed.

2. Start XMS 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 XMS.

3. Select **Hardware lock** for the *Licensing method* and click the **Next >** button.

4. In the *Hardware lock* options, select **Modify lock on this computer with the following code** and click the **Next >** button.

5. In the *Hardware Lock* dialog, enter the number of licenses to burn on the lock. This number is typically the same as the number of licenses purchased and available. Click the **Next >** button to burn the hardware lock.

6. If the registration is successful, click Finish to exit the *Registration Wizard* . Please note that the *Registration* dialog may not show the enabled components. To verify the enabled components on the network lock, refer to the instructions for registering XMS with a Network USB Hardware Lock.

7. Return the network hardware lock to the computer serving as the license server if necessary.

General Notes about Hardware Locks

1. Remote Desktop cannot be used with single-user/standalone locks, only with network locks as stated in Safenet Sentinel's User Manual (pg 28):

- "Please Note that if you attempt to run a Sentinel Key protected application in standalone mode via a remote client (Terminal Server, VNC, WinXP remote client...), the software protected with Sentinel keys will not allow this for security reasons. You will need to run the software while directly logged into the machine."

2. General Information

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

Download SSURGO data

Related Topics

- Getting Started

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 Quick Tour provides a good overview of the interface layout, as well as other things. It's recommended to go through the Quick tour. Specifically the quick tour shows 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 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 to 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 some 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 dragging a file from Windows Explorer into the main graphics window of WMS, the file 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. The files can be opened with Adobe Acrobat and printed if desired.

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

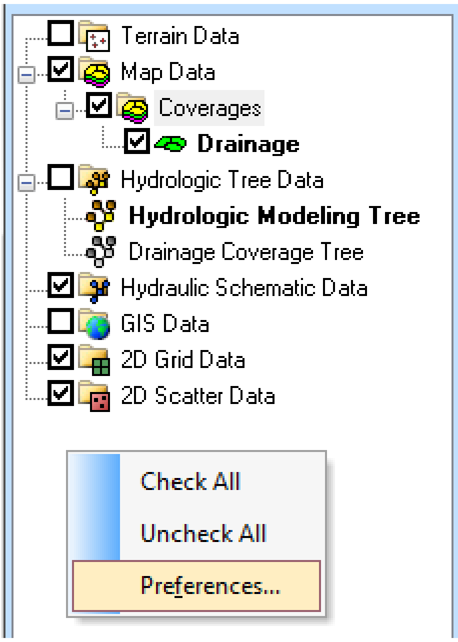
Project Explorer Overview

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:

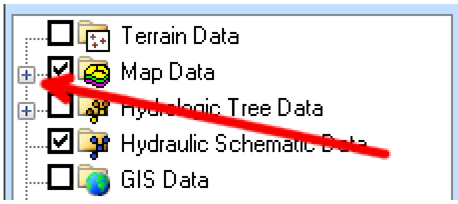
•By default WMS will not switch modules when selecting one of the folders, or data objects within a folder from the Project Explorer, but the preferences can be changed so that each time a data folder/object is selected WMS automatically switches to that modules as if the module icon had been simultaneously 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.

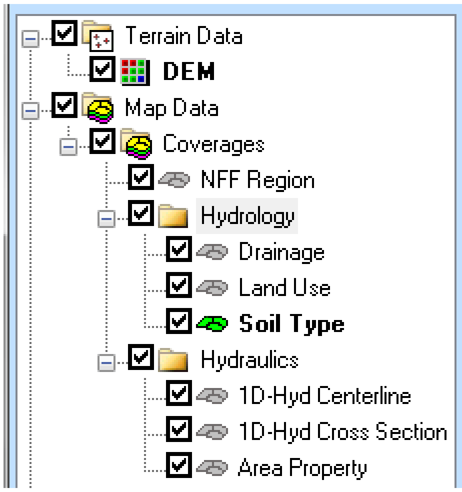
•The scroll Project Explorer when changing the module option forces WMS to scroll the Project Explorer such that the data folder corresponding to the active module appears at the top of the window.

•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.

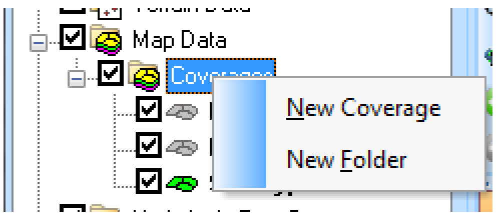


The display of all objects within a folder, or individual data object can be turned on or off using the check box at the left side as shown below.



Creating Data Objects and Folders

New folders and data objects are created by right-clicking the mouse on the folder where the new object is to be stored (new folder and data objects 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]

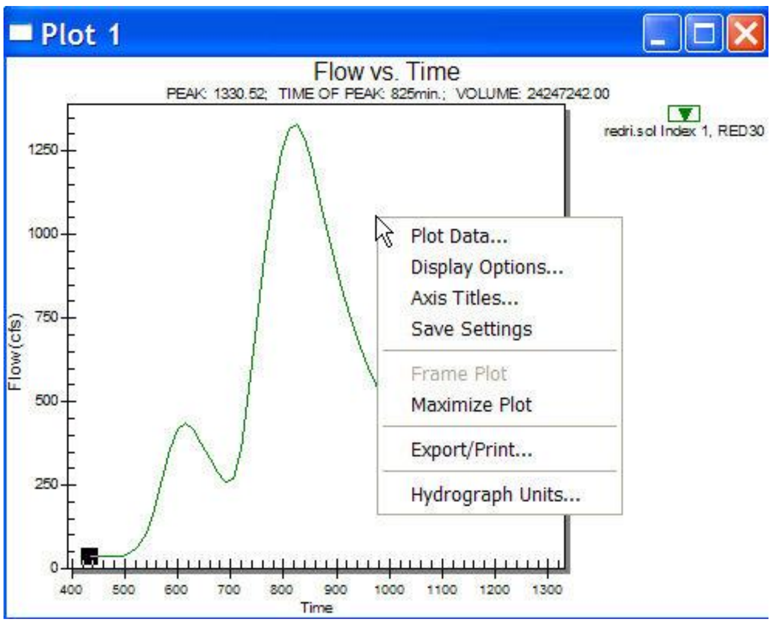
•[Preferences]

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 control of 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 allows exporting the 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.

Related Topics

•Hydrographs

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 regularly working 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.

Step 2 of the

using the MFHWA Storm Drain HGL plot.

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 three menus (*File* , *Edit* , *Display*) 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

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.

Examples of toolbars include:

- Module Palette – Used to switch between modules.
- Macros – Frequently used commands such as **Save** , **Open** , and **Frame** .
- Units Toolbar – Displays the planimetric and elevation units.
- Digitize Toolbar – Used to create TIN vertices with known elevations.
- Static Tool Palette – Has a list of tools available in every module.
- Dynamic Tool Palette – Contains tools that vary with the active module.
- Drawing Tools – Contains tools for creating annotations.
- Get Data Tools – Used to get data from the model or from online sources.

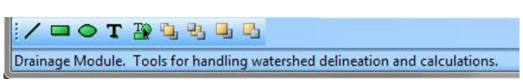
Related Topics

- WMS Screen

Help Strip

There are two ways to get context sensitive help. When holding the 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 which displays 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

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 switching 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

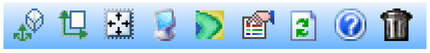
•2D Grid Module

•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:

•**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 as to look down at 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.

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).

Units Dialog

This dialog is also accessible through the *Edit* | **Units** menu command. Model units are the units of the geometric data (i.e. TIN, DEM, Feature Objects, etc.) uses to develop hydrologic and hydraulic models whereas parameter units are the units 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.

Model Units

- **Current Project** – brings up the *Display Projection* dialog.

Parameter Units

- *Basin Areas* – sets units for basin areas. Options include: "Square miles", "Square kilometers", "Acres", and "Hectares".
- *Distances* – options include: "Feet", "Meters", "Kilometers", and "Miles".

Drain Data Display Options

This button will bring up the *Display Options* dialog.

Related Topics

- Units

Digitize Toolbar

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

For example, if there is an image of a contour map with known contour elevations, 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 all the vertices have been defined on the TIN, turn off the digitize mode and triangulate the TIN to use the TIN in the hydrologic or hydraulic model.

Related Topics

- Creating TINs

Static Tool Palette

The tools 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:

•**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, 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:

•**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 to be fitted to 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.

•**Zooming Out** – To zoom out select the **Zoom** tool and hold down the *SHIFT* key (notice that the plus sign inside of the **Zoom** tool changes to a minus sign while moving the cursor). To zoom out, click on a point in the graphics window. WMS will zoom out with the point clicked on at the center of the screen. The display is zoomed out by a factor of two.

•**Framing** – To zoom in or out so that everything just fits within the graphics window, select *Display* | **Frame Image** , or by select 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** tool .

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.

View Previous Tool

View Previous – redraw everything in the graphics window. Selecting the *View* | **Previous View** command of the *Display* , or the *View Previous* tool, restores the Graphics Window viewing parameters to those in place before the last viewing command was issued: rotate, zoom, pan, etc.).

View Next Tool

View Next – Active after using the **View Previous** macro. Redraws everything in the graphics window to how it was before using the **View Previous** command.

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 deselected.

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 deselected by holding down the *Shift* 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.

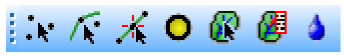
Clear the selection list at any time by clicking on a portion of the graphics window where no objects exist. This effectively clears the selection list because whatever is currently selected becomes deselected, and since having clicked in a location where no objects exist, nothing is placed in the selection list.

Module Tool Palettes

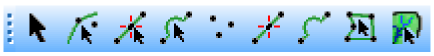
Each set of tools within a dynamic tool palette corresponds to a module, or data objects within a module. For example the tools used to create, edit, and select TIN data are different from those used for feature objects or DEMs. Click on each palette image to get a further description of the tools in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

Terrain Data Module

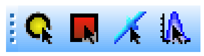
Drainage Module



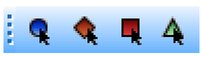
Map Module



Hydrologic Modeling Module



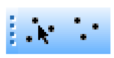
River Module



GIS Module

2D Grid Module

2D Scatter Module



Related Topics

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

Drawing Tools

Drawing tools allow the user to manual manipulate and create certain objects depending on the active module when the Map, Terrain, or Drainage modules are activated. These objects are created in layers, like sheets of paper stacked on top of one another. Only one tool is active at any given time.

Tools

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.

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 the user wishes 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 the user wishes 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 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 the user to enter the text string and choose the font, color, etc.

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.

Arrangement Tools

The arrangement tools allow moving drawing objects up or down in the layers.

Move to Front brings the selected object to the front (or the top) of the layers, so all other layers are below it.

Move to Back moves the selected object to the back, so all other layers are above it.

Shuffle Up moves the selected object up one layer.

Shuffle Down moves the selected object down lone layer.

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.

Add GIS Data

The **Add GIS Data** tool brings up an *Add GIS Data* browser dialog where GIS files can be opened. This tool is not active if ArcObjects are enabled.

Get Data From Map Tool

The **Get Data From Map** tool is allows opening a web map service as an online image and use it as any other image in the WMS interface. This option imports the data as a static (locally saved) image.

Get Online Maps

The *'Get Online Maps* tool allows opening a web map service as an online image and use it as 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 the WMS project and that displays much faster than the online image.

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 information.

Hydrologic Modeling Wizard

The WMS *Hydrologic Modeling Wizard* is a tool that walks 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.

HY-8 Modeling Wizard

The WMS *HY-8 Modeling Wizard* is a tool that walks through all the steps involved in creating a HY-8 model. It can be accessed by selecting the **HY-8 Modeling Wizard** tool in the *Get Data Toolbar* .

See the article HY-8 Modeling Wizard for more information.

Hydraulic Toolbox

The **Hydraulic Toolbox** tool launches the Hydraulic Toolbox software. This software is used in conjunction with WMS for a number of modeling processes.

See the article Hydraulic Toolbox for more information.

HY-8

The **HY-8** tool launches the HY-8 software. This software is used in conjunction with WMS for a number of modeling processes.

See the HY-8 documentation for more information.

MWBM Modeling Wizard

The *MWBM Modeling Wizard* is a tool that walks through all the steps involved in creating a MWBM model. It can be accessed by selecting the **MWBM Modeling Wizard** tool in the *Get Data Toolbar* .

See the MWBM User Manual for more information.

Related Topics

- Get Data Tool
- Toolbars

Get Data Tool

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

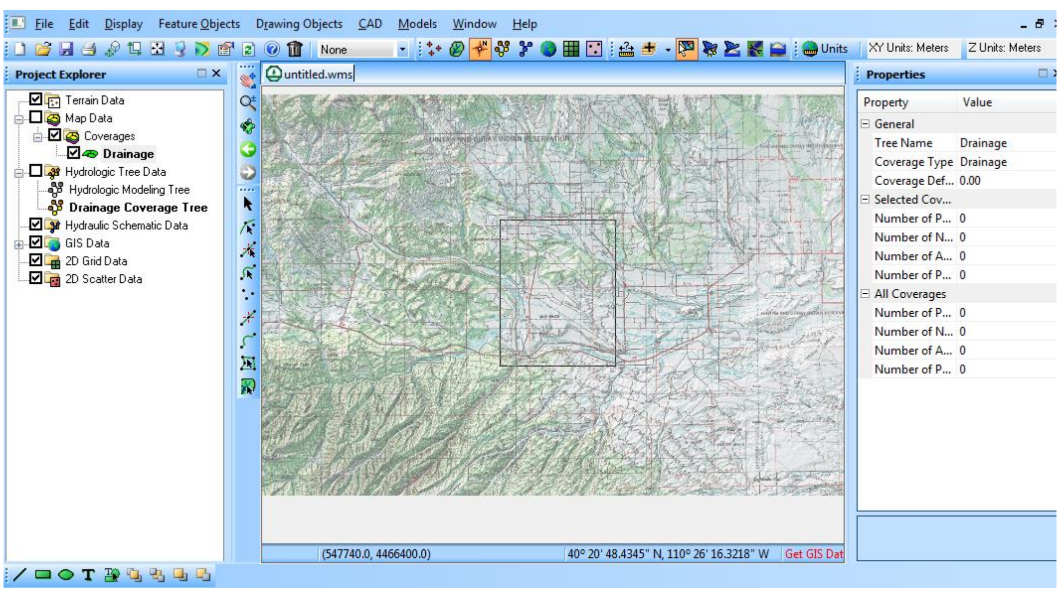
Common problems when beginning a project in WMS include the need to locate the necessary data (e.g. DEMs, Topo Map images, shape files of land use and soils, etc.), make sure the file format is correct, open in WMS, and then ensure that all of the data are in a common coordinate system. This can be tedious, 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

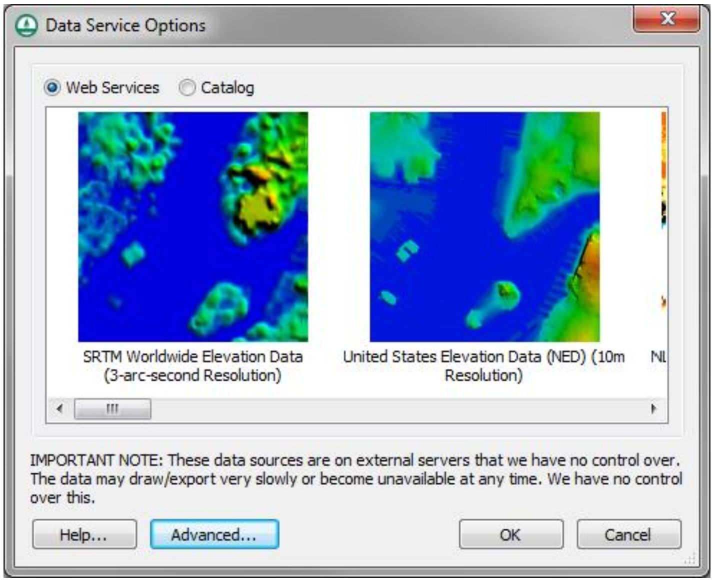
The **Get Data** tool can be used to select a geographic area of interest for a modeling project. Available files, as defined by a data catalog file, within the bounding box are opened for use within WMS. The following steps are followed when opening data with the **Get Data** tool:

1. Open a georeferenced image file that covers the extents of the data defined within the catalog

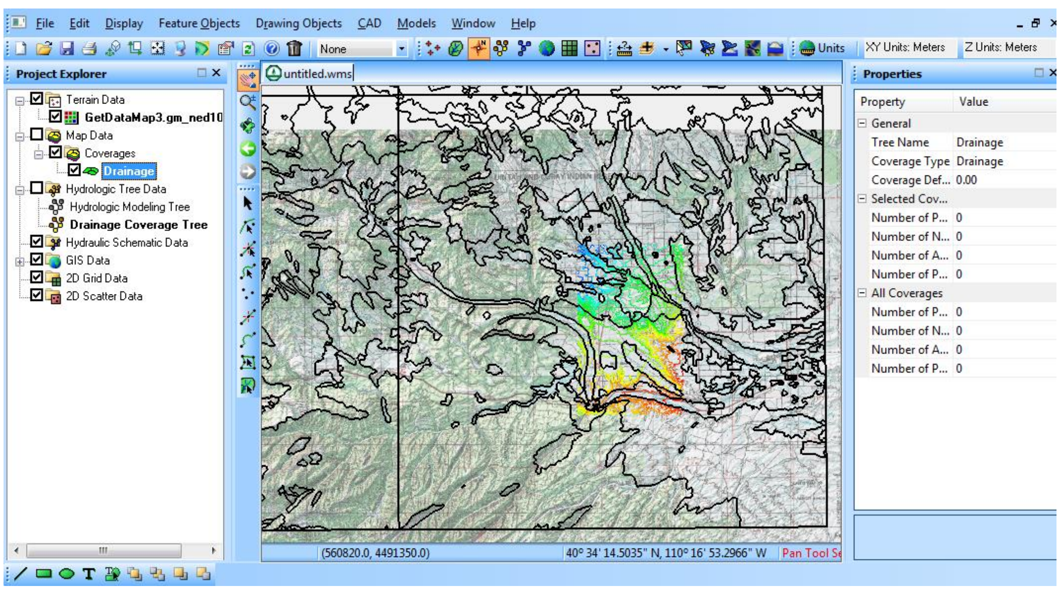
2. Select the **Get Data** tool and drag a rectangle in the main graphics area that contains the area for the project



3. 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 the files to obtain or 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    YMAXC:\temp\Catalog\DEMs\40110a4.dem
542548.3247  553288.3247  4427875.406
4441795.406C:\temp\Catalog\DEMs\40110a5.dem   531898.5074  542608.5074
4427845.404  4441735.404C:\temp\Catalog\DEMs\40110a6.dem   521248.6896
531928.6896  4427785.355  4441675.355IMAGE-TOPO   XMIN    XMAX    YMIN
YMAXC:\temp\Catalog\QuadImages\pq1831.tif  542548.3247  553288.3247
4427875.406  4441795.406C:\temp\Catalog\QuadImages\pq1830.tif  531898.5074
542608.5074  4427845.404  4441735.404C:\temp\Catalog\QuadImages\pq1829.tif
521248.6896  531928.6896  4427785.355  4441675.355IMAGE-PHOTO   XMIN    XMAX
YMIN    YMAXC:\temp\Catalog\PhotoImages\q1831_83.jpg  542548.3247
553288.3247  4427875.406  4441795.406C:\temp\Catalog\PhotoImages\q1830_83.jpg
531898.5074  542608.5074  4427845.404
4441735.404C:\temp\Catalog\PhotoImages\q1829_83.jpg  521248.6896  531928.6896
4427785.355  4441675.355SHAPEFILE-LANDUSE   XMIN    XMAX    YMIN
YMAXC:\temp\Catalog\LandUse\salt_lake_city.shp  414534  584057  4428200
4539297SHAPEFILE-SOILTYPE   XMIN    XMAX    YMIN
YMAXC:\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 WMS cannot mix file types. WMS 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. It is still necessary 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 there are 10 meter and 30 meter DEMs in the same dialog the second part of the descriptor is used in the combo box that allows specifying which group 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.

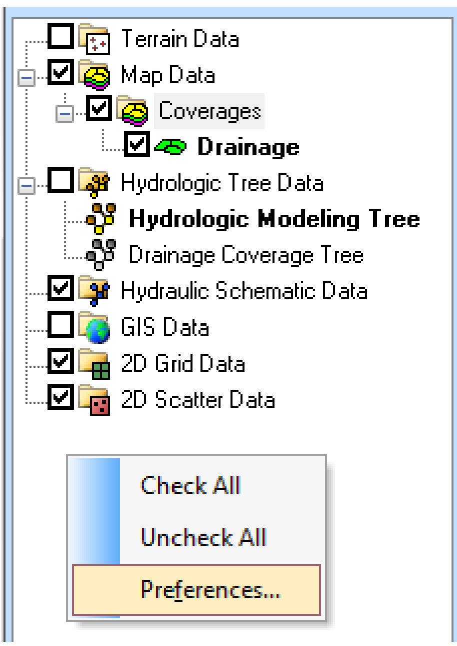
Film Loop Playback

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. WMS will launch the *Play AVI Application* if the film loop has been generated in WMS. The buttons at the upper left of the *Film Loop* dialog are designed to mimic the buttons on a VCR or CD player.

Preferences

Changing the preferences for the WMS interface can be done through the *Preferences* dialog. This dialog can be reached by going to the *Edit* menu and selecting the **Preferences** command or through the Project Explorer right-click menu.

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 the preferences can be changed 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 and later deals with images differently than past versions. The *Image Preferences* tab controls these options, including building pyramids and image conversion.

The *preferences* dialog containes the following tabs:

Files

The *Files* tab of the *Preferences* dialog displays all the system files being used by WMS. The file name and file path are show. Clicking the **Browse** button next to a file allows redirecting to a different file.

General

The *General* tab contains the following options:

•*Defaults*

- **Restore Original Defaults** – Clicking on this button will reset all of WMS's preference to the default factory condition.

•*Confirm deletions* – Selecting this option with cause the *Confirm Deletion* dialog to appear when deleting a feature object.

•*Confirm scale factor before copying*

- *Bitmap scale factor*

•*Use compression with XMDF files*

- *Compression factor*

•*Specify CAD symbols size*

- *CAD symbol size*

•*Show Welcome Dialog* – When turned on, the WMS Welcome dialog will appear when WMS is launched and WMS will check for updates.

•*Show Prince George's Station Dialog*

•*Check for newer version on start-up*

•*Reverse the zoom direction of mouse wheel*

Project Explorer

This tab has two options that can be toggled on or off:

•*Change module when tree selection changes* – toggling this option will cause the selected module in the *Module Pallette* to change to match the module selected in the Project Explorer.

•*Scroll project explorer when changing module* – when this option is toggled on, selecting a module in the *Module Palette* will cause the *project explorer* to scroll so that related module is visible in the Project Explorer. If the module can already be seen in the *project explorer* it will not change.

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, specify where the converted JPEG is saved.

Graphics

The *Graphics* ' tab contains the following options:

•*Active Graphics Library*

- *Ask* – Will cause WMS to ask which library to use if system does not support all functionality.
- *Autoselect* – WMS will use software library if system does not support all functionality.
- *System* – Will cause WMS to always use the system library which may not support all features.
- *Software* – Will cause WMS to always use the software library which may be slower.

Related Topics

•Project Explorer Overview

•Images

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 selecting a feature polygon representing a basin boundary, WMS will also select the associated basin in the hydrologic tree.

When selecting a transient (time-varying) dataset in the Project Explorer, all the time steps associated with the transient dataset are shown in the *Properties* window. 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
    a) Name
    b) Various other properties
2. Selected DEM cell properties
    a) I value
    b) J value
    c) DEM cell elevation
3. Feature Arcs
    a) ID
    b) XYZ Length
    c) Plan (XY) Length
    d) Arc Type
    e) Arc Basin ID
    f) Basin Name
    g) Basin Area
    h) Cross Section Attributes (for cross section arcs)
4. Feature Polygons
    a) Polygon ID
    b) Polygon Area
    c) Polygon Type
    d) Polygon Basin ID
    e) Basin Name
    f) Basin Area
5. Feature Points/Nodes
    a) ID

   b) X Coordinate
   c) Y Coordinate
   d) Z Coordinate
   e) Point Type
   f) HY-8 Crossing Name (if applicable)
6. TIN/2D Scatter Point Vertices
   a) ID
   b) X Coordinate
   c) Y Coordinate
   d) Z Coordinate
   e) Scalar Value
7. TIN Triangles
   a) Number (ID)
   b) Area
8. Hydrologic Tree Outlet Points
   a) Outlet Name
   b) Upstream Area
   c) Downstream Slope
   d) Downstream Length
   e) Tree X Coordinate
   f) Tree Y Coordinate
   g) Tree Z Coordinate
9. Hydrologic Tree Basins
   a) Model-specific Hydrologic Properties
   b) Area
   c) Other Geometric Properties
10. Hydrologic Diversions
   a) Diversion Name
11. Hydrographs
   a) Peak Flowrate
   b) Time of Peak
   c) Volume
12. Hydraulic Reaches
   a) River Name
   b) Reach ID

c) Reach Name

d) Computational Feature Length

e) Measured Length

f) Start Station

g) End Station

h) Hydraulic Schematic X Location

i) Hydraulic Schematic Y Location

j) Hydraulic Schematic Z Location

13. GIS Shapes

a) Layer Name

b) Layer Type

c) Filename

d) Extents (Left, right, top, and bottom)

e) Listing of the fields for the selected shape and their values

14. 2D Grid Cells

a) I Value

b) J Value

c) 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.

Keyboard Shortcuts

Many commands in WMS are can be accessed using keyboard shortcuts.

Standard Menu Shortcuts

Shortcuts for standard menu commands are listed in the table below.

Keyboard Shortcuts

Modifier	***Key***	***Command***
CTRL	*F*	*Display* | **Frame Image**
CTRL	*R*	*Display* | **Refresh**
	DELETE	*Edit* | **Delete**
CTRL	*C*	*Edit* | **Copy to Clipboard**
CTRL	*A*	*Edit* | **Select All**
CTRL	*O*	*File* | **Open**
CTRL	*P*	*File* | **Print**
CTRL	*S*	*File* | **Save**
CTRL	*N*	*File* | **New**
CTRL	*X*	*File* | **Exit**
	F1	**WMS Help**
	F2	**Pan**
	F3	**Zoom**
	F4	**Rotate**
CTRL + SHIFT	*A*	*Display* | *View* | **View Angle**
CTRL + SHIFT	*W*	*Display* | *View* | **Set Window Bounds**
CTRL + SHIFT	*F*	*Display* | *View* | **Front**
CTRL + SHIFT	*O*	*Display* | *View* | **Oblique**
CTRL + SHIFT	*V*	*Display* | *View* | **View Options**
CTRL + SHIFT	*P*	*Display* | *View* | **Plan**
CTRL + SHIFT	*Z*	*Display* | *View* | **Previous**
CTRL + SHIFT	*S*	*Display* | *View* | **Side**
CTRL + SHIFT	←	*Display* | *View* | **Previous View**
CTRL + SHIFT	→	*Display* | *View* | **Next View**
CTRL	*Z*	*Display* | *View* | **Z Maginification**

Related Topics

- Graphical User Interface (GUI)
- Menu Bar

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 will appear asking if still wanting to save changes to the project, 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 wanting to restore all of the settings, 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, the extensions for *.tin and *.dem are visible, 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

Dragging 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 accessing 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 *File Import Wizard* . Like the *File 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 *File 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 File 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 authenticating 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 or the **Print** macro. 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 what is needed. 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, a prompt appears 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 entering 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 is read into WMS. For example if a TIN is read in which has coordinate vertices in meters then WMS should be told that the project is 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.) the project is using to develop hydrologic and hydraulic models whereas parameter units are the units 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.

Units can also be changed in the Units Toolbar

Reproject

See Reproject for more information.

Single Point Reprojection

The **Single Point Reporjection** dialog converting 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 Projections 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.

The top material is the "Disable" material. Assign this material type to a feature object to remove a previously assigned material type. Objects with the "Disable" material type are read as though no material was assigned. The "Disable" material cannot be deleted.

Preferences

Opens the *Preferences* dialog. See *Preferences* for more information.

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 pasting tabular data (generally this is xyz data used to create a TIN or scattered dataset) 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 *File Import Wizard* is launched.

Obsolete Commands

The following commands are not longer available in current releases of WMS.

Coordinate Conversion

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

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** – Brings up the *Display Options* dialog.
- **Contour Options** – Brings up the *Contour Options* dialog.
- **Grid Options** – Brings up the *Grid Options* dialog.
- **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.

    1. *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* – a submenu with the following commands:

- **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 looking down at 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** – Brings up a *Plot Window* for the selected hydrograph .
- **Plot Wizard** – Launches the *Plot Wizard* .

Related Topics

- File Menu
- Edit Menu

Help Menu

The *Help* menu is one of the standard menus and is available in all of the modules. The commands in the *Help* menu are to assist in using WMS.

The commands in this menu are:

WMS Help

Accesses the WMS Wiki if an internet connection is available.

Register WMS

Brings up the *Register* window. Here which components have been registered are shown and changes can be made to the registration. See the article Registering WMS for more information.

About

Brings up the *About WMS* dialog that contains information about the software version and build date. It also includes copyright information and information for contacting technical support.

Check for Updates

Searches for updates to the current version. This command requires an internet connection to function. If updates are found, the option is given to install the latest version.

Related Topics

- Menu Bar

Window Menu

The *Window* menu is one of the standard menus available regardless of the current module and model. It contains commands on how to arrange windows that are active in WMS. The *Window* menu includes the following commands:

Cascade

Arranges all windows in an overlapping fashion within the WMS Graphics window.

Tile

Arranges all windows as non-overlapping vertical tiles within the WMS Graphics window.

Tile Horizontally

Arranges all windows non-overlapping horizontal tiles within the WMS Graphics window.

Active Window

A list of the currently open graphics and plot windows is shown at the bottom of the *Window* menu. A check mark appears in front of the active window. Choose a window from the list to make it active.

Related Topics

- Menu Bar

3.2. Coordinate Systems

Projections

Related Versions	
GMS	v9.1
SMS	v11.1
WMS	v9.1
version note	

Projection refers to a map projection like UTM . In XMS software, a projections are associated with the project and individual data objects.

Starting with SMS 11.1 and GMS 9.1, the XMS software works on the concept of a "Display Projection". This is the projection being worked in. Each geometric object loaded into the XMS package, such as a Scatterset, DEM, grid or mesh, also has an associated projection. These two projections may not be the same. If the two projections are compatible (i.e. it is a viable option to convert from one projection to the other), the XMS package will convert the data from the object projection to the display projection, just for display purposes. This is referred to as "Project on the fly". The data itself maintains the values associated with the object projection so when the XMS package saves a project, the data files are not modified.

Alternatively, data can be reprojected from one projection to another, actually changing the data values that will be saved as part of a project. This is done by right-clicking on the geometric object in the project explorer (data tree) and selecting the **Reproject...** command. The SMS package includes a feature to reproject all data from whatever projection the data is currently in to a single projection using the **Reproject All...** command in the *Display* menu.

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

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

Display Projection

The display projection, or the projection currently associated with the project, can be specified via the *Display* | **Projection** menu command. This setting controls how the XMS application displays (or interprets) data. Data defining objects with a specified projection are converted to the display projection (if it is different from the object projection) for display purposes only. This is referred to as "projection on the fly". The data saved to files as part of a project, or exported for a simulation are exported in the object projection. Display projection only affects the display.

Objects without a specified projection are assumed by the XMS application to be referenced to the display projection.

The display projection is saved as part of a project file. Specify a display projection as part of the system settings for new sessions/projects.

When a data object is read into an XMS application, for which no data has yet been loaded and no projection has been specified, and the data object has its own projection, the display projection is reset to the object projection. This allows defining a working projection simply by loading data that uses that projection (and has a projection definition such as a *.prj file to define it.)

No Projection (Previously Local Projection)

Many numerical models work in local or model space, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). XMS software allows for this using a Display projection set to local or no projection option. This is standard practice when building a numerical model of a flume test. The units of the model are also specified as part of the projection. If the display projection is in this mode, no global projections are allowed on individual objects.

(Note: when the display projection is set to *No Projection* or *Local Projection* , the data may still be referenced to a projection. The display projection can be changed to reflect that projection if desired.)

Global Projection

Data referenced to a global projection can be easily correlated and used with other applications that utilize projections including GIS and CAD. When the display projection is specified as a global projection, the XMS application can export georeferenced images, shapefiles, and KMZ files that may be directly imported to other applications.

Selecting the *Global Projection* option will automatically bring up the *Select Projection* dialog where a global projection can be chosen. If the *Select Projection* dialog does not automatically appear, or desiring to change the current global projection, then the **Select Projection** button in the projection dialog can be used to access the dialog.

Object Projection

Each geometric object loaded into a session can have an associated projection. When an object is loaded from a file, the XMS application looks for a projection either in the object data file or in an associated *.prj file. If no projection is found, the object is left with no projection or floating. In this case, the object is assumed to be related to the display projection, regardless of what that projection is. The object projection can be specified by right-clicking on the object in the project explorer and selecting the **Projection...** command. The default projection displayed in the dialog that appears is the object's projection if it has one, and no projection otherwise. In the case of no projection, the display projection is filled in as the default global projection should that option be selected.

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 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 projection or coordinate system.

There are four basic reprojection tasks:

•Reproject on the fly, which just displays all data in a specified projection without changing the base values

•Reprojecting the entire project from one system to another

•Reprojecting one geometric object (i.e. mesh or grid) from one coordinate system to another

•Single point reprojection, which allows entering 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.

Reproject on the Fly

When data from multiple projections are loaded into and XMS application, without a defined projection, they do not overlay and the display shows data clusters at two distinct locations. With project on the fly, if the data object has a defined projection (such as a *.prj file), this data would be reprojected on the fly to the display projection.

If data does not line up due to incorrect or incomplete projection specification, specify different object projections to attempt to align the data correctly. Object projection is specified by right-clicking on the object in the project explorer.

Reproject everything

Reprojecting everything can be done by selecting the *Display* | **Reproject All...** menu command. This will convert all the data loaded into the XMS application from the object projection(s) to a specified projection. This operation brings up a dialog which allows specifying the desired projection. The default value is the display projection currently specified for the project.

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 and sececting **Reproject...** . If the object does not have a specified projection, this command is not available. It can be accessed by selection the **Projection...** command for the object in the same right-click menu and defining a projection for the object.

When the object has a projection, this command reprojects from one projection to another. The command brings up a dialog with a "from" projection specified on the left and a "to" projection specified on the right. The "from" projection is defaulted to the object projection. The "to" projection is defaulted to the display projection.

Single Point Reprojection

Single Point Reprojection command is found in the *Display* menu and allows entering 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 allows creating a feature point at the new location.

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) library which supports hundreds of standard projections.

Related Topics

•Projection Dialogs

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:

$$newpoint_x = R * \left(point_{longitude} - origin_{longitude}\right) * \cos\left(origin_{latitude}\right)$$

$$newpoint_y = point_{longitude} * R$$

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

$$newpoint_{longitude} = \frac{origin_{longitude} + point_x}{R * \cos\left(origin_{latitude}\right)}$$

$$newpoint_{latitude} = \frac{point_y}{R}$$

$R = 6378206.4m.$ (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).

Related Topics

•Projections

•Projection Dialogs

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 projection options 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:

3. 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.

4. 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.

5. 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.

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, the display options of any module can be changed 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.

Changing Color

If selecting the drop-down arrow, a simplified color chooser appears that allows changing the color only of the attribute.

Changing Symbols

If wanting to change the symbol style or size, select the **symbol** button.

Changing Line Styles

If wanting 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* dialogs.

Changing Text Fonts and Styles

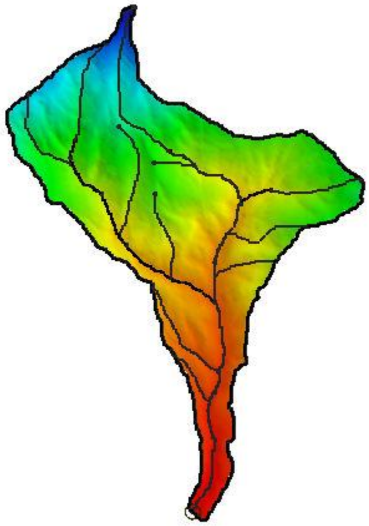
If wanting to change a font or font style, select the **text symbol** button.

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, 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 dataset). 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 there can be color filled contours for a DEM and linear contours on a TIN at the same time. The set of contour options being edited 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.

Specified Values

The *Specified Values* option allows entering 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 dataset 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 dataset 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 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
- 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 the user's eye are displayed in front of objects that are further. 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.

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 the 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 convert WMS data to CAD prior to saving if desiring 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 wanting to convert WMS data objects such as feature object coverages, TINs, contours, etc., 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, 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 the Map module is active (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 the 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 the TIN, select convert, select **TIN Contours→Feature** .
3. Remove any unwanted arcs from the floodplain boundary.
4. Delete any data not needed 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 the file as a DWG file.

Converting to Feature Objects

Many times CAD data layers are used to create streams, watersheds, or other feature objects. To do this, 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 are to be converted. When converting data, 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

3.6. 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 dataset is created by importing from a file, interpolating, or using the data calculator, the dataset becomes the active dataset for the object.

Data Calculator

The *Data Calculator* can be used to perform mathematical operations with datasets to create new datasets. The *Data Calculator* is accessed by selecting the **Data Calculator** command from the *Data* or *Edit* menu.

- 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.

Deleting Datasets

Datasets can be deleted by selecting the dataset in the Project Explorer and selecting the **Delete** key on the 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, it's necessary to classify the results of one or two datasets to create a single index map. For example, if there is one dataset showing sediment erosion and a second dataset showing sediment deposition, it could be desired to find areas with high sediment erosion and low deposition or vice versa. The **dataset zonal classification** tool helps to find these areas. When selecting the **dataset zonal classification** command from the *Data* menu, the following dialog appears:

Modify the parameters in this dialog to generate the type of index map to be generated. The *Select Dataset* option allows selecting 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 selecting a dataset, select the **Dataset Info** button to show the dataset properties. It is also possible edit the minimum values for each range. The minimum value for each range cannot be set above the maximum for the current range or below the minimum for the previous range.

If wanting to classify values in a single dataset, 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 viewing 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.

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 reading a TIN in from a file and then deleting 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 *Data* | **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 a prompt asks 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 to be plotted 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 selecting which datasets 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 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.

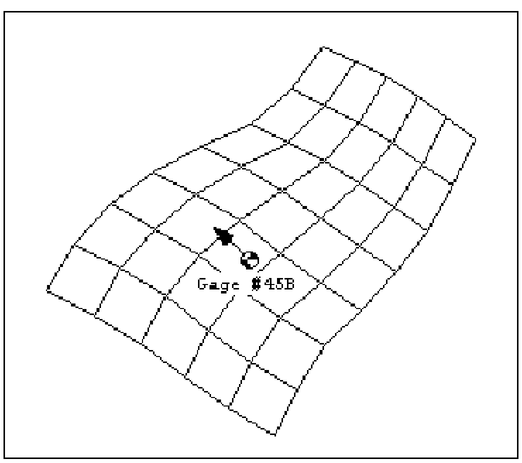
Gages Dialog

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

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

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 entering 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 defining 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 defining 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 deleting 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 entering 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 wanting 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, open the first HEC-1 file into an editor, then would insert the second file below (or above the first). Eliminate the job control records and the zz record in the middle of the new file and save. When opening 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. Some may wish to move individual icons around in order to display the tree differently. However, if wanting at anytime to have WMS rebuild the default display, choose the **Rebuild Tree Display** command of the *Tree* menu.

Related Topics

- Tool Palette

3.7. 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 most 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 some may want to see it, but often it is convenient to remove the collar (especially when 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 the new area can be saved as an image file.

Related Topics

- Map Module
- Registering an Image

Registering an Image

If an image file is not geo-referenced then it's necessary to define the coordinate system of the image. The *Register Image* dialog allows specifying 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 Image* 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® or Arc/Info® . 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 there are (x,y) coordinates in a different coordinate system than the project, the coordinates will need to be converted.

GMS Point Conversion

The **Convert Point** button in the image registration dialog will allow converting the coordinates.

SMS Point Conversion

The Single Point Conversion command in the *Edit* menu can be helpful if it becomes necessary to convert between any two coordinate systems. 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 it becomes necessary to convert between any two coordinate systems. Perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

4. Equations and Interpolation

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 dataset 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 $(x3, y3, z3)$:

$$A = y_1(z_2 - z_3) + y_2(z_3 - z_1) + y_3(z_1 - z_2)$$

$$B = z_1(x_2 - x_3) + z_2(x_3 - x_1) + z_3(x_1 - x_2)$$

$$C = x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)$$

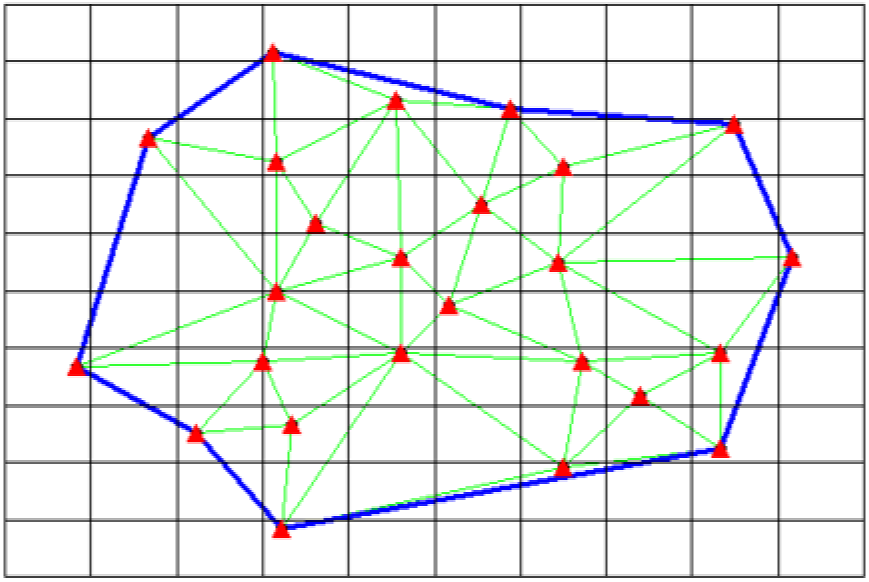
$$D = -Ax_1 - By_1 - Dz_1$$

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.



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 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_i 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.

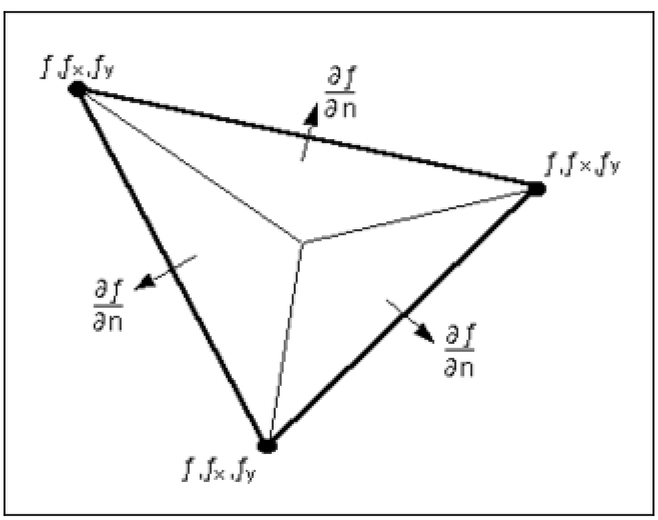
Related Topics

- 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:

$$F(x,y) = \sum_{j-0}^{3-i} c_{ij} x^i y^j$$

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.

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.

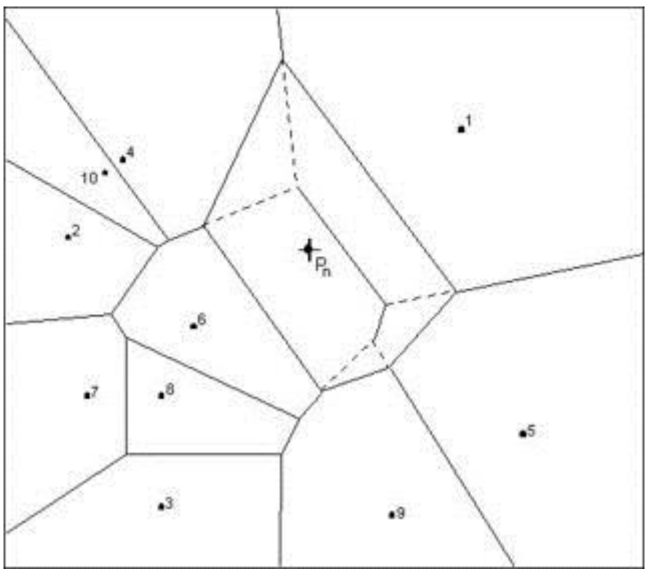
Each Thiessen polygon is constructed using the circumcircles of the triangles resulting from a Delauney triangulation of the scatter points. The vertices of the Thiessen polygons correspond to the centroids of the circumcircles of the triangles.

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.



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

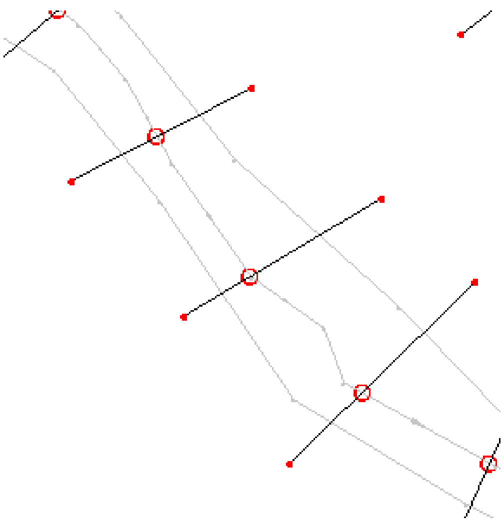
As shown in the figure above, the Thiessen polygons for scatter points on the perimeter of the TIN are open-ended polygons. Since such polygons have an infinite area, they cannot be used directly for natural neighbor interpolation. Thus, a special approach is used to facilitate extrapolation with the 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). 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

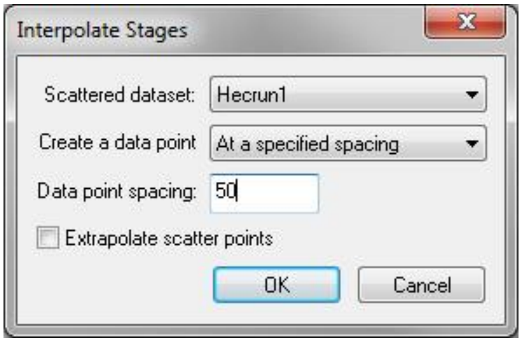
- Interpolation Options
- 2D Scatter Point Module

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.

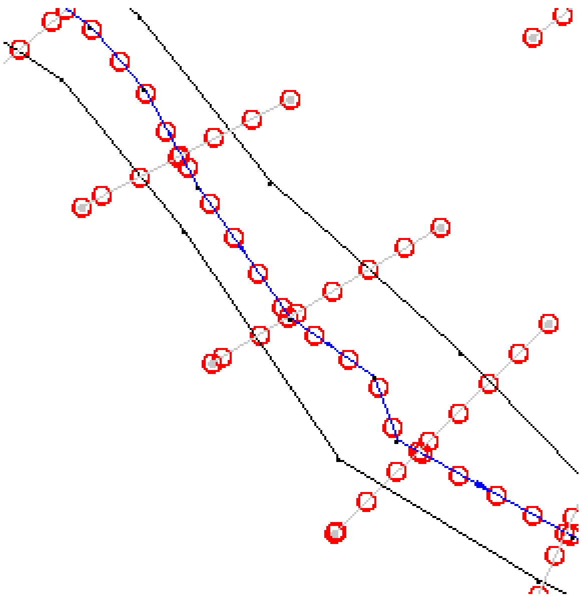


The flood plain delineation uses interpolation from the scatter set created by the hydraulic model solution, but the interpolation is not very accurate with such a sparse dataset (it often takes 6-8 points to interpolate and with such a sparse dataset these six or eight points could cover a long distance 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).

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.



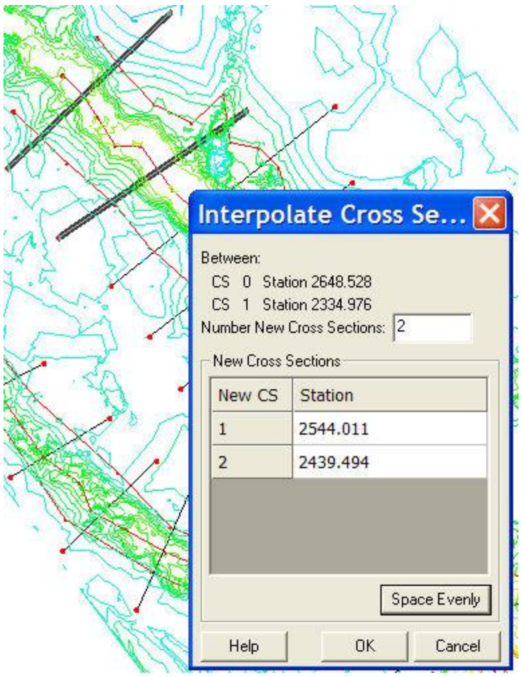
Related Topics

- Read HEC-RAS Simulation
- Flood Plain Delineation
- Preparing Stage Data

Interpolate Cross Sections

Using the **Interpolate Cross Sections** command in the *River Tools* menu, a user can create any number of additional cross sections between two selected cross sections. This allows a user 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

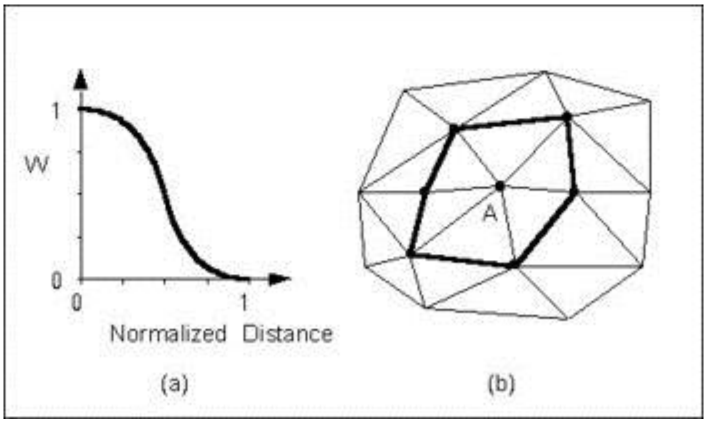
If the *Use subset of points* option is chosen, the *Subset Definition* dialog can be used to define a local subset of points.

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.

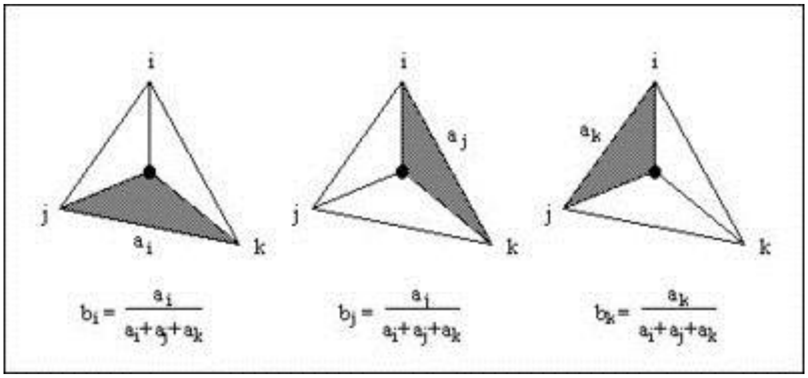


The influence of the weight function extends over the limits of the Delauney point group of the scatter point. The Delauney point group is the "natural neighbors" of the scatter point, and the perimeter of the group is made up of the outer edges of the triangles that are connected to the scatter point 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:



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.

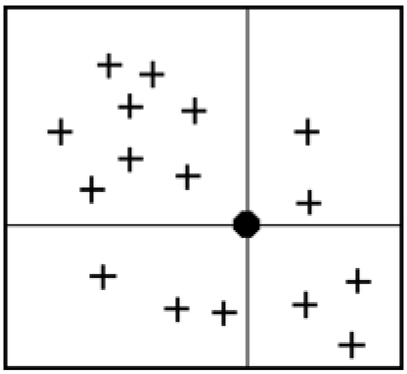
Related Topics

- 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 **Subset...** 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 selecting 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 an equation has been selected, the *Instructions / Results* window will identify any variables with zero values and instruct 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.

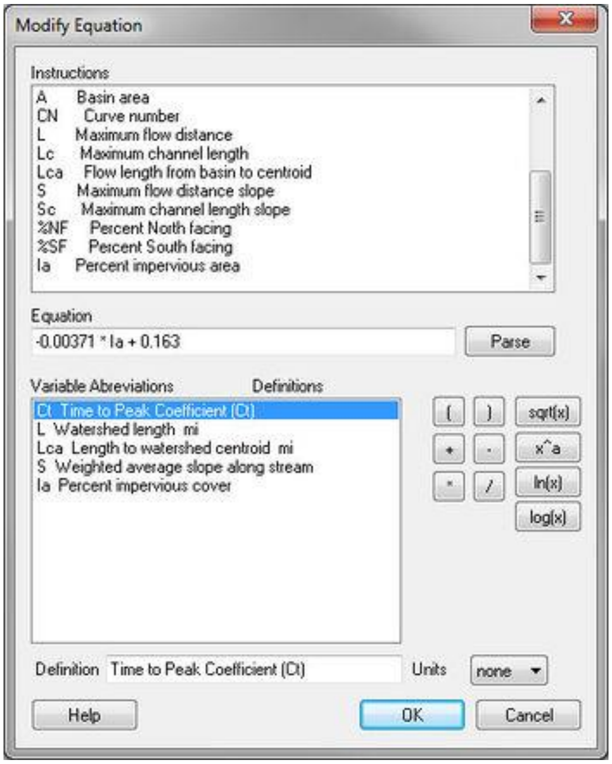
Related Topics

•[Overview of Basin Data Equations]

•[Travel Times from Basin Data]

Customizing Equations

Any of the equations can be edited (i.e. pick an equation and then modify it slightly to a particular need) or created using the **Modify Equation** or **User Defined** buttons. Both of these buttons bring up the *Modify Equation* dialog shown below.



If entering 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 WMS will ask to choose which equation to modify. If using the **User Defined** button to enter this dialog then the *Equation edit* box will be blank. Type a new equation or modify an existing equation from the keyboard or by using the buttons for add, subtract, multiply, divide, etc. Also enter variable names corresponding to the basin data computed by WMS, or enter variables that to define. 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 finished typing the equation, 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 the local hydrology manual requires the use of an equation not currently supported by WMS and this equation must be repeatedly used for projects, set up the equation in a text file in the same directory as WMS. It will be read in and set up as the user defined equation automatically. To do this, create a file named "wmstc.equ" and define one or more equation groups (a group may be used because sometimes the equation to compute a travel time may be a function of another equation). The following is an example wmstc.equ file.

```
EQGROUP  Myeq1EQ  1.49  /  n  *  R  *  Sc  DEF  Manning's  equation  UNITS
laghourVAR  n  DEF  Manning's  roughness  coefficient  VAL  .002  UNITS  none
WHICH  -1VAR  R  DEF  Hydraulic  Radius  VAL  0.0  UNITS  ft  WHICH  2EQ  XA
/  Pw  DEF  Hydraulic  Radius  UNITS  ftVAR  XA  DEF  Cross  section  area
VAL  0.0  UNITS  sqft  WHICH  -1VAR  Pw  DEF  Wetted  Perimeter  VAL  0.0
UNITS  ft  WHICH  -1EQGROUP  Myequ2EQ  Lc  /  V  DEF  Time  of  travel  UNITS
lagminVAR  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 selection 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 it is not necessary 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, 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 using a variable with units not in this list, 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 the variable should follow the WHICH keyword with the equation number. If it does not represent another equation then the variable should be followed with a -1. See the example above where the hydraulic radius variable R represents the second equation in the group (XA/Pw).

A user may define as many equations as desired in the file and each time WMS is started the equations will be read and become members in the list of equations that can be selected 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 or cells comprising this basin. A triangle or cell'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 choosing 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.), consider the assumptions made about a given equation, and try to identify one that used watershed conditions similar to the one being studied. 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 selecting from a series of pre-defined equations (Dodson & Associates, 1992), or create a user defined 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 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 (also any of the variables computed by WMS can be edited).

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 not wanting to have the equation updated when basin data changes then 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 either appending to an existing file (so that a single report for multiple basins can be created) or creating 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:

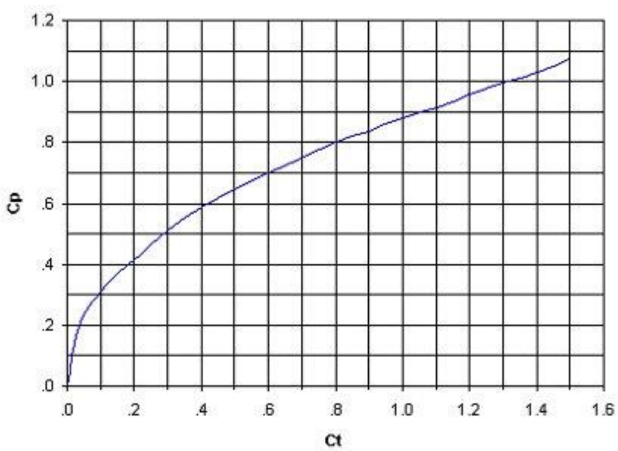
C_t = coefficient of watershed topography based on impervious area.

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.



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 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 Δt 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   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 R equals 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   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	0.2

bare soil	
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. When switching from one module to another module, the *Tool Palette* and the menus change. This allows focusing only on the tools and commands related to the data type being currently used 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.f

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. Terrain Data Module

Terrain Data Module

The Terrain Data module imports, edits, and prepares 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.

The Terrain Data module is included will all paid editions of WMS.

Related Topics

- DEMs
- TINs
- Floodplain Delineation
- Project Explorer Contents for Terrain Data Module
- Scattered Data
- Drainage Delineation

Terrain Data Tools

The toolbar for the WMS Terrain Data module has a variety of tools useful for editing and creating objects within that module. The tools are described below.

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 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.

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 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 having 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.

Place Contour Labels

The **Place Contour Labels** 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.

Set Contour Min/Max

Using the **Set Contour Min/Max** tool to dragging a box around part of a project in the Graphics Window changes the contours in the selected area to match the minimum and maximum settings in the *Contour Options* dialog.

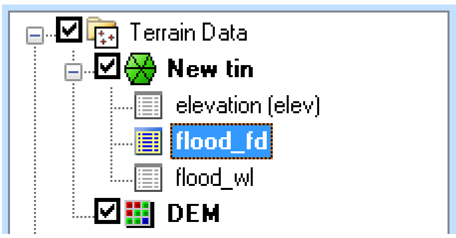
Right-clicking with the tool brings up a menu which has a command to clear the minimum and maximum contour settings.

Related Topics

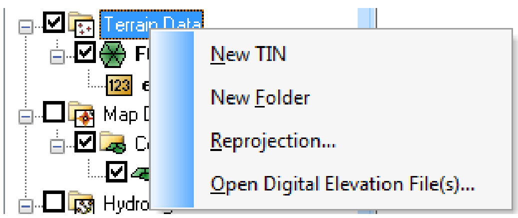
•Tool Palettes

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 creating a **New folder** or **TIN** . Because TINs can be created by digitizing in WMS, a new, blank TIN can be created. However, DEMs are only created by opening a DEM file type. If neither a TIN nor a DEM exists, then a **Reprojection** can be performed 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.



TIN Object Right-Click Menu

Right-clicking on a TIN allows selecting any of the TIN processing or visualization options in WMS. The commands in this menu are as follows:

•**Trim** : Opens the *Polygon Selection Options* dialog.

•**Merge** : Opens the *Merge TINs* dialog.

•**Filter** : Brings up the *Maximum filter angle* dialog.

•**Fill** :

•*Data* : A submenu for adjusting datasets in the TIN. It contains the following commands:

- **Calculator** : Launches the *Data Calculator* dialog for modifying or creating datasets in the TIN.
- **Smooth Dataset** : Rounds the data values and creates a new dataset under the TIN with the smoothed values .
- **Map Elevation**
- **Film Loop** : Starts the *Film Loop Setup* dialog for creating a animation from the TIN.

•*Vertices* : A submenu for adjusting vertices in the TIN. It contains the following commands:

- **Lock/Unlock**

- **Delete Duplicates**
- **Smooth Pits**
- **Transform**

•*Triangles* : A submenu for triangulatin options in the TIN. It contains the following commands:

- **Triangulate**
- **Insert Breaklines**
- **Optimization Triangulation**
- **Check Long/Thin Triangles**
- **Remove Flat Triangles**

•*Interpolate* : A submenu for interpolating the TIN data to another module or TIN. It contains the following commands:

- **To 2D Grid**
- **To TIN**
- **To Feature Objects**

•*Convert* : A submenu for converting the TIN data to another module or DEM. It contains the following commands:

- **TIN Contours→Feature**
- **TIN Boundary→Feature**
- **TIN→2D Scatter Points**
- **TIN→DEM**

•*Projection* : A submenu for changing the coordinate system of the TIN. It contains the following commands:

- **Projection**
- **Set as Display Projection**
- **Reproject**
- **Transform**

•**Options** : Brings up the *TIN Options* dialog.

•**Import Scalar Dataset** : Imports a scalar dataset by first selecting the type of scalar dataset to import and then launching the *Open* dialog.

•**Save As** : Used to designate the path and prefix file name for saving a TIN.

•**Properties** : Brings up the *Properties* window for the TIN.

•**Display Options** : Brings up the *Display Options* dialog.

•**Delete** : Removes the TIN from the project.

•**Duplicate** : Creates a copy of the TIN.

•**Rename** : Lets the name of the TIN be changed in the Project Explorer.

•**Zoom To Layer** : Will resize the TIN data so it fills the Graphics Window.

- **Metadata** : Brings up a *Metadata* dialog.
- **Generate Metadata XML From XMDF**
- **Compute Area Between Elevations**

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 .

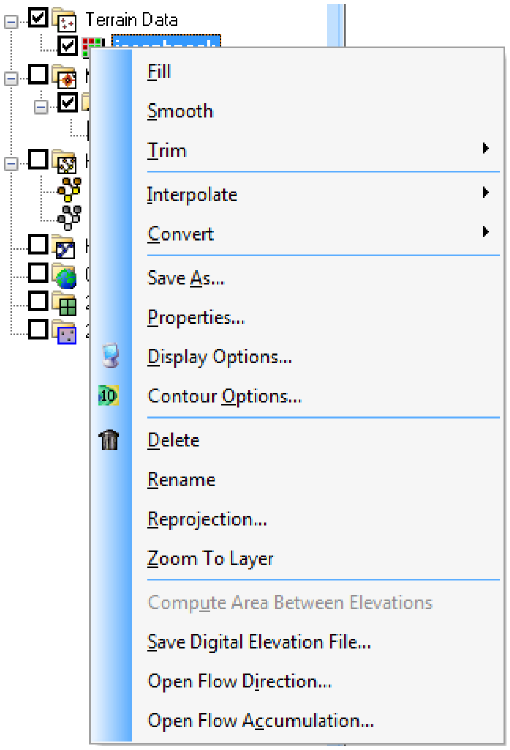
TIN Dataset Righ-Click Menu

Right-clicking on a dataset of a TIN allows the following options:

- **Rename** : Changes the name of the dataset in the Project Explorer.
- **Export Dataset** : Selects the file format and save the dataset.
- **Delete** : Removes the dataset from the TIN.
- **Properties** : Brings up the *Properties* window for the dataset.
- **View Values** : Opens the *Edit Dataset Values* dialog.
- **Contour Options** : Brings up the *Contour Options* dialog.
- **Set as Elevations** : Assigns the dataset as the elevation dataset for the TIN.

DEM Object Right-Click Menu

Right-clicking on a DEM allows selecting 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

5.1.1. 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.

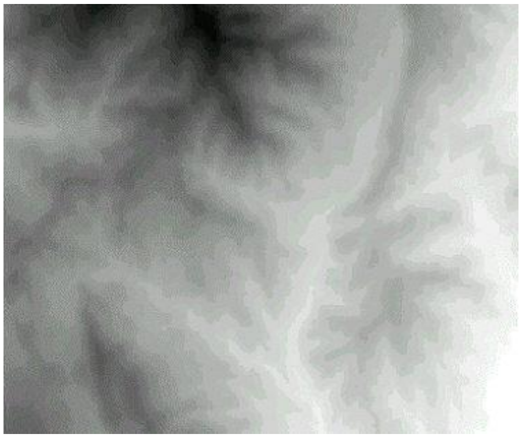
The most common form of DEM elevations are the USGS digital maps. DEMs can be downloaded free of charge from the EROS home page at http://edcwww.cr.usgs.gov/doc/edchome/ndcdb/ndcdb.html . Other sources of elevation data may include federal, state, and local government agencies, universities, or private 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:

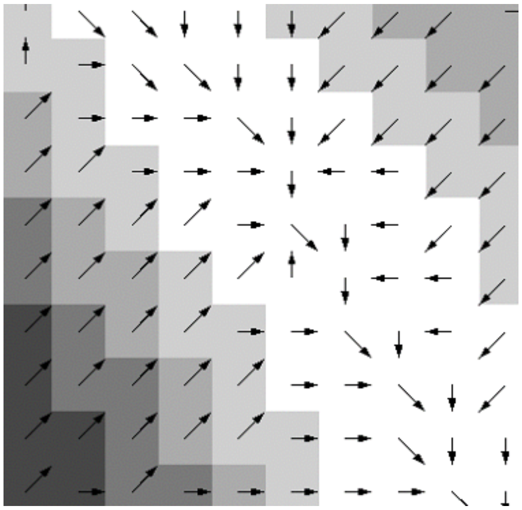
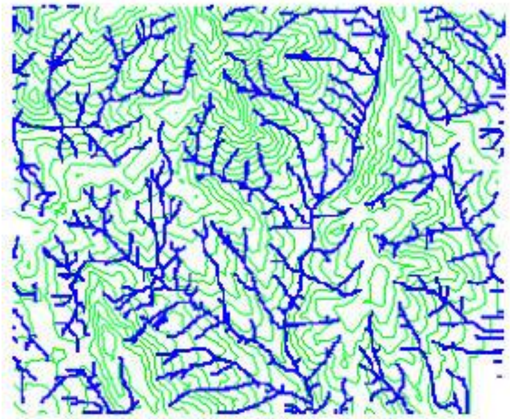
1. Obtain and Import a Digital Elevation Model (DEM)

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.



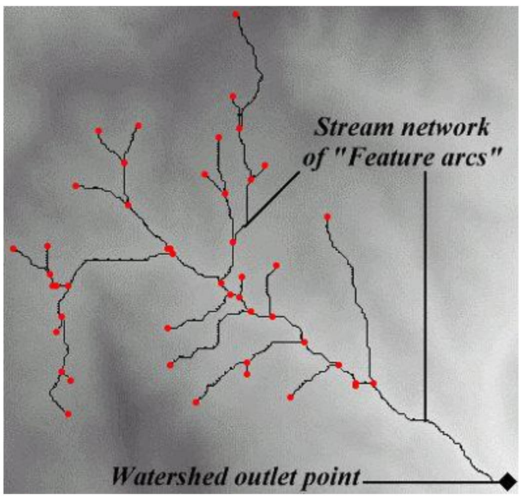
2. Compute Flow Directions and Flow Accumulations

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

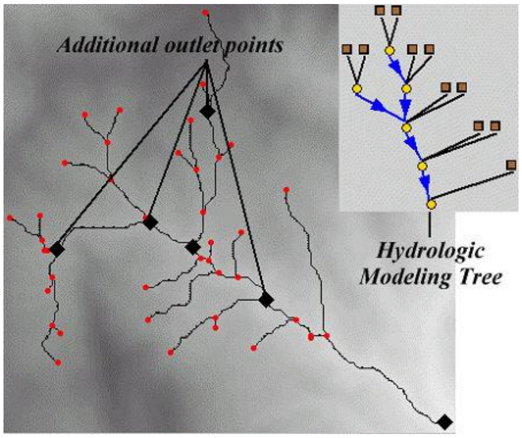
With the aid of the flow accumulations, the location of the watershed outlet needs to be determined and an outlet feature point created there. A minimum threshold is then defined and all of the DEM points "upstream" from the defined outlet(s) are connected together to form a stream network of feature arcs .



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, think of WMS modifying the flow directions of the DEM points underlying the stream vectors so that flow always follows the defined stream vectors.

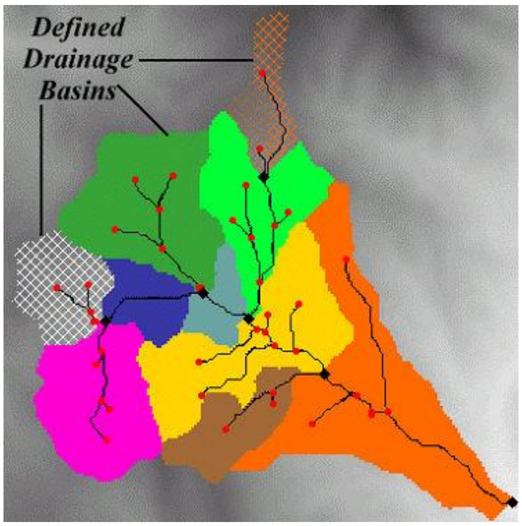
4. Define Interior Sub-basin Outlet Points

If wanting 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.



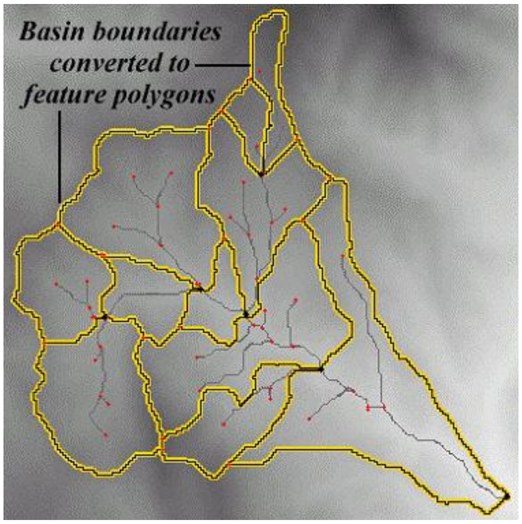
5. Define Basins

Using the outlets on the stream network and the flow directions, the contributing DEM points for each outlet are assigned the proper basin ID.



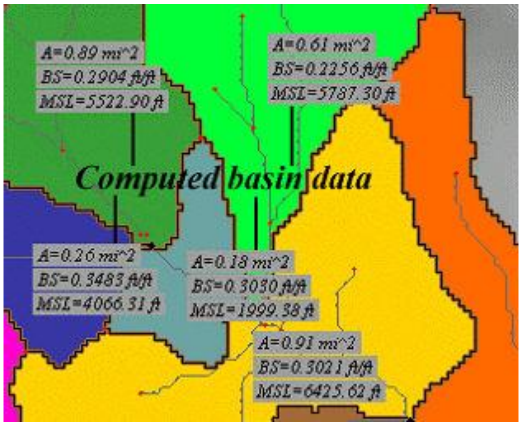
6. Convert DEM Basins to Polygons

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, there should exist 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

DEM Menu

The *DEM* menu contains the following options:

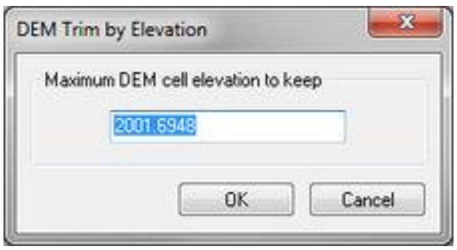
•**Delete** – Removes all DEM data

•**Fill**

•**Smooth** – Brings up the *DEM Smoothing Options* dialog. See Smoothing DEMs for more information.

•**Trim** – See Editing DEMs for more information on the *Trim* commands.

- **Polygon**
- **Elevation** –



•**Edit Elevations** – See Editing DEM Elevations for more information.

•**Point Attributes** – Brings up the *DEM Point Attributes* dialog. See DEM Point Attributes for more information.

•**Interpolate**

- **...to 2D Grid** – 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.
- **...to TIN** – 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 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.
- **...to Feature Objects**

•**Conversion** – Options for DEM conversion include:

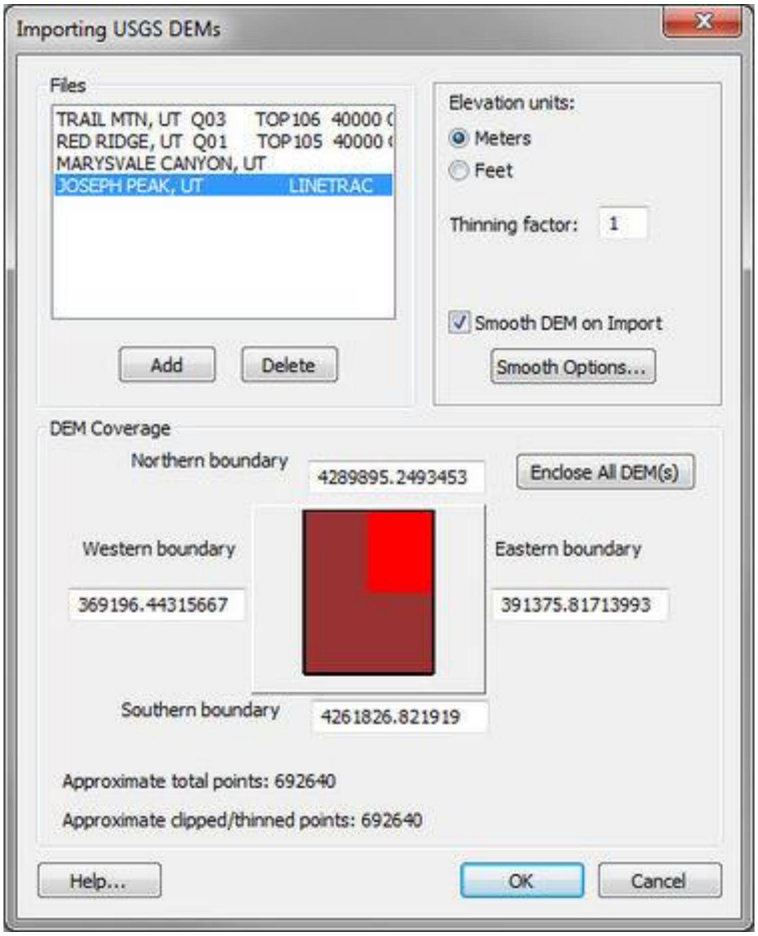
- **DEM Contours → Feature** – See DEM Contours to Feature Objects for more information.
- **DEM → TIN** – See Converting DEMs to TINs for more information.

Related Topics

•DEM Guidelines

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 to be read. It is possible to 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. The standard file opening dialog appears where DEM files can be selected. 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, click near one of the four edges of the bounding rectangle and drag it to a new location. If clicking near a corner, both edges will be adjusted. If clicking in the center of the rectangle, then the entire rectangle can be translated to a new location. When dragging 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. 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 the system is capable of reading. For example, each DEM point requires 5 bytes of memory, so that if reading 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 the system 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

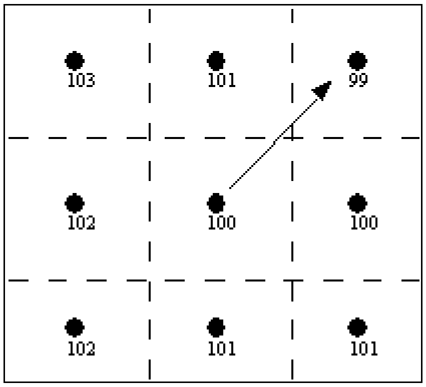
•Finding Data for use in WMS

•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.



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 being used to create flow directions, import them using the **Import Flow Directions/Accumulations** command. If using 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 the 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 creating 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, redistribute vertices along a cubic spline. If using a 30 meter resolution DEM, the average length between vertices will be approximately 30 meters and it is suggested to redistribute to about 100 meters spacing (be sure to turn on the cubic spline option). In general, redistributing to about three 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 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

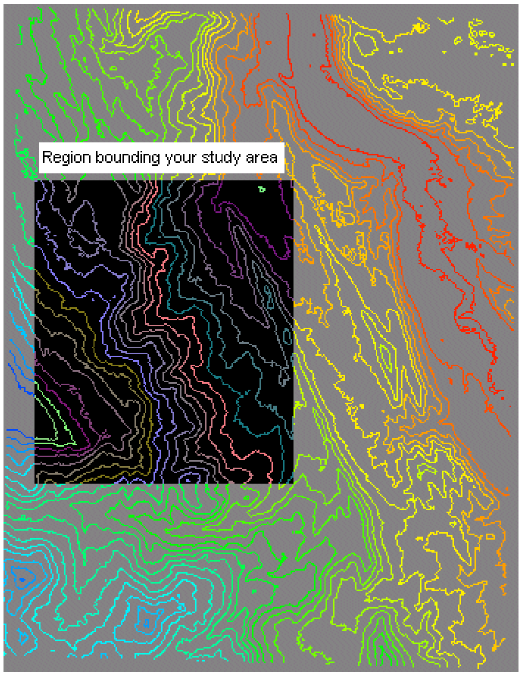
- Creating TINs from Feature Objects
- Creating 2-D Grids

Editing DEMs

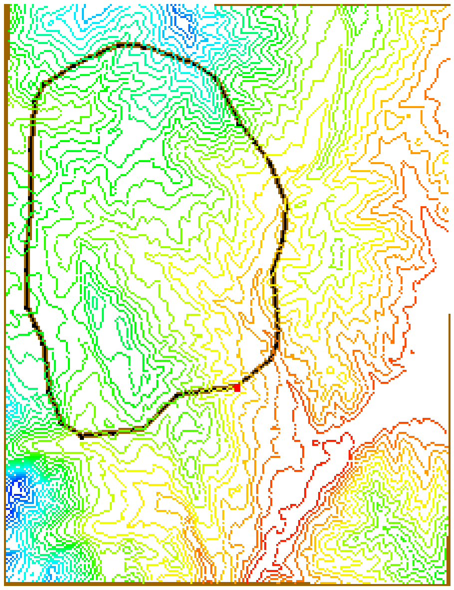
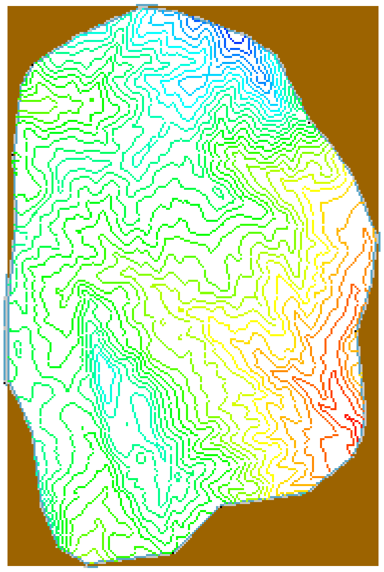
Trimming DEMs

The **Trim** command is used to set the area of DEM that will be worked with. A DEM by definition includes a rectangular area, but some 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, 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

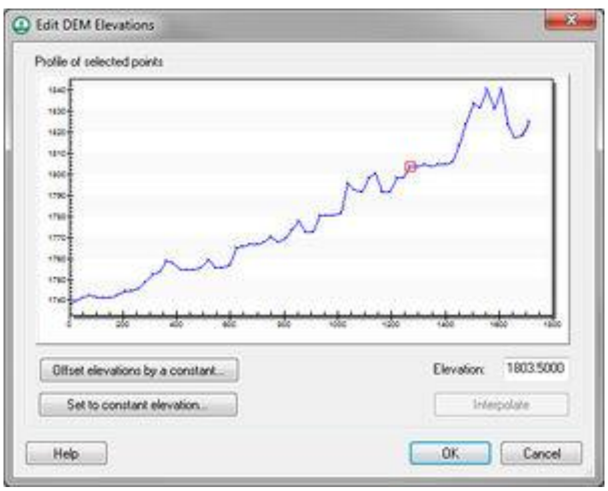
While DEMs are simple to use and more commonly available than TIN data, they are more difficult to edit or adjust, primarily because of the number of elevation points that must be changed in order to have a corresponding effect on the drainage characteristics of a watershed. However, there are a few things that can be done, and if kept to a reasonable amount can be powerful in using a DEM for watershed delineation, but remember that data can not be manufactured. If making numerous unexpected edits, then consider 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

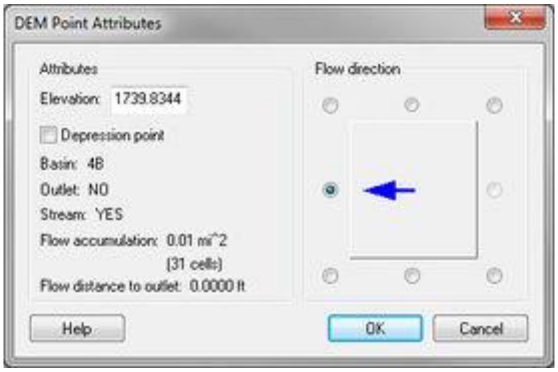
The **Edit Elevations** command in the *DEM* menu is used in conjunction with a selected arc to adjust the elevations of all DEM cells that lie beneath the selected arc. Double-clicking on an arc while in the Terrain Data module will also activate the *Edit DEM Elevations* dialog.



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:

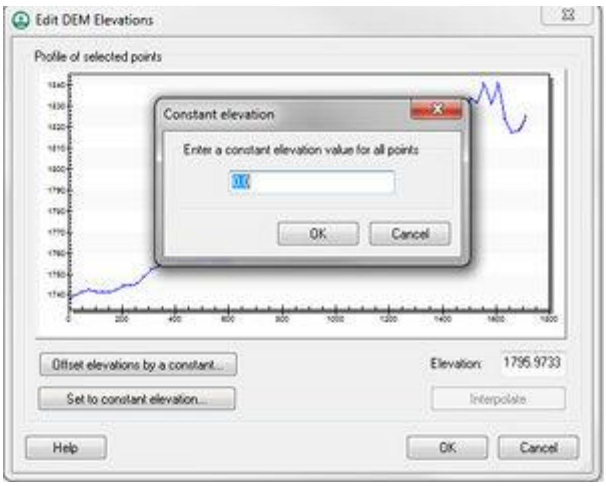
Select individual DEM points and adjust the elevation

Using the **Select DEM Point** tool in the *Edit DEM Elevations* dialog, select a point along the arc and edit the elevation.



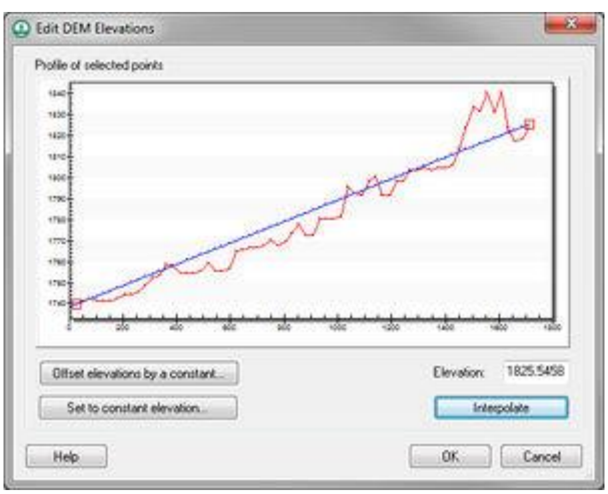
Set/Offset all elevations by a given value

It is possible to 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

It is possible to linearly interpolate all elevations between the first and last elevation. Some 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, 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 not wanting WMS 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 *DEM 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.

Memory

Memory required to store current elevations:

- Store current elevations

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 there is already computed flow directions and accumulations with TOPAZ and editing the elevation, re-run TOPAZ. Editing elevations one at a time can be very tedious and should only be used for small local changes. If desired, edit elevations using feature arcs if wanting 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 using 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 only do so with care and for limited areas. WMS will check to make sure that when changing the flow direction a circular path is not created (the flow ends up back in the cell that is being edited). If it does, a prompt will inform that it is not possible and that direction will become dimmed.

After editing a flow direction and selecting **OK**, a prompt will apear to recompute flow accumulations, and if basins have been delineated to recompute basin data. This only needs to be done after having finished all edits. If planning 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 information only and 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, 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.

Related Topics

•DEM Guidelines

Converting DEMs

Converting DEMs to TINs

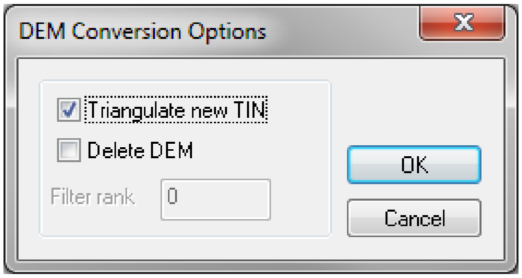
All DEM Points

Using the *Conversion* | **DEM→TIN** command from the *DEM* menu will convert all of the 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

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 modify the results of a delineation by creating user-defined 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. It is possible to 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.

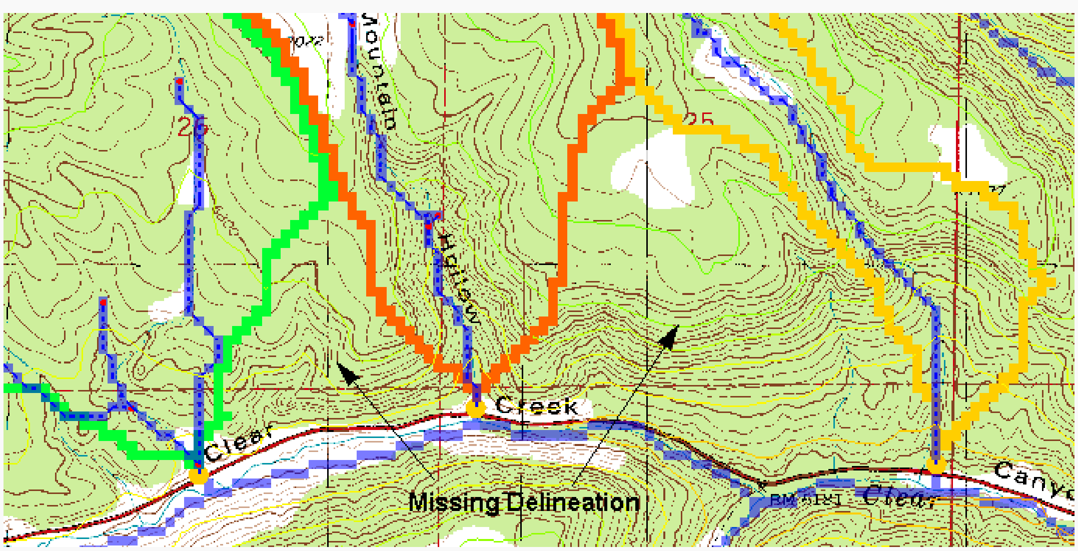


Figure 1: Delineation from the TOPAZ results only (not including the roads as streams)

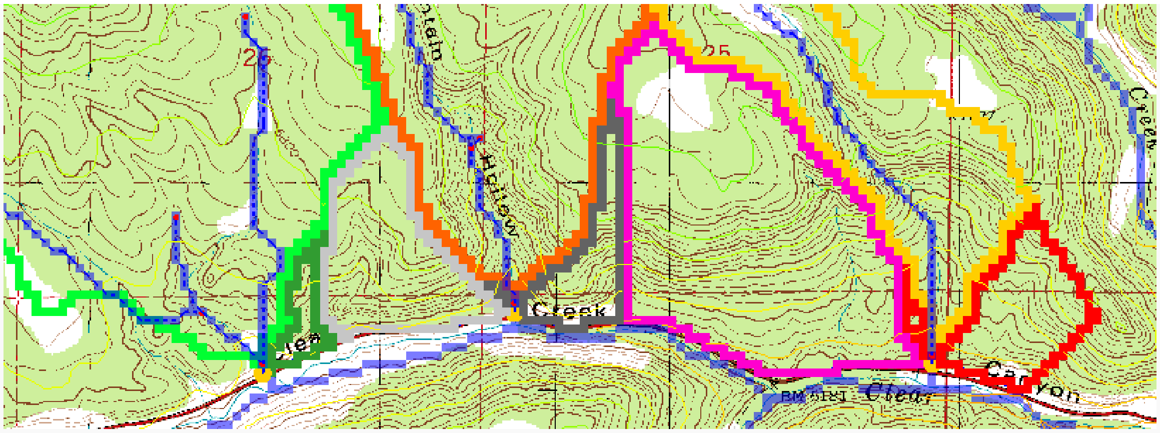


Figure 2: Delineation after including the roads as streams. It is important to note that streams have been made in both directions from the outlets in order to "capture" the area that will flow to each outlet.

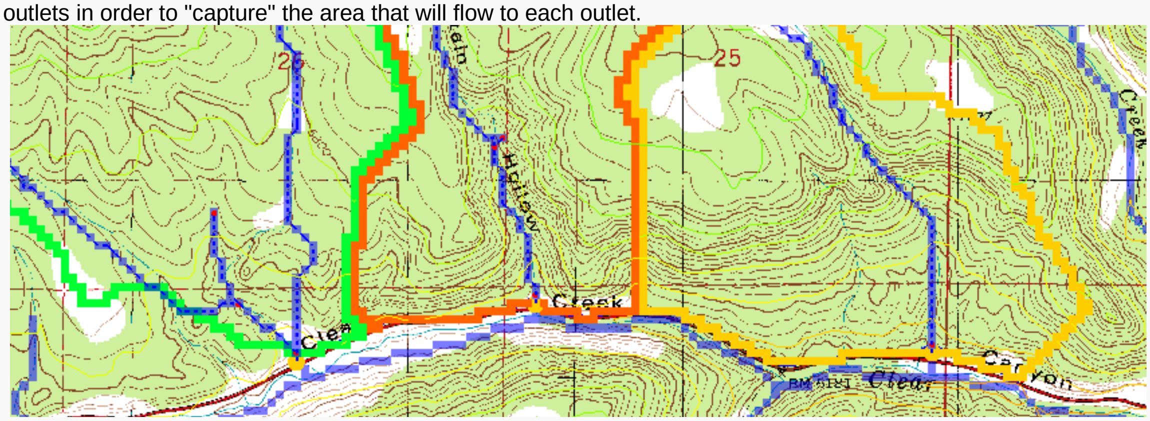


Figure 3: If leaving the outlet of the basin at the bottom node, WMS will create separate basins for the main branch and each branch along the road. In order to combine them into a single basin a small (about one cell in length or less) stub stream is created so that only a single drainage will be used. Remember that WMS will create a separate basin for each upstream branch from an outlet point.

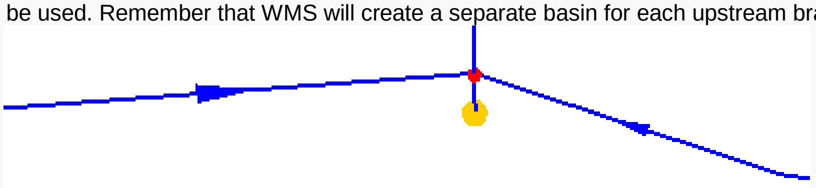


Figure 4: Close up of an outlet so that a single basin is defined.

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, redistribute vertices along a cubic spline. If using a 30 meter resolution DEM then the average length between vertices will be approximately 30 meters and it is suggested to redistribute to about 100 meters spacing (be sure to turn on the cubic spline option). In general, redistributing to about three times the DEM resolution will produce good results.

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 a watershed has been derived from feature objects, and also has a background DEM, 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, 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.

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 view view or edit any of the basin variables that are computed by WMS . While it is unnecessary to edit these variables, in some cases it may be desired to override what WMS has computed and use a value 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, some may wish to combine basins of all branches of an outlet into a single basin. The **Merge Selected Basins** command allows doing 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.

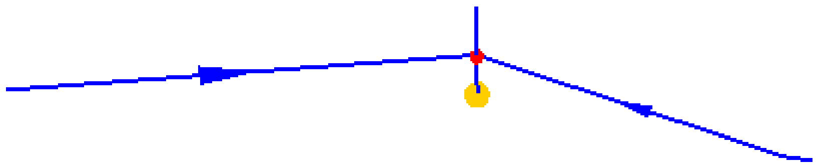


Figure 5: Define basin just downstream of junction

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 not delineated the basin using the flow directions and flow accumulations from within WMS, but instead using 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.

Only use this option if there is a set of feature objects already and/or wishing 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. Rather than needing to perform them one at a time, select the **Delineate Basins Wizard** command and the steps will be performed in succession.

In order to run the **Delineate Basin Wizard** , the project should have already read in elevations (edited if desired), run TOPAZ to compute the flow directions and flow accumulations, and established 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 there is 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 the polygon is where it should be, switch to the Drainage Module then select *DEM* | **Polygon Basin IDs→DEM.** This should redefine the basin where the polygon was.

Related Topics

- Watershed Delineation with DEMs
- Importing Flow Directions and Accumulations
- Node to Outlet
- DEM → Stream Arcs

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 interested in obtaining the latest, complete version of TOPAZ, visit the United States Department of Agriculture's (USDA) Agricultural Research Service website (here]) and click on TOPAZ.

After obtaining 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, specify the directory where WMS/TOPAZ perform this interaction. If the specified directory has an unusually long name TOPAZ may have some difficulties. In such cases, 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 having identified 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, it is possible to 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

5.1.2. Triangulated Irregular Networks(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.

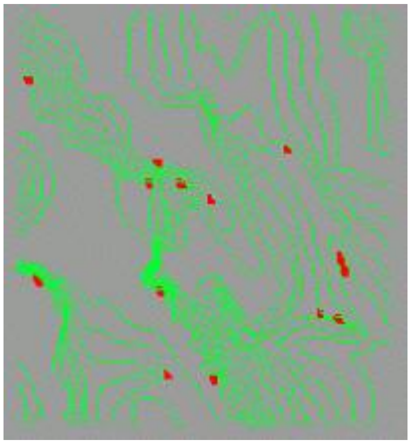
Developing watersheds from TINs often involves the use of both feature objects and DEMs. The following steps can be used as a guideline for watershed characterization with TINs.

1. Obtain Background Elevation

An elevation source is required for creating a TIN. Even if there is already a TIN data source, it's recommend that in most cases to 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, 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

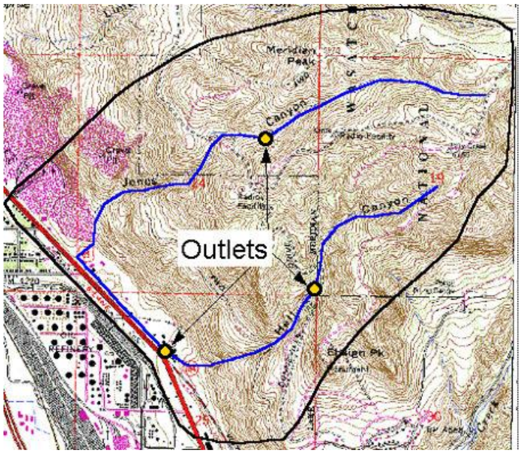
TIN creation builds on the previous four steps. The outer polygon is used to define the limits or extents of the TIN. TIN vertices are created inside this polygon at a density proportional to the spacing of vertices on the nearest arc. After the TIN is created, the stream (and other interior) arcs are 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.

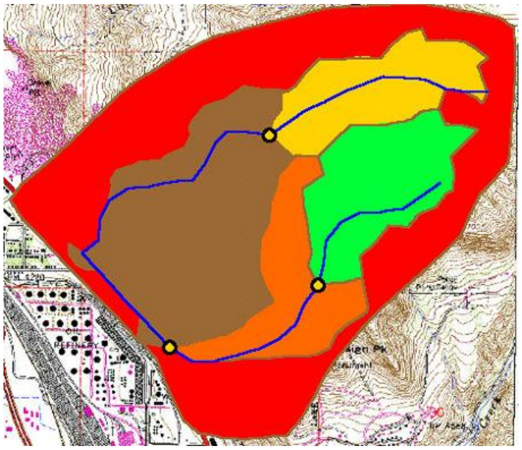
7. Complete Stream Network and Outlet Definitions

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 it is possible to return to this step and redefine the locations where sub-basins are to be 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 correcting these problems automatically and manually. The automatic method will correct the problem about 90% of the time, while in 10% of the cases it may be necessary 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 having corrected them, refine the 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 satisfied with the 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, several important geometric parameters can be computed . 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

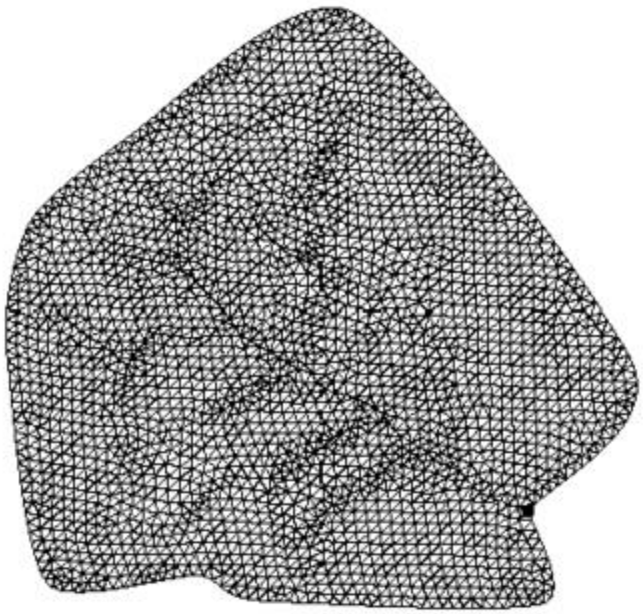
Along with the watershed definition on the TIN, an accompanying topologic model is created. Then interact with the TIN or tree representation of the watershed to complete input for and run the supported hydrologic models .

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, it is possible to 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.

Some may wonder why, if there is already TIN data, to 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 simply triangulating a set of xyz scatter points, or importing 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 . (If desired, mark the map with the 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 to finish.

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 Vertices

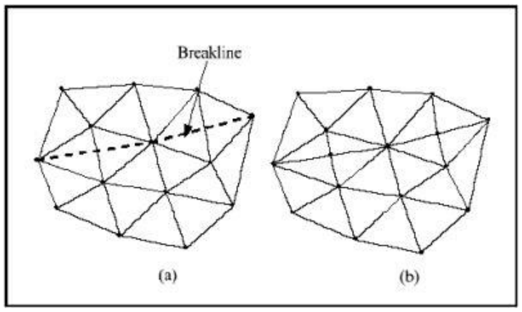
TIN Breaklines

A breakline is a feature line or polyline representing a stream channel, ridge or some other feature 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 specifying either to 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, a prompt will ask if wanting 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 datasets, it is difficult to get a continuous set of stream bed elevations. Within this dialog, select and edit the elevation of individual stream nodes or can select two different stream nodes from the profile plot and linearly interpolate the elevation of all nodes in between. In this way it is possible to smooth out pits in the stream that may exist only because of a lack of resolution.

When using this command, It's necessary to 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 an alternative method to merge basins is needed, there is an option to merge selected basins together using the **Merge Basins** command. The **Merge Basins** command allows combining 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 viewing/editing any of the basin variables that are computed by WMS . While it is unnecessary to edit these variables, some may find some cases where overriding what WMS has computed is desired so as to use a value derived through a separate analysis.

Assigning Triangles to Basins

Occasionally it is desirable to manually assign triangles to a basin. This is particularly important if there is a large flat area (such as a lake) within a basin and there is no need to edit the elevations such that they would flow to the outlet.Select the triangles to assign and then choose the **Assign Triangles to Basin** command in order to assign the triangle to a basin. A prompt will appear to select the outlet of the basin 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 which allows specifying the current Units of the model (TIN vertices) and the units 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 wanting to change the units of computed parameters at a later point, recompute the basin parameters with different selections. It may be desirable 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

The *TIN* menu has the following commands:

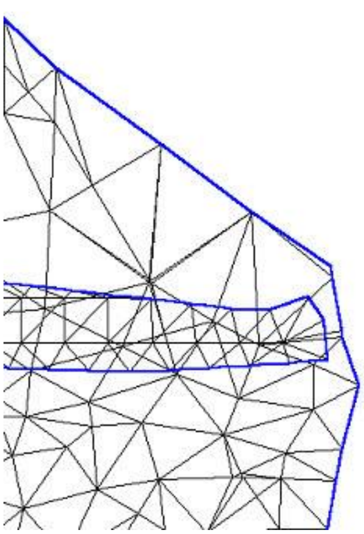
TIN Options

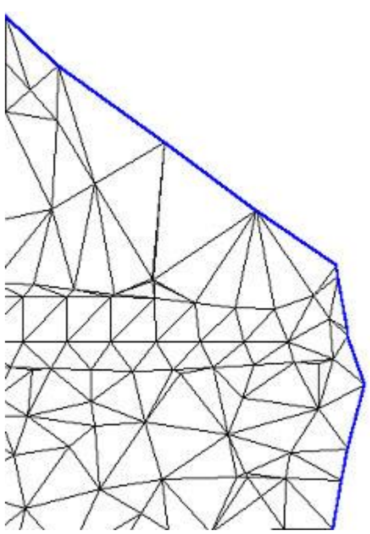
Brings up the *TIN Options* dialog. See the article TIN Options for more information.

Merging TINs

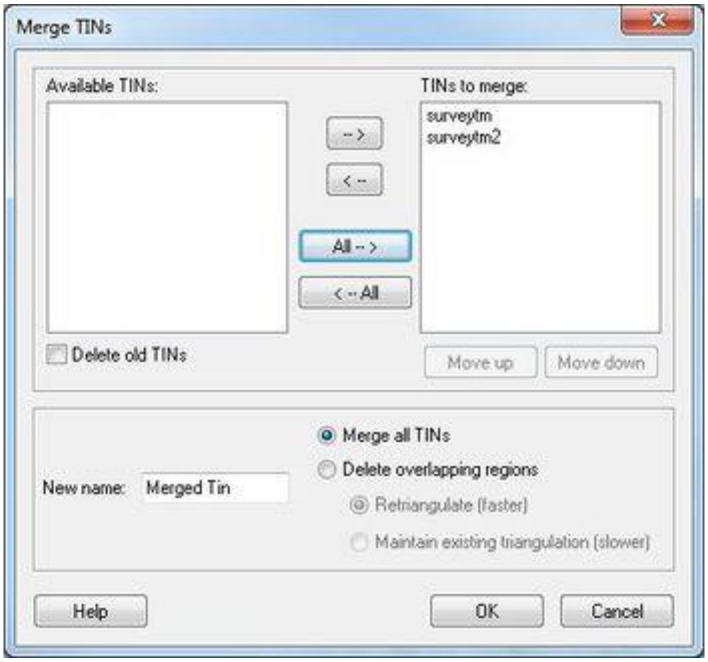
It is possible to merge a selected TIN with another TIN using the **Merge** command in the **TIN** menu. This is particularly useful if wanting to merge a TIN generated from one program with a TIN derived from a background elevation source such as a DEM.

For example, a project may have surveyed data with a lot of detail for a part of the project (such as a roadway profile) and it is desired 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 specified to have vertices in overlapping regions deleted. The old TINs can be kept or deleted.



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.

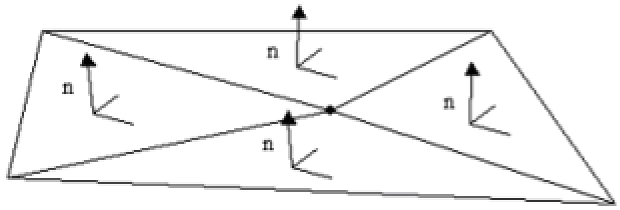
Fill

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 to retriangulate to a higher density.

It's necessary to 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.

Filter

Redundant and overlapping data may exist in a scattered XYZ datasets. 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. This brings up the *Maximum Filter Angle* dialog where an angle is specified. 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 a 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.)

Trimming TINs

Trimming allows eliminating all vertices that are outside of a selected feature polygon. First create the feature polygon in the Map module, then select the polygon 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 it's necessary to add new vertices, adjust elevations, swap multiple edges or some other form of manual editing technique.

Data

The *TIN* menu offers four options for the using data:

- **Calculator** – Brings up the *Data Calculator* . See Datasets for more information.
- **Smooth Dataset** – See Smooth Dataset for more information.
- **Map Elevation** – See the Mapping Elevations section of the Datasets article.
- **Film Loop** – See Setting up Film Loops for more information.

Vertices

The *Tin* menu has following options for vertices:

- **Transform** – Used to move TIN vertices according to a specified transformation.
- **Lock/Unlock** – Selected vertices can be locked to prevent them from being dragged or edited.
- **Delete Duplication** – Removes vertices that duplicate the same xy location as another vertex.

•**Smooth Pits** – Adjusts the elevations of pits in order to remove them.

See Vertices for more information.

Triangles

The *TIN* menu gives the following options for triangles:

•Triangulate

•Insert Breakline

•Optimize Triangulation

•Check Long/Thin Triangles

•Remove Flat Triangles

See the article Triangulation for more information.

TIN Interpolation

Interpolate TIN To TIN

The **Interpolate to TIN** command of the *TIN* menu allows interpolating the elevations of the active TIN to one or more TINs. If more than one other TIN exists, a dialog will appear that allows selecting the TIN(s) 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.

Convert

The following conversion options are available for TINs:

•TIN Contours → Feature

•TIN Boundary → Feature

•TIN → Scatter Points

•TIN → DEM

See Convert TINs for more information.

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

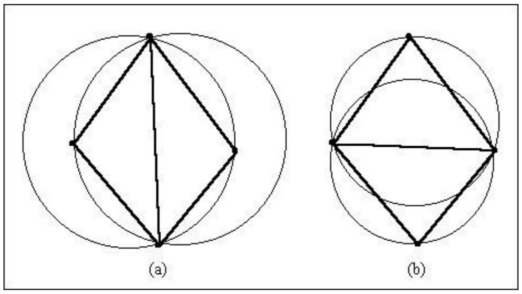
•DEM Contours to Feature Objects

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 Delaunay criterion The Delaunay 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 Delaunay 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 Delaunay 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 Delaunay criterion. If the Delaunay 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 beginning with TIN data, 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 Delaunay 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, 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, it is possible to distinguish the new vertices from the original by observing the ones that are unlocked. It may be desirable 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 *CTR* L 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.

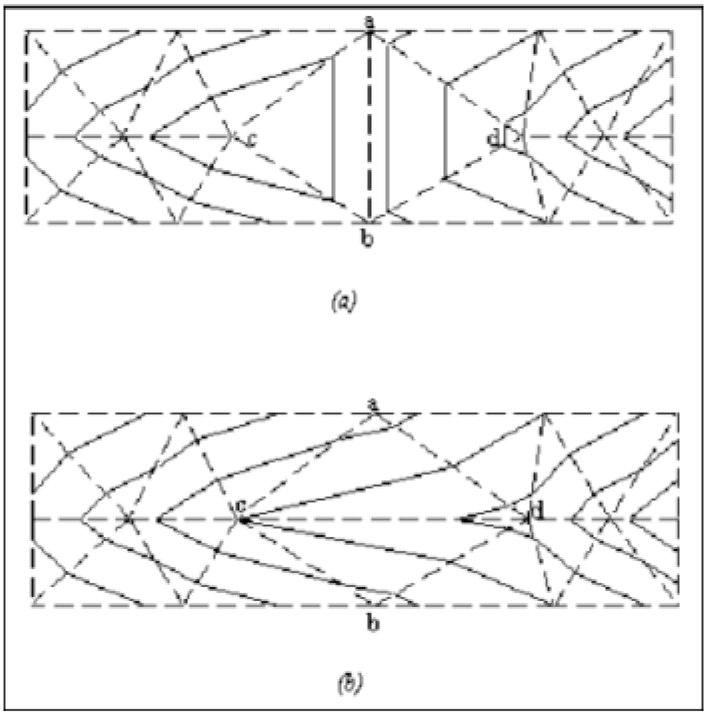
Thin Triangle Aspect Ratio

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 Delaunay 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 to prepare the TIN for drainage analysis by using feature objects with a background elevation source. The elevation source could be TIN or DEM, but retriangulate using feature objects as guides in order to insure that stream edges are honored by triangle edges.

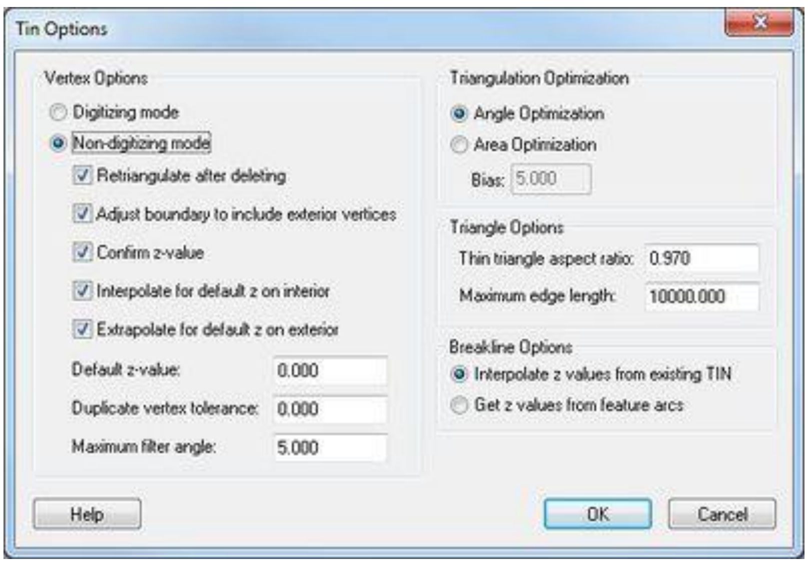
Detailed steps for delineating watersheds using TINs can be found on this page. The first process in performing drainage analysis is to edit the TIN where necessary. Flat triangles, flat channel edges, and flat ridge edges must all be eliminated before trying to delineate stream networks and basin boundaries. 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

- Guidelines for Watershed Delineation with TINs
- 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. Normally is is not necessary 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

The *Vertices* menu has the following commands:

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, a prompt will ask 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 needing 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:

$$newz = WSE - oldz$$

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, 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 it is desired to smooth elevation data or another dataset associated with a TIN. This may be because of a need to remove flat areas or other anomalies in the elevation dataset. When smoothing 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 the TIN.

Related Topics

•Smoothing DEMs

•Smoothing Pits

5.2. 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, consider modifying an initial delineation with an elevation source manually with the feature objects. However, understand 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 modifications are being made.

The Drainage module is included will all paid editions of WMS.

Related Topics

•Watershed Delineation with DEMs

•Watershed Delineation with TINs

•Watershed Delienation with Feature Objects

•Drainage Coverage

•Drainage Data to Feature Objects

•Drainage Module Feature Objects

•Drainage Tools

•Project Explorer Contents for Drainage Module

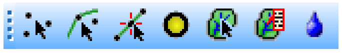
Drainage Data to Feature Objects

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



The toolbar for the WMS Drainage module has a variety of tools useful for editing and creating objects within that module. The tools are described below.

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 Feature Point/Node

The **Select Feature Point/Node**  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* .

Create Outlet Point

The **Create Outlet Point** tool is used to create an outlet point for a drainage basin or drainage unit.

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 Label

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 Label** 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.

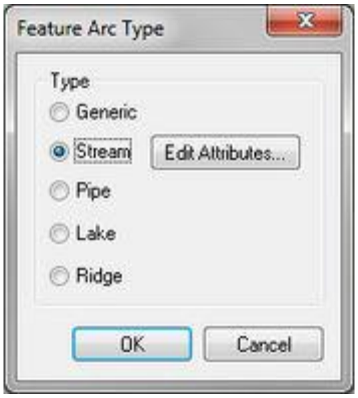
The length and slope of overland and stream flow is displayed in the help window each time a new path is drawn. This can be helpful in obtaining parameters used to compute lag times with some empirical formulas. Stream distances are shown only after a stream has been created. In other words, channel 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.

Drainage Module Feature Objects

The drainage module has a number of tools to edit feature objects. The dialogs can be reached by double-clicking on feature object or by selecting the feature object then selecting **Attributes** from the *Feature Objects* Menu.

Feature Arc Type



This dialog is accessed by selecting a feature arc. The type of arc can be selected from this dialog. Available types are:

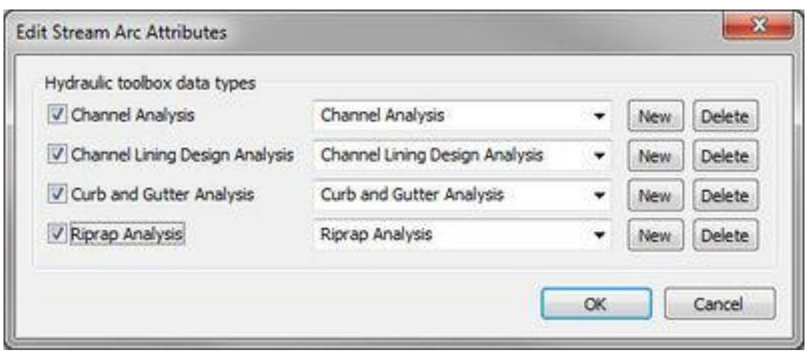
•*Generic* – Have no attributes and are used when developing drainage boundaries or when establishing the boundary polygon for creating a TIN from feature objects.

•*Stream* – Used to define stream reaches hydrologic models.

•*Pipe*

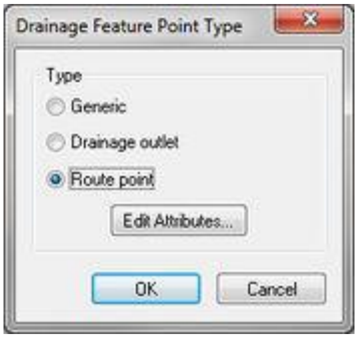
•*Lake* – Used to trace around the boundary of a lake.

•*Ridge* – Used when creating a TIN from feature objects and are used to designate any other (besides boundary, stream, and lake) linear segment to be enforced as a breakline in the resulting TIN.



Selecting the *Stream* option makes the **Edit Attributes** button active. Clicking on this button will bring up the *Edit Stream Arc Attributes* dialog.

Feature Point Type

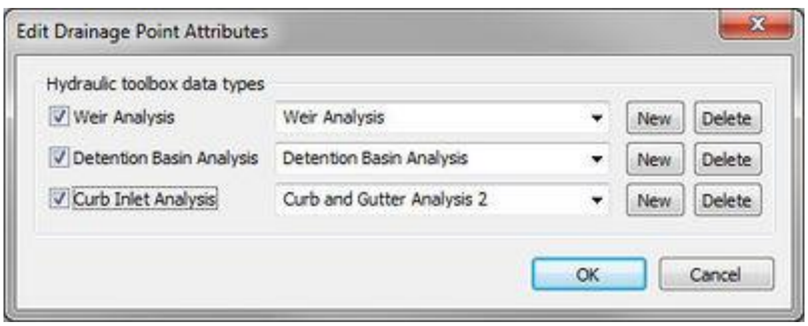


This dialog allows selecting the point type. It is accessed by selecting a feature node point. Available types in the drainage module are:

•*Generic* – Have no attributes and can be used for drawing purposes.

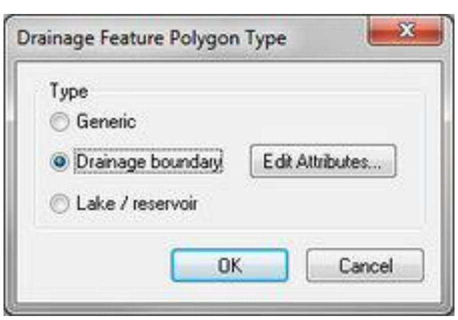
•*Drainage outlet* – Directly linked to the hydrologic modeling tree.

•*Route point*



When the *Route point* type is selected the **Edit Attributes** button becomes available. Clicking on this button will open the *Edit Drainage Point Attributes* dialog.

Feature Polygon Type



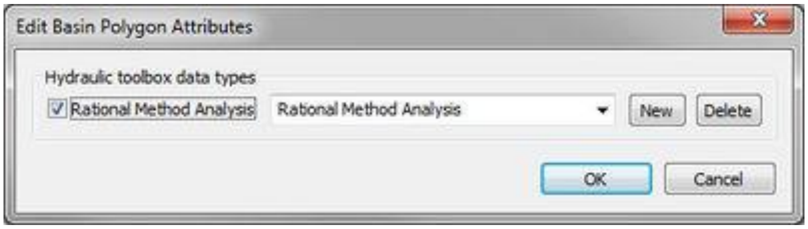
Selecting a feature polygon can bring up the *Drainage Feature Polygon Type* dialog. The dialog allows designating the polygon type. Available types in the drainage module are:

•*Generic* – Used as an intermediate polygon type when importing data from another source.

•*Drainage boundary* – Default setting for polygons in the drainage model.

•*Lake / reservoir*

Selecting the *Drainage boundary* option will make the **Edit Attributes** button active. This button will bring up the *Edit Basin Polygon Attributes* dialog.



Related Topics

- Drainage Module
- Drainage Coverage

Draw Flow Patterns in DEMs

The **Draw Flow Patterns** command (in the *DEM* menu when the Drainage module is active) 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

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. When selecting 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 creating a **New folder** or **TIN** . Because TINs can be created by digitizing in WMS, it is possible to 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 selecting 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 bring sup a menu with options 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 selecting 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.

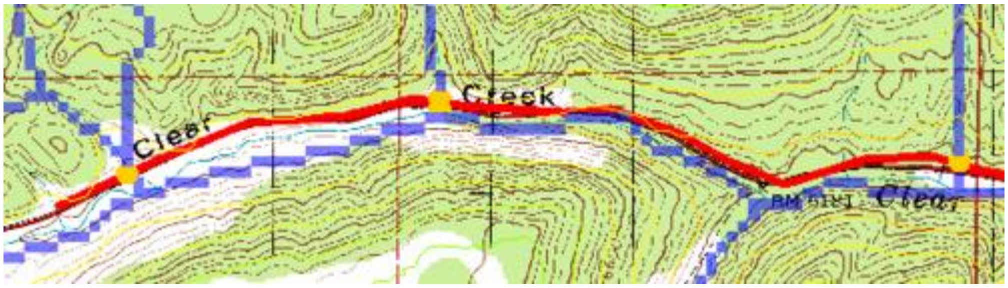
Refer to the Terrain Data Module information page for more information about managing TINs and DEMs and their attributes in WMS.

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 wanting to establish outlets for a series of drainages that a highway or other important line feature will cross as illustrated in the figure below.



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

5.3. 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.

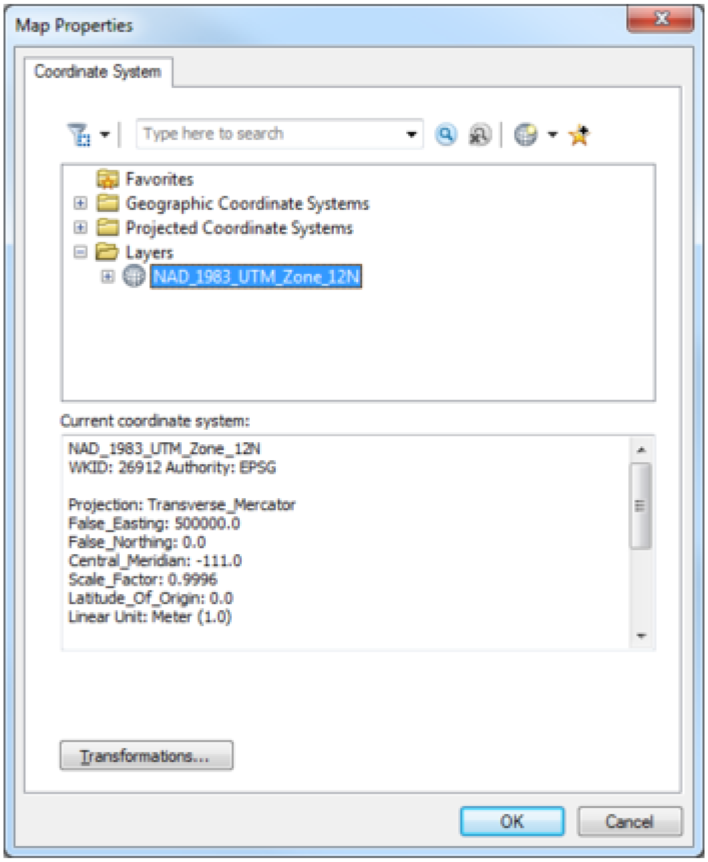
Within the Map module there are several other tools which can be helpful in either setting up models or presentation of results to a client. Tools for reading and writing of CAD files, and text annotation are part of this module.

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 to display (or 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* , specify the coordinate system to use to display features or 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* dialog.



Map Properties Dialog

The dialog is accessed through the *Data* | **Map Properties** command when ArcObjects© is active. The dialog has the following options:

•*Spacial Filter* – This drop-down menu has two commands:

- **Set Spacial Filter** – Brings up the *Spacial Filter Extent* dialog.
- **Clear Spatial Filter** – Removes anything entered int the search field and reset the coordinate system browser field.

•**Search** – It is possible to type a desired coordinate system into the search field. After pressing the **Search** button, any matching coordinate systems found will appear in the coordinate system browser field below.

•**Clear Search** – Clears the search field and resets the coordinate system browser field below.

•*Add Coordinate System* – This drop-down menu contains the following sub-menu and commands:

- *New* – This sub-menu is for commands that create new coordinate systems. It has the following options:
  - **Geographic Coordinate System** – Brings up the *New Geogrphic Coordinate System* dialog.
  - **Projected Coordinate System** – Brings up the *New Geogrphic Coordinate System* dialog.
- **Import** – Launches a browser where a coordinate system file may be imported.
- **Clear** – Removes any coordinate systems that have been added.

•**Add to Favorites** – Places the selected coordinate system in "Favorites" folder for easier access in the future.

•*Coordinate systems browser* – By opening the tree items in this field, coordinate systems can be selected to be applied to GIS data. The browser contains a right-click menu with the following commands:

- **Add to Favorites** – Places the selected coordinate system in "Favorites" folder for easier access in the future.
- **Remove from Favorites** – Deletes the selected coordinate system in "Favorites" folder.
- **Copy and Modify** – Brings up the *Projected Coordiante System Properties* dialog.
- **Save As** – Launches a browser were the selected coordinate system can be saved as PRJ file.

•*Current coordinate system* – Displays the current selected coordinate system.

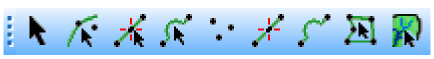
•**Transformations** – Brings up the *Geographic Coordinate System Transformations* dialog.

Related Topics

•Coordinate Systems of GIS Layers

•Mapping to Feature Objects

Map Tools



The toolbar for the WMS Map module has a variety of tools useful for editing and creating objects within that module. The tools are described below.

Select Feature Objects

The **Select Feature Objects** tool is a generic selection tool that can be used in place of many other Select tools 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 Point/Node

The **Select Feature Point/Node** 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 that it is within 10 screen pixels of an existing arc. Once the vertex is created, it can be repositioned with the **Select Feature 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 clicking 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 clicking 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 *Esc* 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, 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 user defined 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 user defined 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 the 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 the model to a schematic or tree-based representation of the model in either the hydrologic modeling or river module. When creating a conceptual model of *hydrologic* data (basins, streams, and outlet points), the schematic representation is created automatically. When 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.

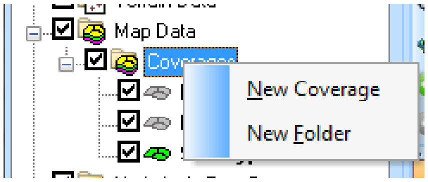
As an example, a SWMM hydraulic model with arcs representing links and nodes in a SWMM model and the symbols corresponding to links and nodes in the hydraulic schematic is shown below:

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.

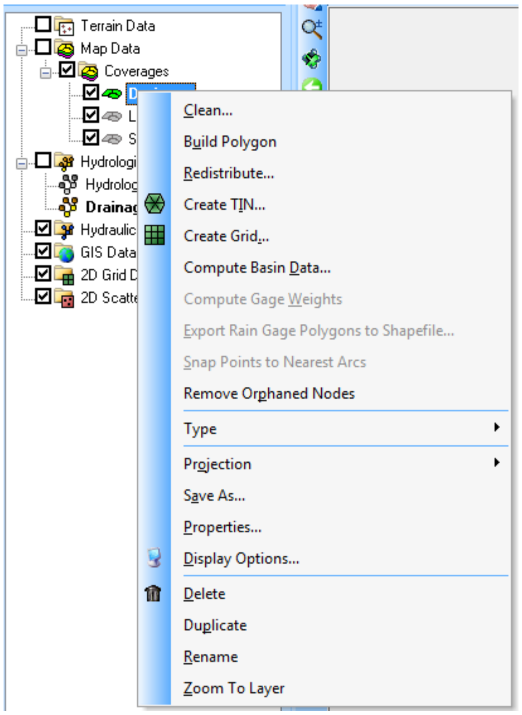
A toggle is next to each object; the toggle controls the visibility. The display is automatically updated when the toggle is checked or unchecked. The active coverage is set in the Project Explorer by selecting it. The active coverage is indicated in the Project Explorer display with a color map module icon, while the inactive coverages are gray.

Right-clicking on the Map Data folder allows creating a **New coverage** or **New folder** for organizing coverages.



The Project Explorer replaces the *Coverage Options* dialog in previous versions of WMS. See Coverages to learn about coverages in the Project Explorer.

Right-clicking on a coverage brings up a menu with options to **Delete** , **Duplicate** , **Rename** , set/view the **Properties** of the coverage, or perform a **Coordinate Conversion** . Properties include the name and coverage type.



Related Topics

•Project Explorer Overview

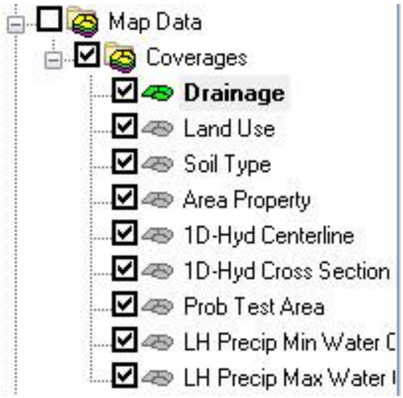
•Coordinate Conversions

5.3.a. Coverages

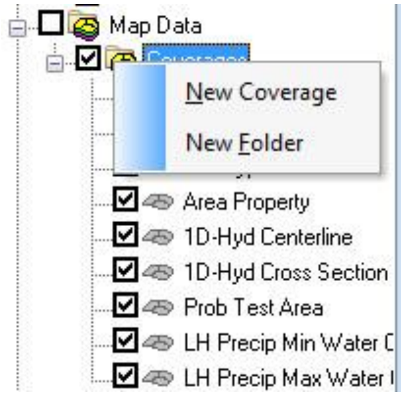
Coverages

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.

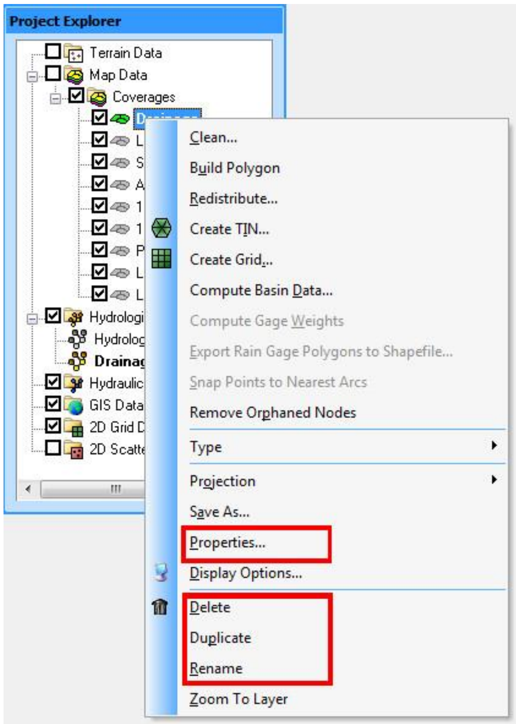


A new coverage can be created by right-clicking on a folder and selecting the **New Coverage** command in the pop-up menu.



Right-clicking on a coverage bring up a menu with the following options: **Delete** , **Duplicate** , **Rename** , and **Properties** .

The **Delete** command will remove the selected coverage. The **Duplicate** command makes a copy of the selected coverage. The **Rename** command allows changing the coverage name.



The **Properties** command brings up the *Coverage Properties* dialog. The *coverage properties* dialog allows specifying 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.

See the article Coordinate Conversions for information about the **Coordinate Conversion** command.

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

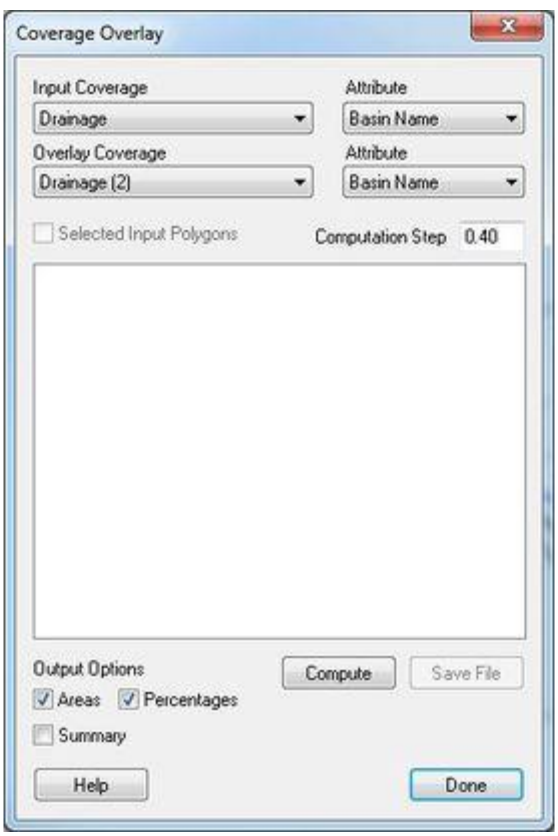
•Storm Drain

•Time Computation

Related Topics

•Feature Objects

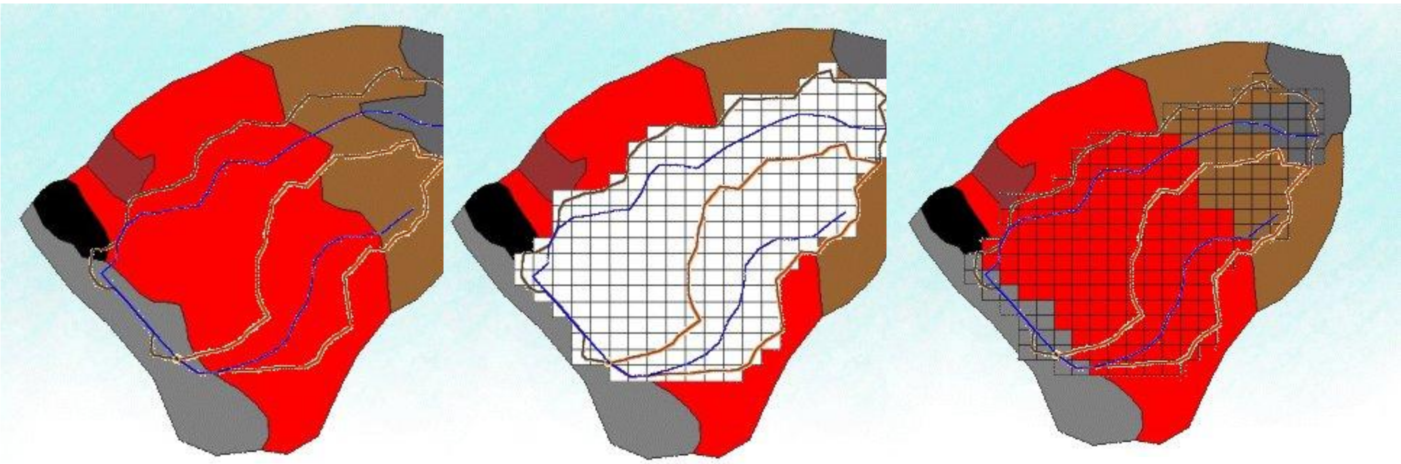
Coverage Overlays



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 a user to overlay any two polygon coverages (such as land use and drainage boundaries) and derive information from them. For example a user may want to know the percentage of a certain land use type (i.e. lakes, forest, pavement, etc.) covering the 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 being overlayed 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. The user then specify the overlay coverage. If wanting 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 a user 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

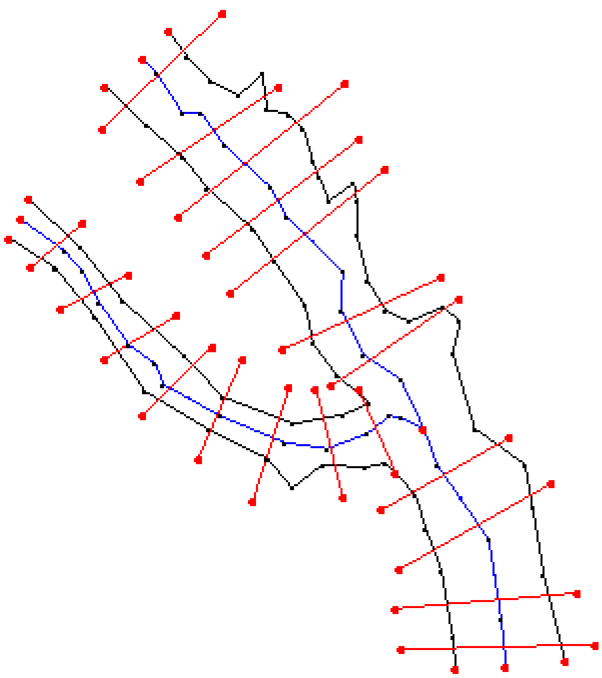
- Hydrologic/Hydraulic Calculators
- Compute GIS Attributes
- NSS

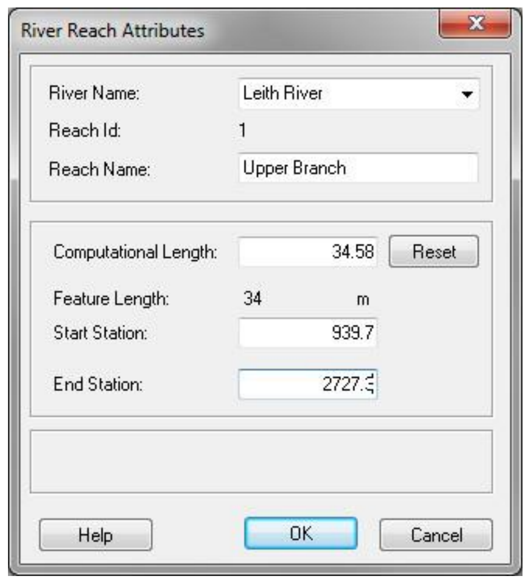
1D-HYD Centerline

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 a digital terrain model.

A centerline arc defines the stream thalweg, and provides the backbone of the hydraulic model definition. It generally follows the flow of the river, and may not actually be centered between the two defined left and right bank arcs of the floodplain or channel. Its 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 *River Reach Attributes* 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), a *Feature Length* , a *Start Station* , and *End Station* .



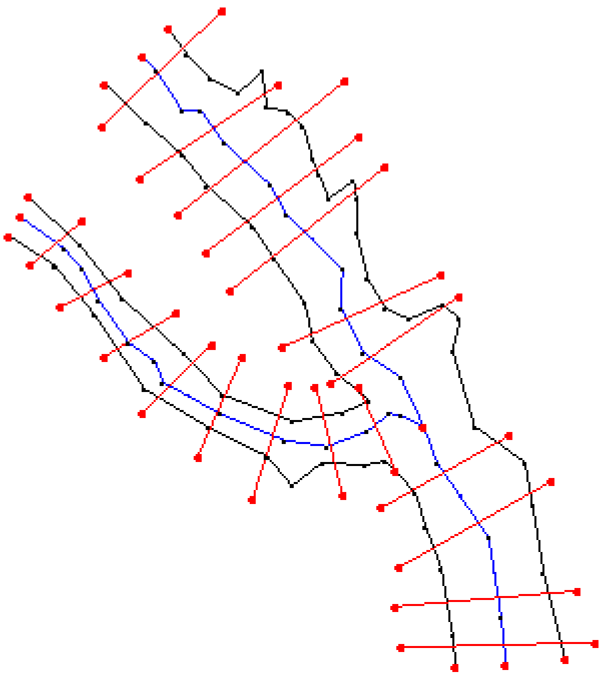


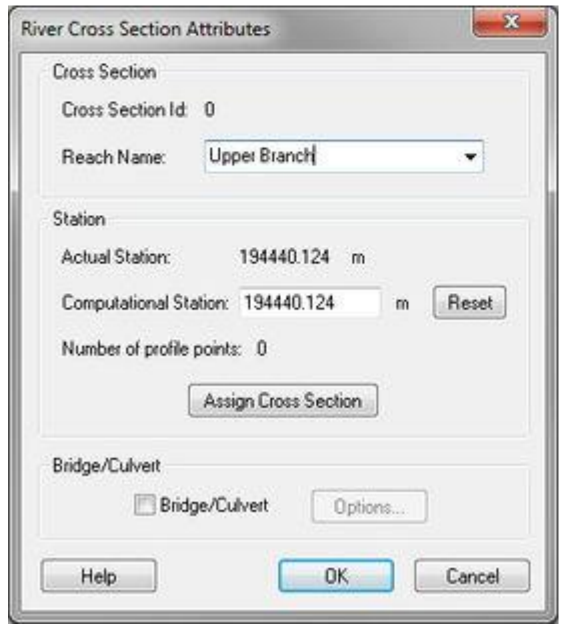
Related Topics

- [Cross Section Coverage]
- [Area Property Coverage]
- [1D Hydraulic Modeling (HEC-RAS)]

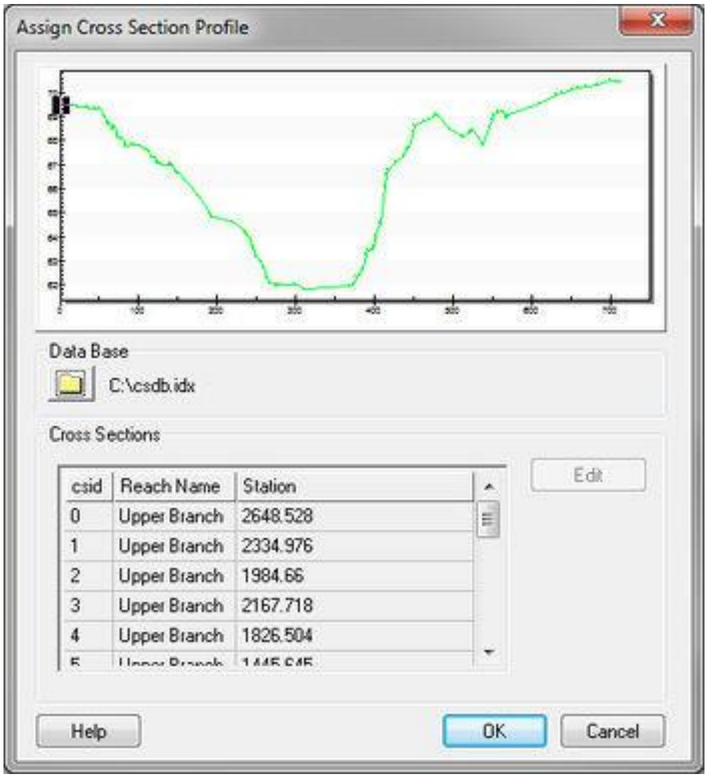
1D-HYD Cross-section

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.

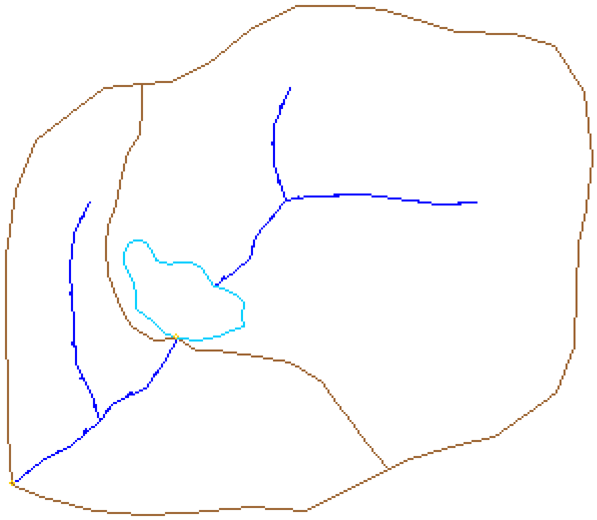
Related Topics

- [Materials]
- [1D-Hydraulic Centerline Coverage]
- [1D-Hydraulic Cross-section Coverage]
- [Cutting Cross-sections]
- [Hydraulic Modeling]
- [Export GIS File]
- [Extracting Cross Sections]

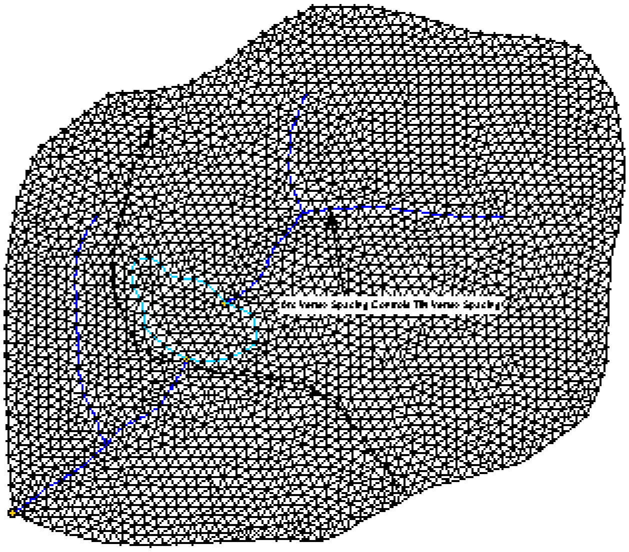
Drainage

The *Drainage* Coverage is the primary coverage used by WMS. Whenever starting a new WMS session, an "empty" coverage is created and assigned the drainage coverage type. When starting to creating points, arcs, and polygons, by default they will belong to this drainage coverage. Most of the work done 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, 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 wanting 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. Ridge arcs are used to designate any other (besides boundary, stream, and lake) linear segment to be 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 projects will be working with 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 there are multiple sub-basins, WMS will ignore sub-basin boundary arcs interior to the watershed (define the sub-basins boundaries from the TIN).

Related Topics

•Feature Objects

•Coverages

•Feature Object Guidelines

•Drainage Module Feature Objects

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 incorporating 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

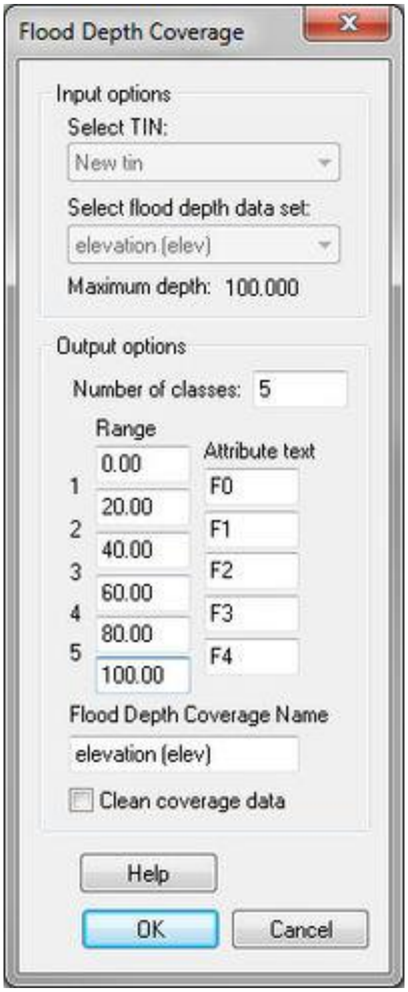
Flood Extent

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

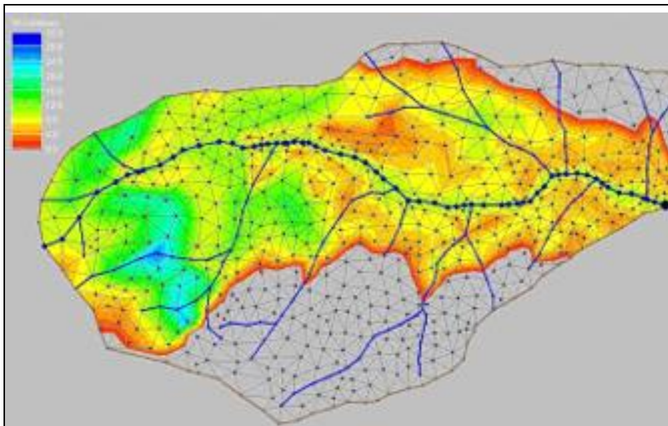
The classified flood depth coverage (*Conversion* | **Flood→Depth Map** command in the *Flood* menu) shows the variation of flood using different classes. Each class represents a range of flood depths (or water surface elevations) specified. On the other hand the flood extent coverage contains the land-water 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 specifying the options for creating the depth map.

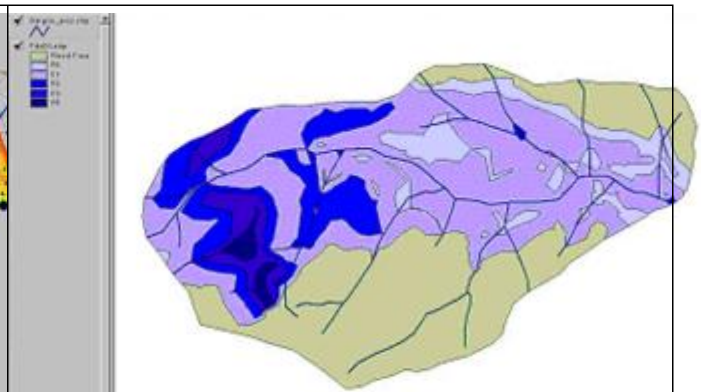


First select the TIN (if more than one TIN is present) and the dataset to create the coverage for (this would typically be a water surface or flood depth dataset computed from the flood delineation process, but note that the elevation dataset may be used as well). The first range will be from 0.0 to 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 comparing 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. It is possible to specify what the change increments 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 creating 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 any positive depth value can be specified as the inundation limit and WMS will generate a polygon that conforms to that limit.

To create the flood extent map aspecify the flood depth and water surface elevation datasets along with the inundation limit. Optionally, have WMS force or stamp in the flood extent coverage to the TIN. This will insure that there are 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

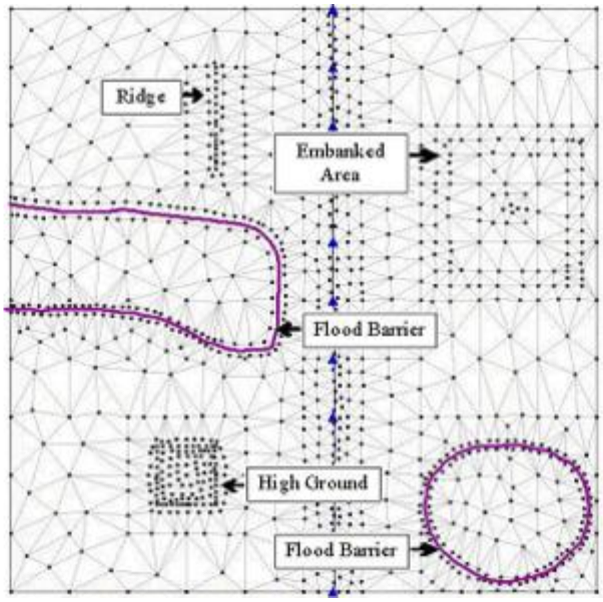
•Coverages

•Overview of Flood Plain Delineation

•Delineate Flood Plain

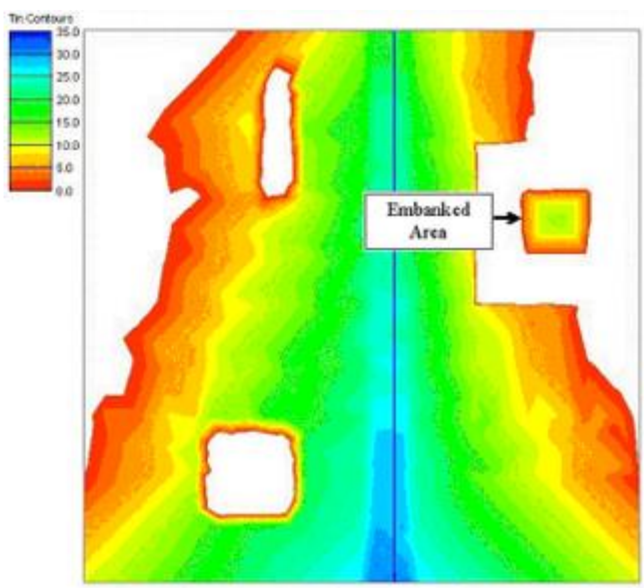
Flow Paths and Barrier Coverages

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.



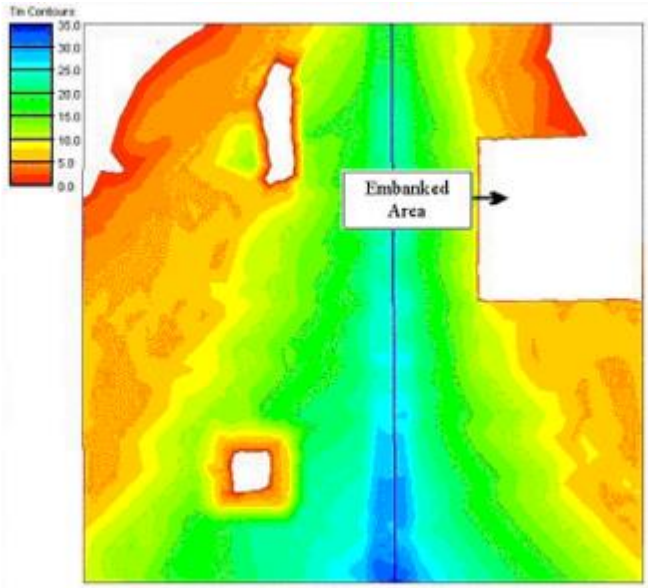
a) Sample TIN for illustrating flood plain delineation options

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.



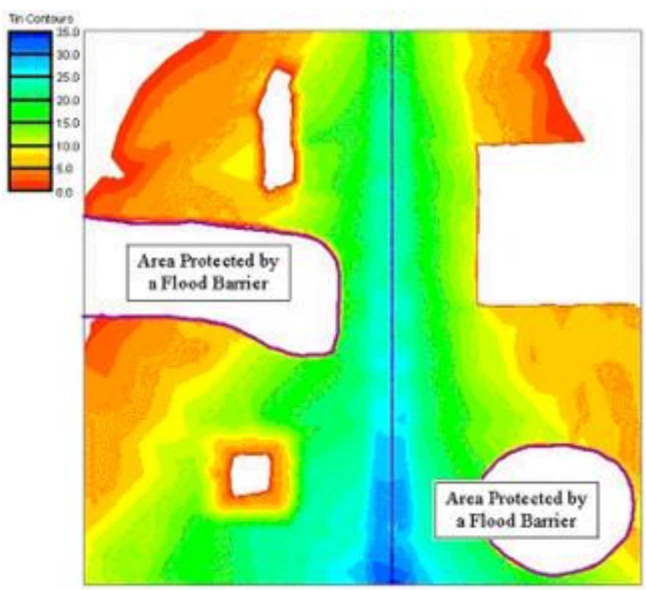
(b) Flood plain delineation without consideration of flow paths

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

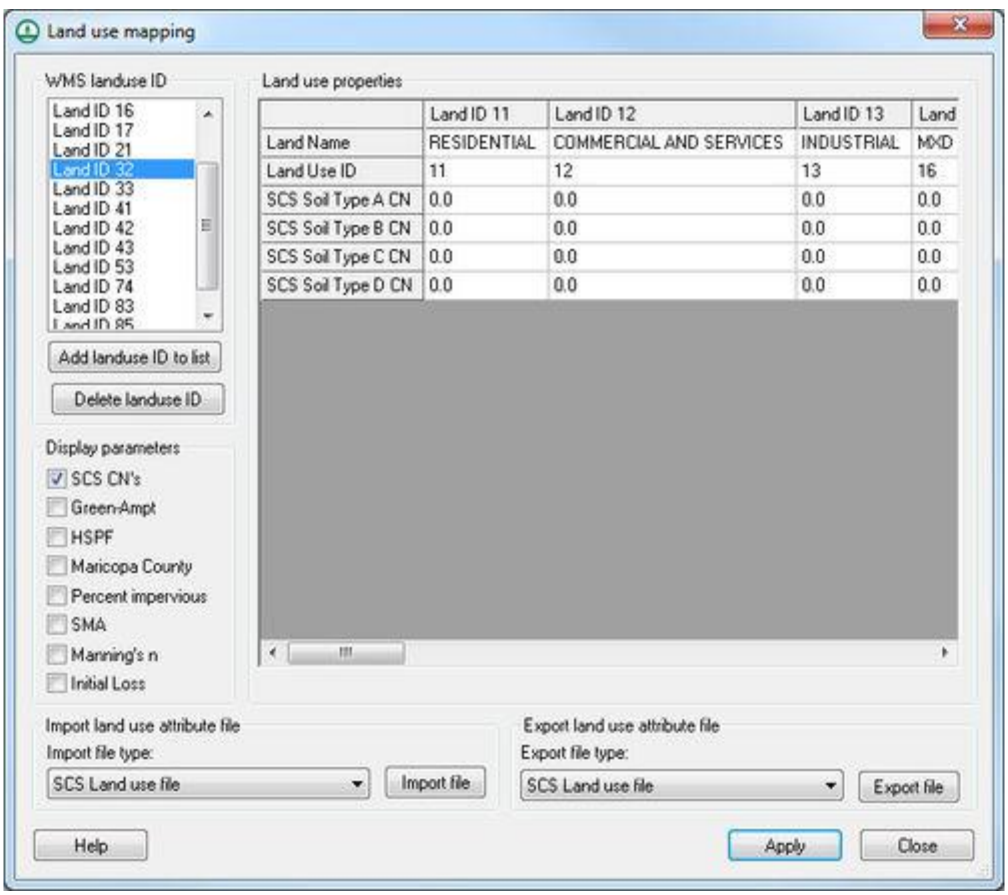
Sometimes, which coverage attributes to use is uncertain, especially when importing a layer from another source (GIS, CAD). The general coverage allows creating or editing points/nodes, arcs, and polygons without defining attributes, or worrying about special rules associated the different coverages and attributes. For examples WMS requires that stream and time computation arcs be defined from downstream to upstream. If importing data from another source, there is no guarantee of the ordering, and if importing them to a Drainage Coverage type, it may cause problems. Therefore, 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 assigned 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.

Indicate which application(s) 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. Parameter values can only be defined 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 it's necessary to link the appropriate land use variables to each land use ID. WMS allows completing this in one of two ways. First of all, with the land use coverage active and after selecting the appropriate polygon, 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, it is possible to 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 choosing to define all the data manually using the *Land Use Mapping* dialog, export the data to a file so that there is no 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 constantly using the same landuse id attributes it is not necessary 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 the state).

If wanting to digitize the state:

1. 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.
2. Load the *.tif, or *.jpg image into WMS and register according to the latitude and longitude coordinates found on the map.
3. Then convert the coordinates to the coordinate system that will be used in delineating the watershed in.
4. With the image in the background, digitize the lines and build the polygons of the coverage.
5. 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

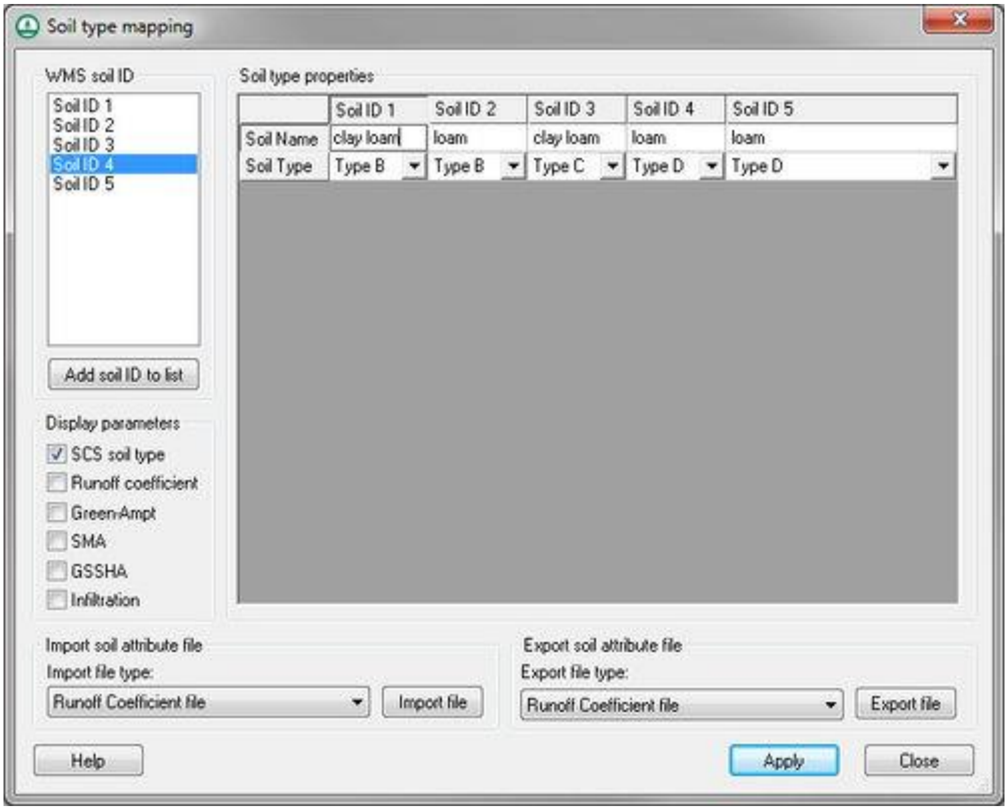
Soil

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.

Indicates which application(s) 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). Parameter values can only be defined 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, it's necessary to link the appropriate soil type variables to each ID. WMS allows completing this in one of two ways. First of all, with the soil type coverage active and having selected the appropriate polygon, 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, it is possible to 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 choosing to define all the data manually using the *Soil Type Mapping* dialog, export the data to a file so that there is no 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

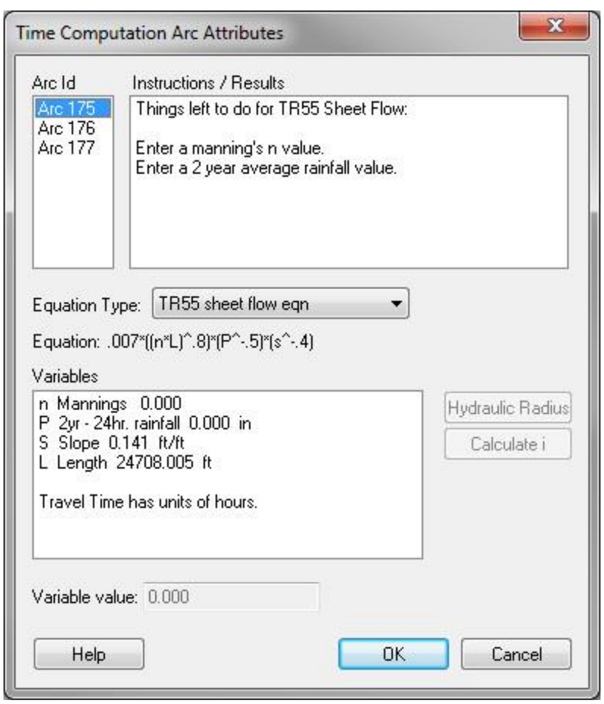
- Feature Objects
- Coverages
- Computing Composite Curve Numbers
- Obtaining Soil Type Data

Time Computation

The Time Computation coverage type allows users 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, a user 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 automatically developing 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. A user 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 to be used to create time computation arcs are selected.
3. Choose the **Node→Flow arcs** command.

A prompt will ask if wanting to create one continuous arc or multiple arcs. With one continuous arc, WMS will create an arc representing the flow path from the selected feature point to the next downstream outlet. With creating 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 wanting to compute the lag time between consecutive outlet points then it's necessary 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, it is possible to direct WMS to define the flow path to the outlet when the **Node to Flow Path** command is issued from the *Feature Objects* menu.

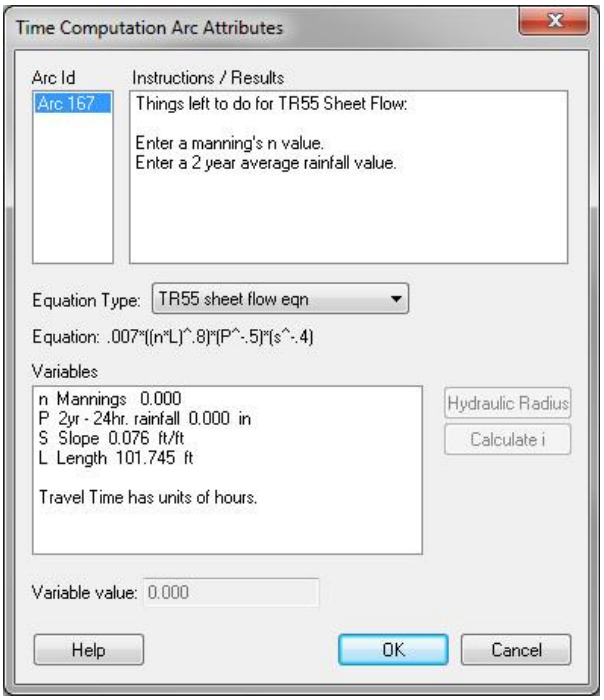
NOTE: the flow path arc for each basin must not cross the basin boundary at any point. Such a crossing will result in errors.

Automated T_c Calculations

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, it is possible to edit either of these values. For example, if decideing 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 it is possible edit the equations or variables of any selected arc by choosing it from this window. While the selected arc is highlighted in the Graphics Window, it may be 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 show 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 to modify from the text window and then setting the value in the adjacent edit field. The *Instructions/Results* window will show 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 the watershed data has been computed and assigned rainfall , soil type , and percent impervious , all the variables will be assigned the appropriate values. If not, select the variables and enter a value manually.

NOTE: Once having selected the LACDPW TC equation for one T_c arc in the model, it becomes the default for all other arcs.

Regression T_c Arc 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 the 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, a prompt will ask 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

5.3.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

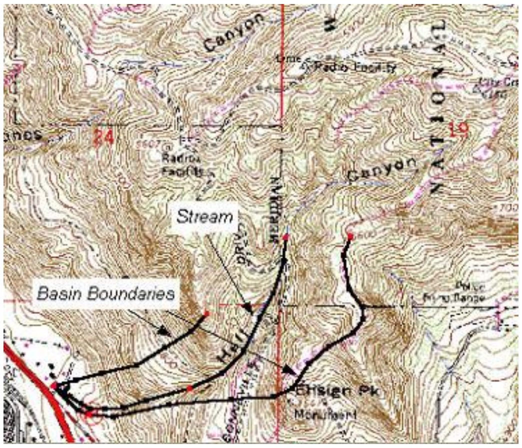
The first step is to obtain a map that defines the streams and basins which will be modeled. If such a map already exists digitally as a CAD drawing or as part of a GIS database then it can be imported directly and the next step can be skipped.

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, complete one of two things:

1) digital elevation data that can be contoured by WMS

2) a scanned TIFF image that can be read into WMS and used as a background map.



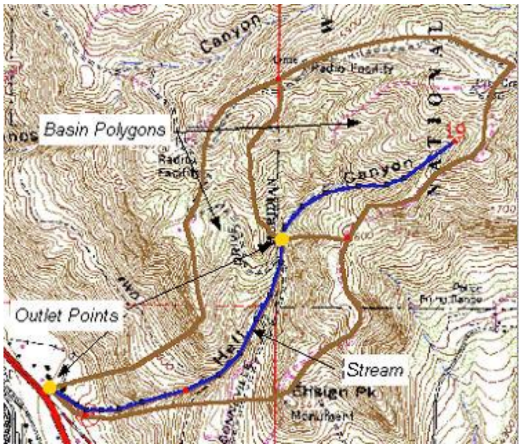
3. Construct Feature Object Topology

The points and lines must be assigned the right attribute types, and the polygons for sub-basins constructed from the lines.

If using data already developed in a GIS then some editing may be needed. 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.

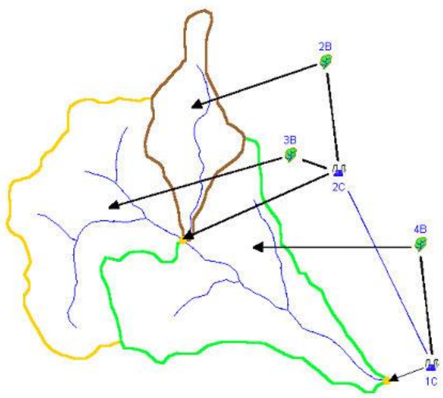
If all three layers exist then construction of the watershed model topology can proceed, but if one or more of the layers are absent, they must be created manually from within WMS. For example if the the project only has a file that contained sub-basin boundaries, digitize the stream network and define 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, always create streams from downstream to upstream. If importing a set of lines that has been previously created by another program, it is possible 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 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

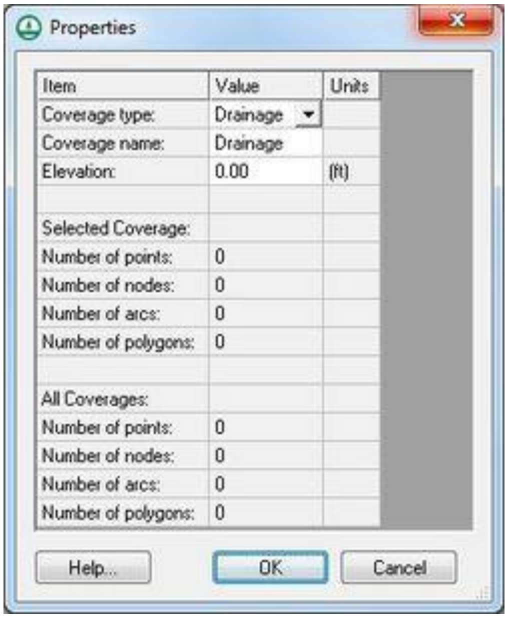
The *Feature Objects* menu has the following commands:

Delete

The **Delete** command will delete all feature objects and any unsaved data.

Coverage Properties

The **Coverage Properties** command will bring up a *Properties* dialog. From the Coverage *Properties* dialog the model coverage can be can be changed. Information about feature objects built in the coverage can also be viewed in this dialog.



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 selecting the **Attributes** command in the *Feature Objects* menu, the attributes dialog of the currently selected feature object will appear. It is also possible to 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.

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.

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, there is the option of intersecting only the selected arcs. If any points, nodes, or vertices are selected, 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:

4. 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 wanting a stream arc to also form a basin boundary, then build polygons according to method two below.

5. 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.

Redistribute

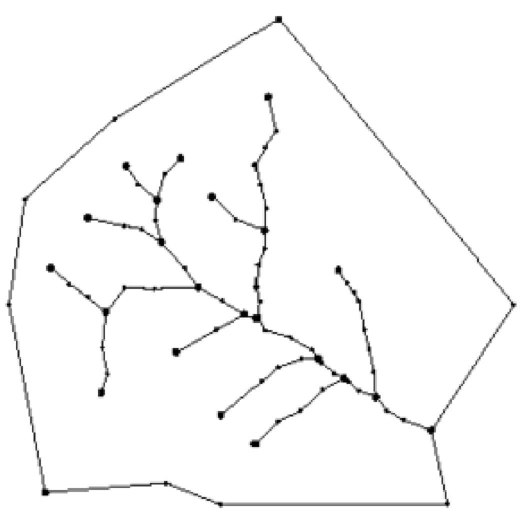
Vertices along arcs can be redistributed at either a higher or lower density using the **Redistribute** command in the *Feature Objects* menu. The vertex density along arcs determines the density of TIN vertices when issuing the **Create TIN** command from the *Feature Objects* menu. Vertices are redistributed 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

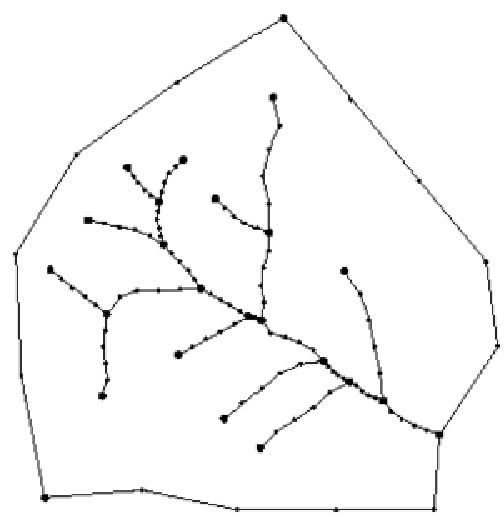
If the Subdivide each end uniformly options is specified then either a number of intervals, or a specified spacing can be given to determine how points are redistributed along the selected arcs. If the specified spacing is greater than the length between adjacent arc vertices, the vertices are moved to reflect this larger spacing.

Spline Redistribution

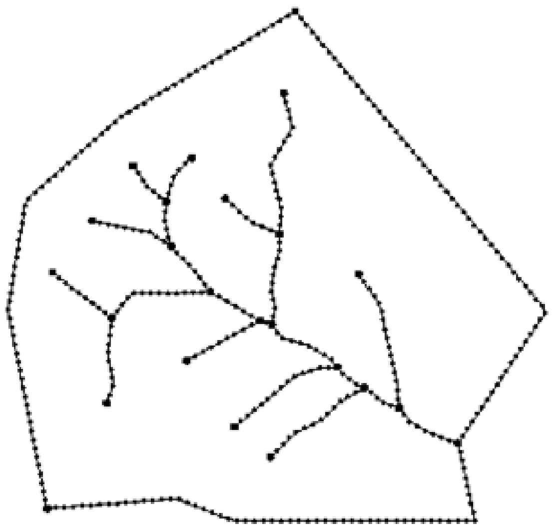
If the Redistribute along a cubic spline option is specified vertices between arcs are redistributed by creating a series of splines from the vertices of selected arcs and then redistributing a new set of vertices at the specified distance. If the flag for preserving points with an angle greater than 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

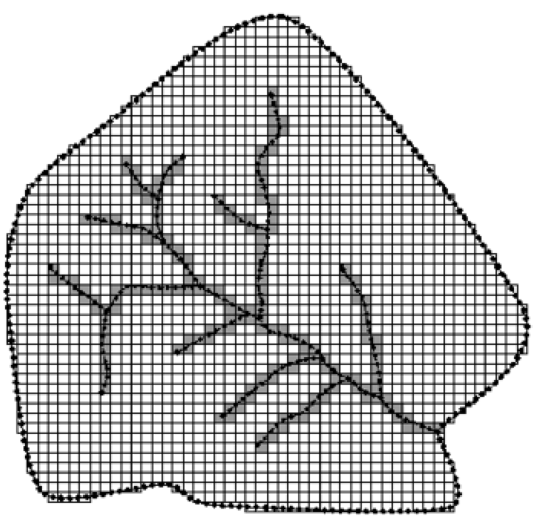
Arcs with a splined redistribution

Create TIN

See the article Creating TINs for more information on this command.

Create Grid

A grid can be created from a feature polygon using the **Create Grid** command in the *Feature Objects* menu when using the Map Module or selecting **Create Grid** from the *Grids* menu when the 2-D Grid Module. Either command will bring up the *Create Grid* dialog. 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 dataset of the grid.



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.

Node to Flow Arcs

The **Node → Flow Arcs** command requires the "Time Computation" coverage have been created and a DEM dataset must have been loaded into WMS.

Stream to Flow Arcs

The **Stream → Flow Arcs** command requires the "Time Computation" coverage have been created.

Compute 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 .

The **Compute Basin Data** command brings up a *Units* dialog.

Related Topics

- Feature Objects
- Coverages
- Creating Watershed Models
- Creating TINs

5.4. Hydrologic Modeling Module

Hydrologic Modeling Module

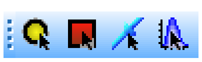
Hydrologic analysis is typically done using lumped parameter models such as HEC-1. The Tree module provides a graphical interface to HEC-1, TR-20, HSPF, TR-55, Rational Method, the National Flood Frequency (NFF), and other programs. In the absence of terrain data, topological or tree representations 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 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.

The Hydrologic Modeling module is included will all paid editions of WMS. This module is the primary module available for the public domain version.

Related Topics

- HEC-1
- HEC-HMS
- TR-20
- TR-55
- NSS
- Rational Method
- HSPF
- MODRAT (LA County Modified Rational Method)

Hydrologic Modeling Tools



The toolbar for the WMS Hydrologic Modeling module has a variety of tools useful for editing and selecting items within a topological tree. The tools are described below.

Tools

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

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 users to copy a hydrograph from the clipboard, or open a text file using the *File 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 the selected hydrograph(s). If wanting to view more than one hydrograph, select multiple hydrograph icons from the graphics window before choosing this command. It is also possible to 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

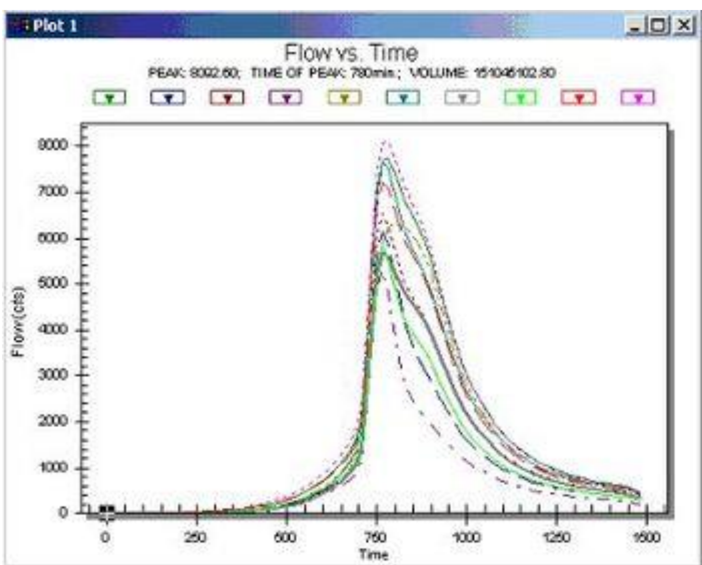
Hydrographs can be read into WMS from a results files using the **Open** command in the *Hydrographs* menu. Hydrographs are generated for each basin and outlet point when running one of the hydrologic models. Two hydrographs may be generated for the outlet points, one representing the combination of "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 *File Import Wizard* . The basin or outlet where defining the hydrograph (i.e. the point where wanting 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 wanting 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. It is also possible to double-click on the last of the hydrographs selected to bring up all selected hydrographs.

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. It's possible to 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.

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 wanting to associate the new hydrograph. When selecting this command, 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 *File 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 *File Import Wizard* and header rows can be idnetified, starting row set, and the columns mapped to their respective data fields.

Exporting Hydrographs

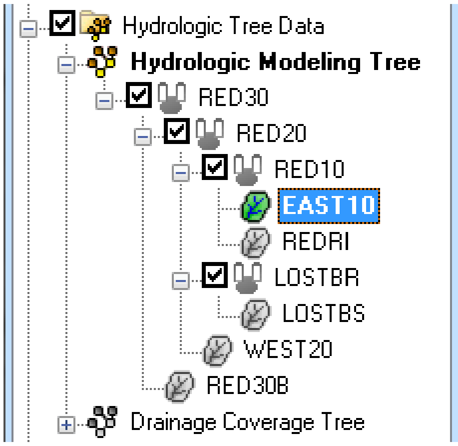
Selected hydrograph(s) may be exported to either the native WMS file format or to a space, comma, or tab delimited text file that can be read into a spreadsheet program like Microsoft Excel.

Related Topics

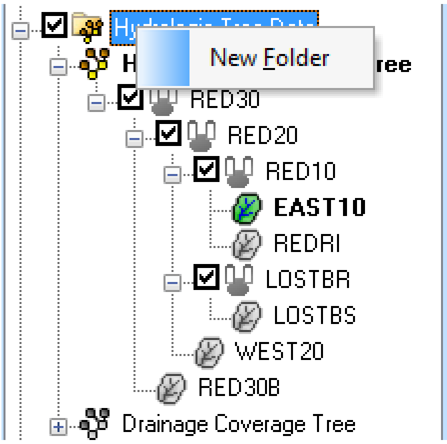
- Detention Basin Calculator

Project Explorer Contents for Hydrologic Modeling Module

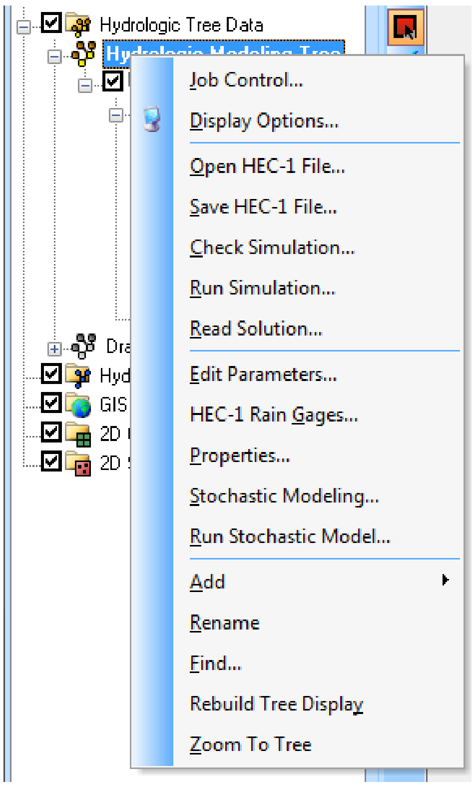
In the hydrologic modeling module the Project Explorer displays the identical structure as the topologic tree providing an alternate hierarchical view of the hydrologic model (diversions are not displayed in the Project Explorer). A check box controlling the display of an entire folder (model) and outlet points is displayed to the left of the icon. If unchecking 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.



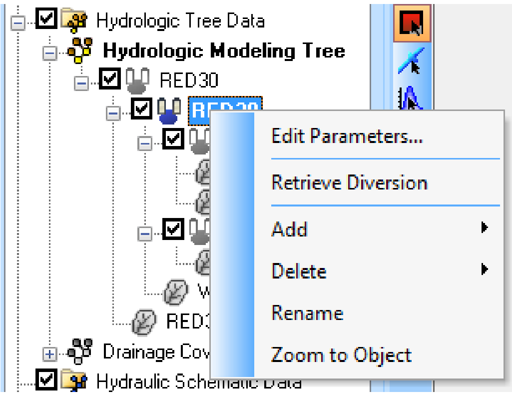
Right-clicking on the main Hydrologic Tree Data folder allows creating a New folder (at this point a topologic tree for hydrologic model cannot be created directly from the Project Explorer).



Right-clicking on the Hydrologic Modeling Tree object will bring up a menu with commands for the current hydrologic model.



Right-clicking on a basin or outlet icon gives the option 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.



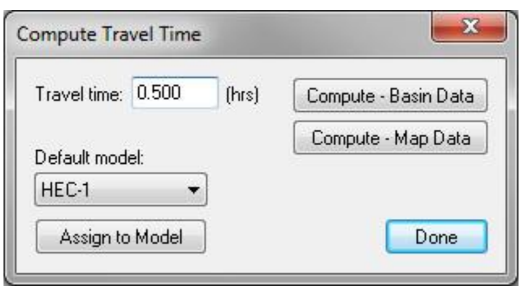
Related Topic

•[Hydrologic Modeling Module]

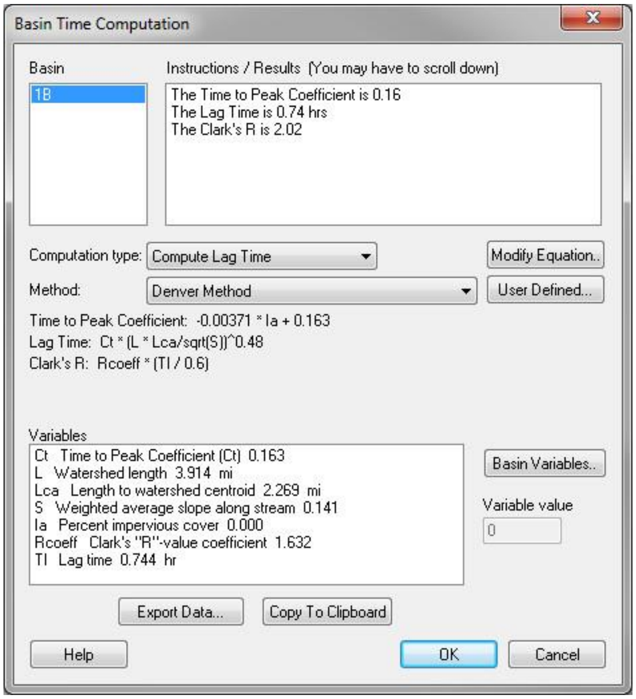
•[Project Explorer Overview]

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. This is done by selecting the basin or outlet then choosing the **Compute Travel Time** option from the *Calculators* menu in the Hydrologic Modeling module. 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.

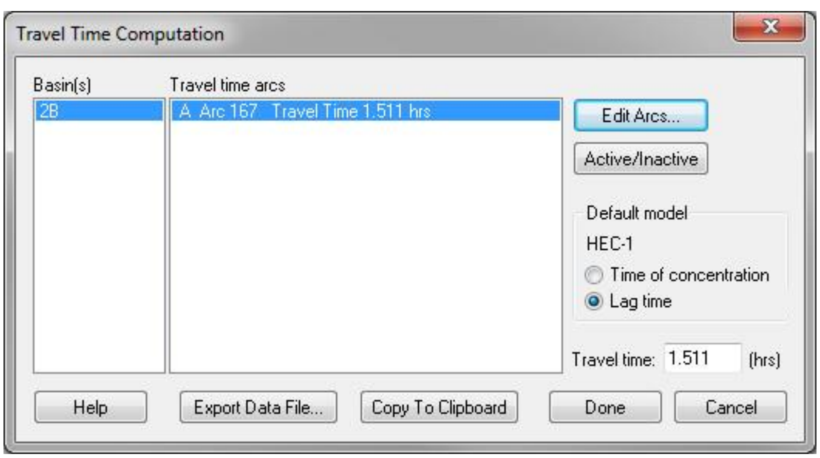


Selecting the **Compute – Basin Data** button brings up a dialog similar to the *Assigning Equation to Arcs* dialog where time computation types and methods can be selected. This dialog has a Modify Equation and a *User Defined* option where parameters can be changed for various equations.



The *Travel Time Computation* dialog is accessed by selecting the **Compute – Map Data** button from the *Compute Travel Time* dialog. When it is accessed, the list appears with the 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 having 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. It is possible to either accept the computed value for travel time or edit the value (override computed value) as may be appropriate. If wanting to change the equation definitions or variable values for any or all of the arcs, 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 either appending to an existing file (so that a single report for multiple basins can be created) or creating 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, it's necessary to enter a series of *Elevation* and *Discharge* values (it's necessary to have the same number of values in each series) to define the relationship.

If the *Discharge Structures* option is chosen, it is possible to 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, 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 specified. Be certain to specify a range of discharges in the HY-8 analysis that corresponds to the input hydrograph discharges.

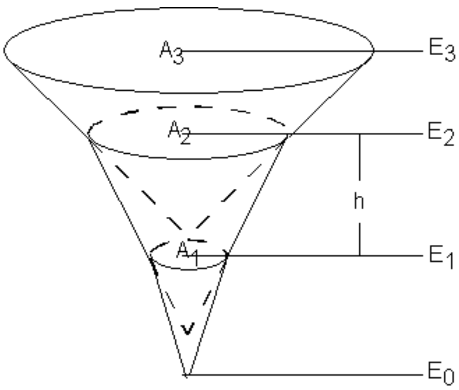
Related Topics

- 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

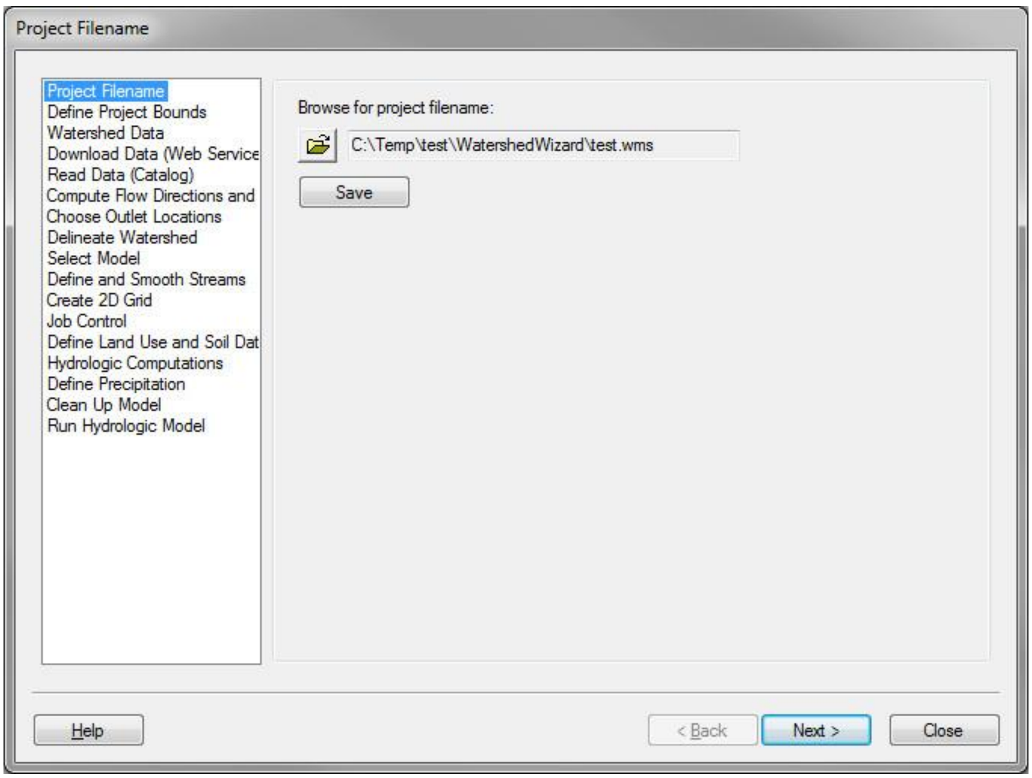
Hydrologic Modeling Wizard Overview

The WMS **Hydrologic Modeling Wizard** ( ) is a simple tool that guides 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

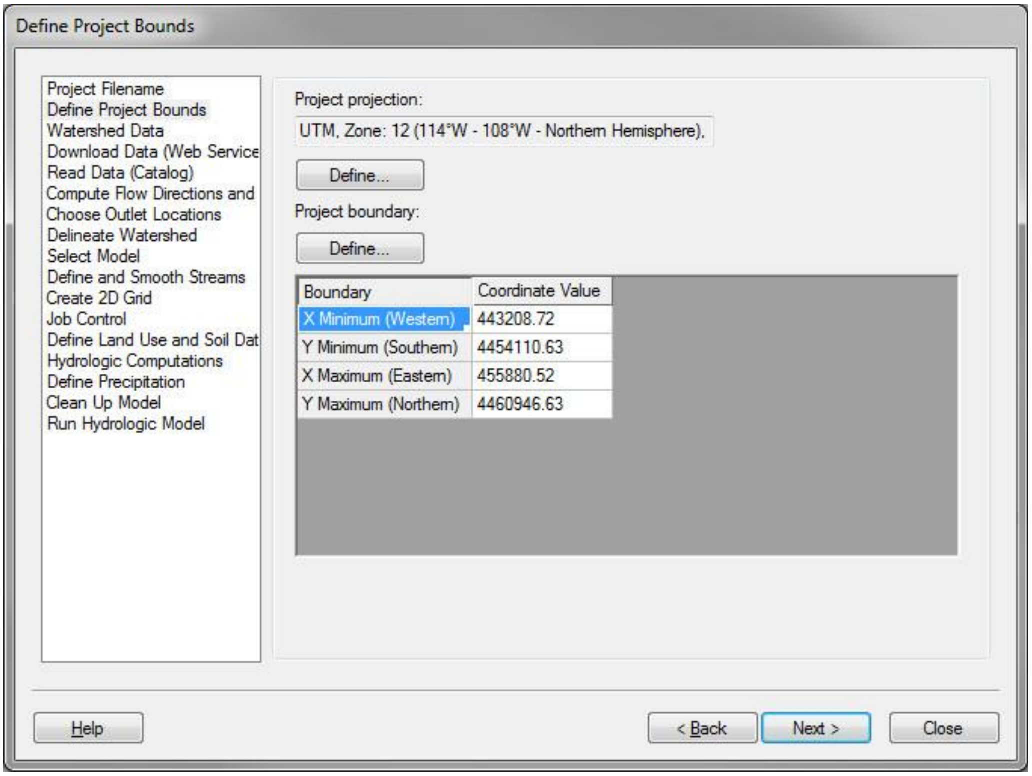


The project filename step is used for defining a project filename. This filename is used when saving files from the hydrologic modeling wizard.

Help

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

Define Project Bounds



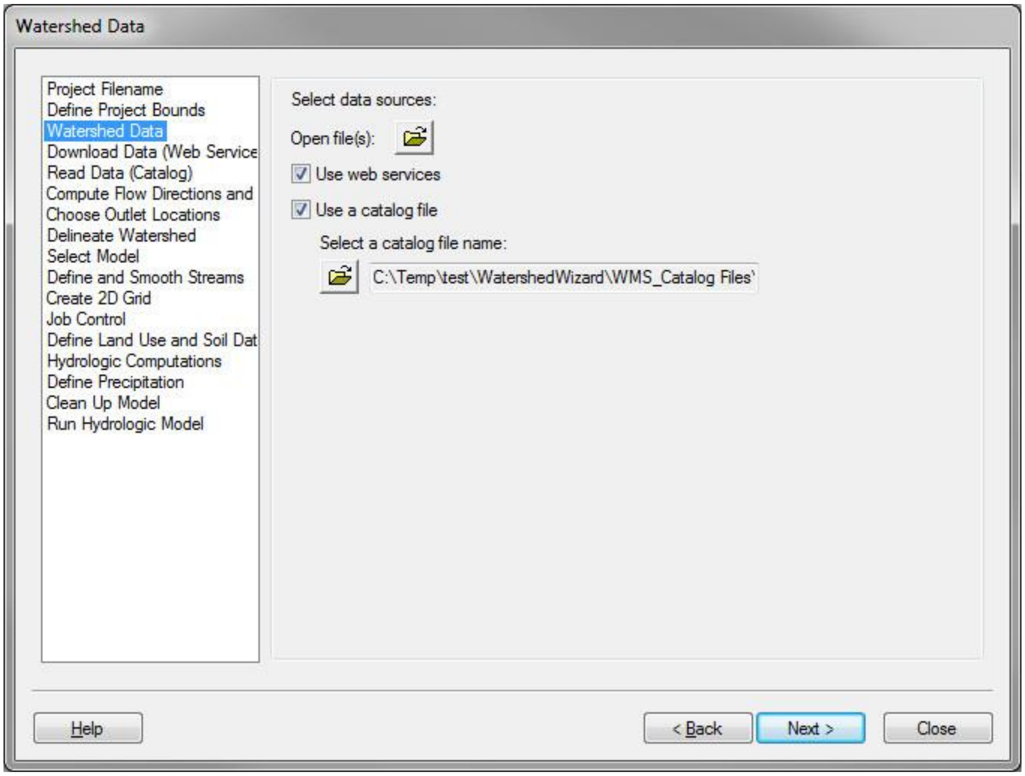
The define project bounds step is used for defining the project boundary.

Help

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

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

Watershed Data



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

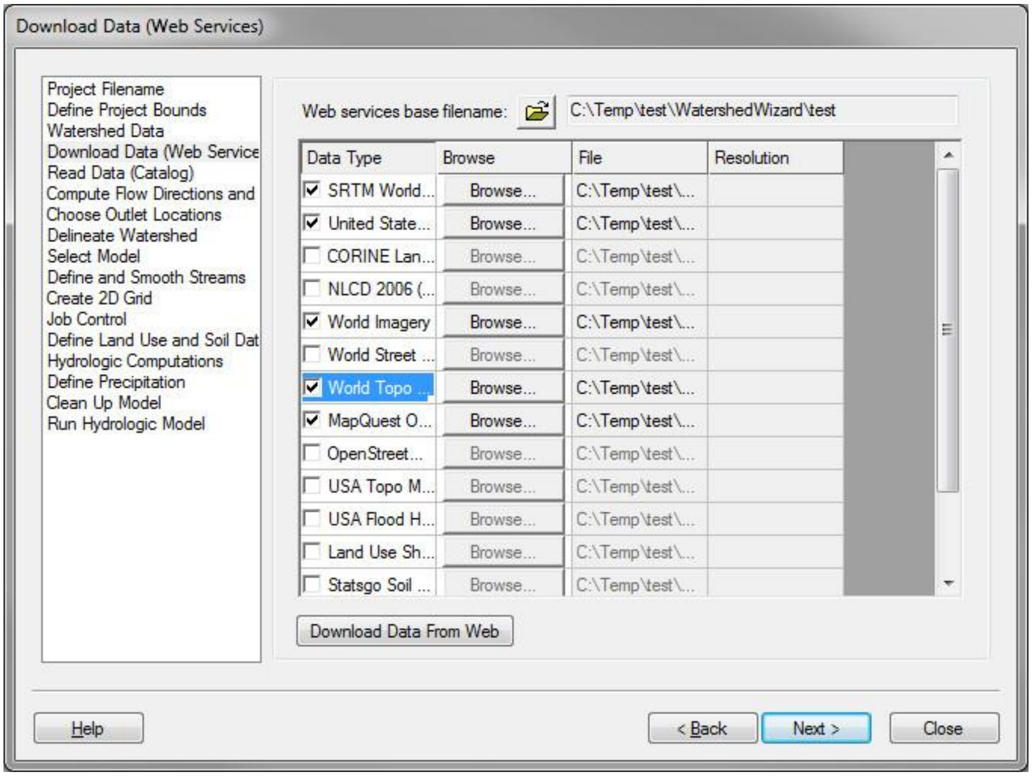
Help

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 selected.

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

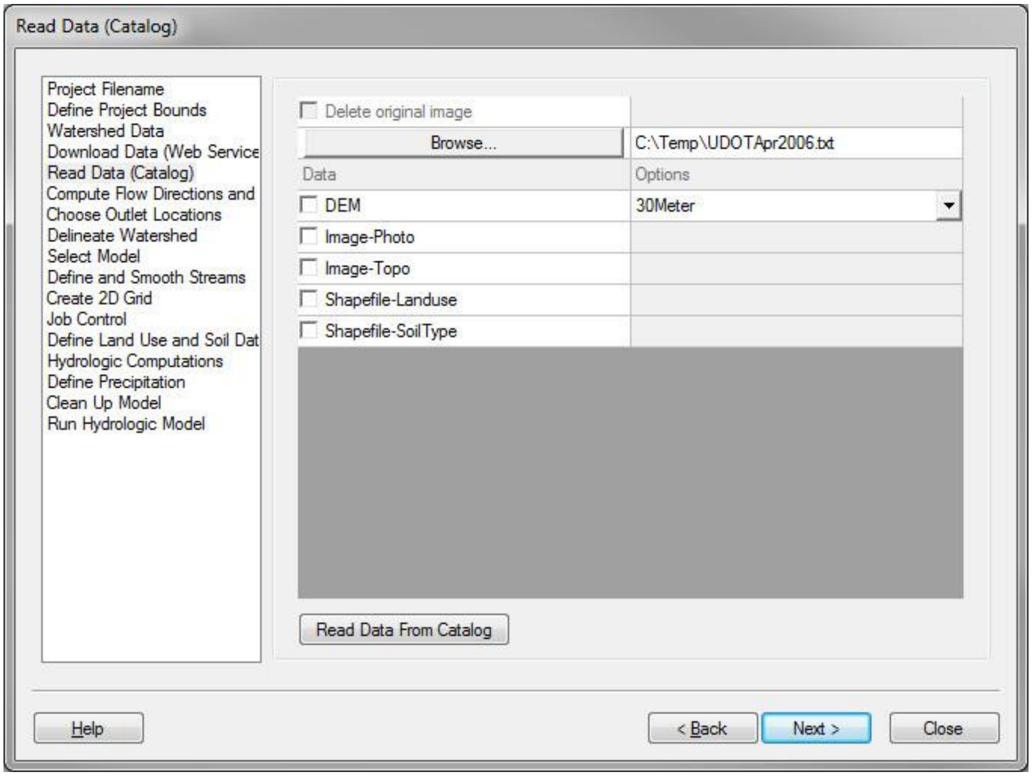


The download data (web service client) step is used to download data over the internet using the WMS web service client if a web service is available. NED and SRTM data may be downloaded from this dialog, but coverage may not exist or download times may be slow for the hydrologic modeling area. The NED 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 the selected modeling area from the web service providers.

Read Data

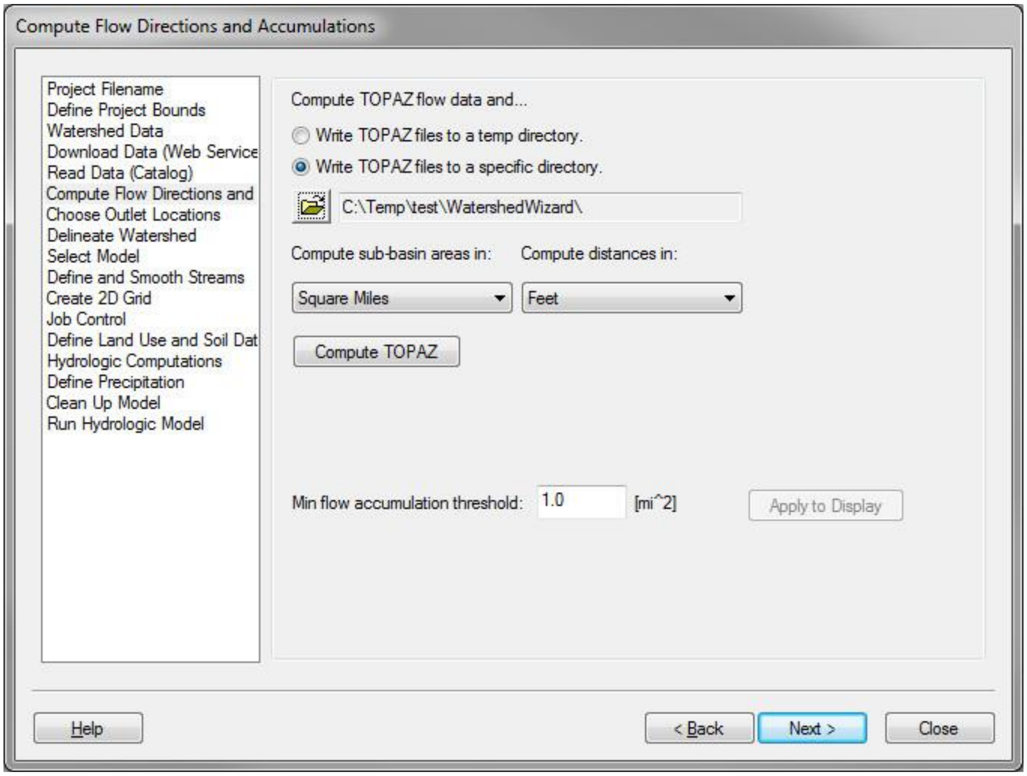


The read data from catalog step is used to select and read data from a WMS catalog.

Help

Read Data From Catalog – This button reads data from the catalog inside the 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.

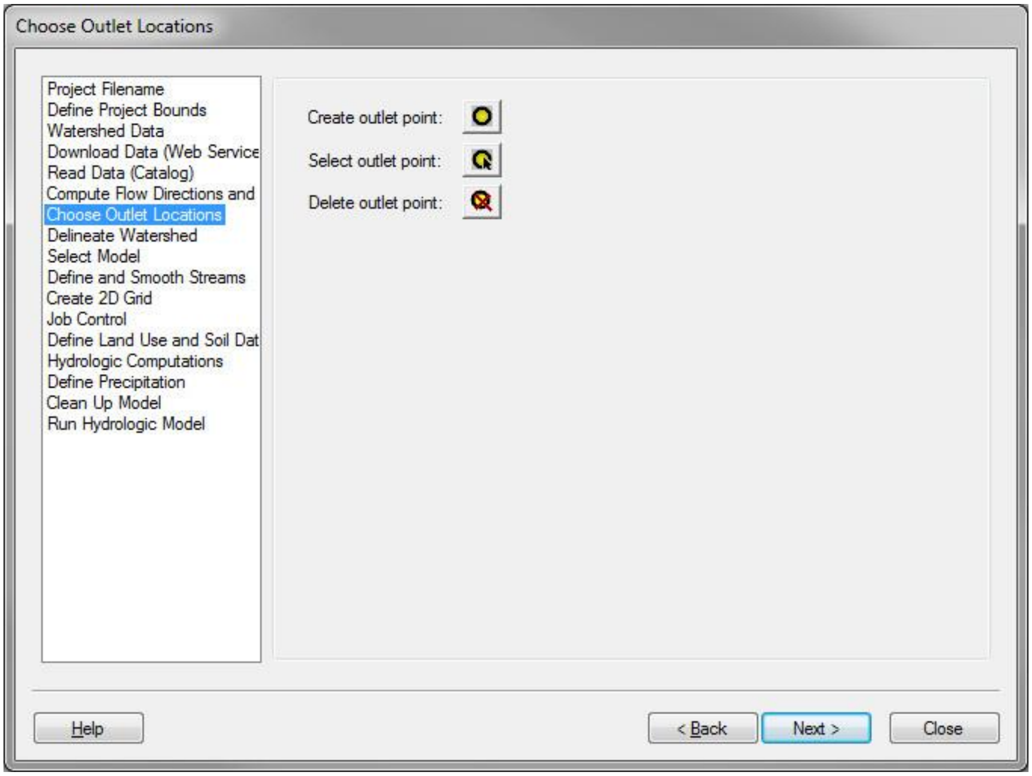
Help

Compute TOPAZ flow data and... – Choose either to write the TOPAZ output to a specific directory or to the temp directory. Most frequently, write the TOPAZ output to a temp directory.

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

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

Choose Outlet Locations



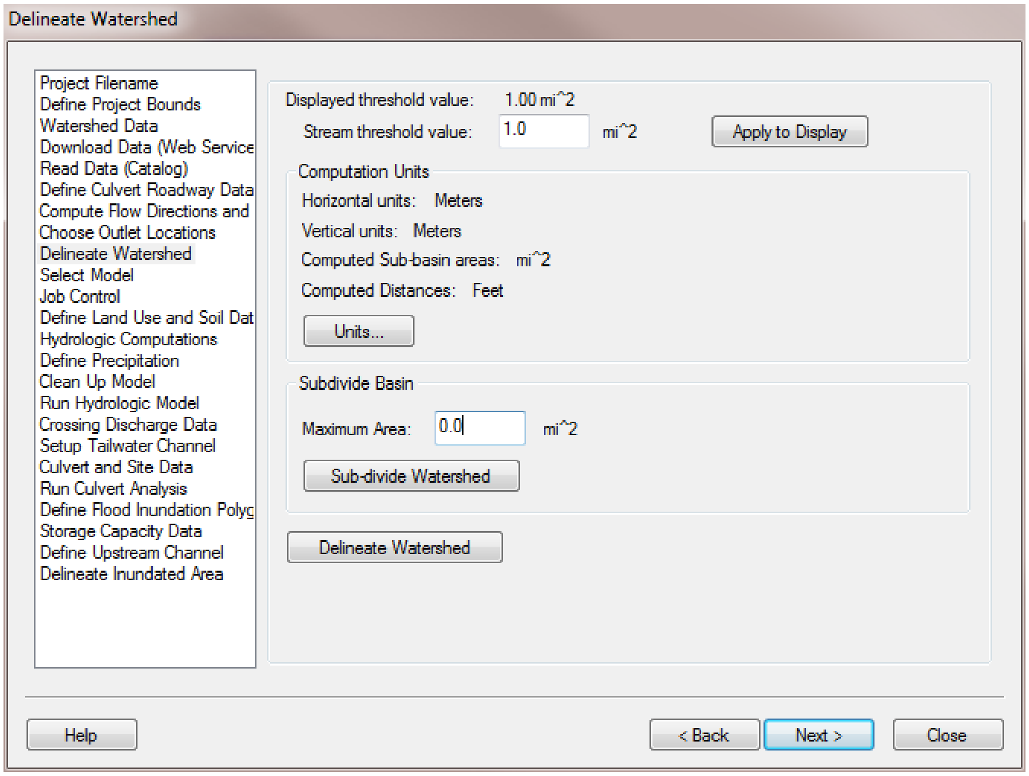
The Choose Outlet Locations step is used to define, move, or delete outlet locations.

Help

Create , **Select** , and **Delete** outlet point buttons – To create an outlet point, select a tool 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.

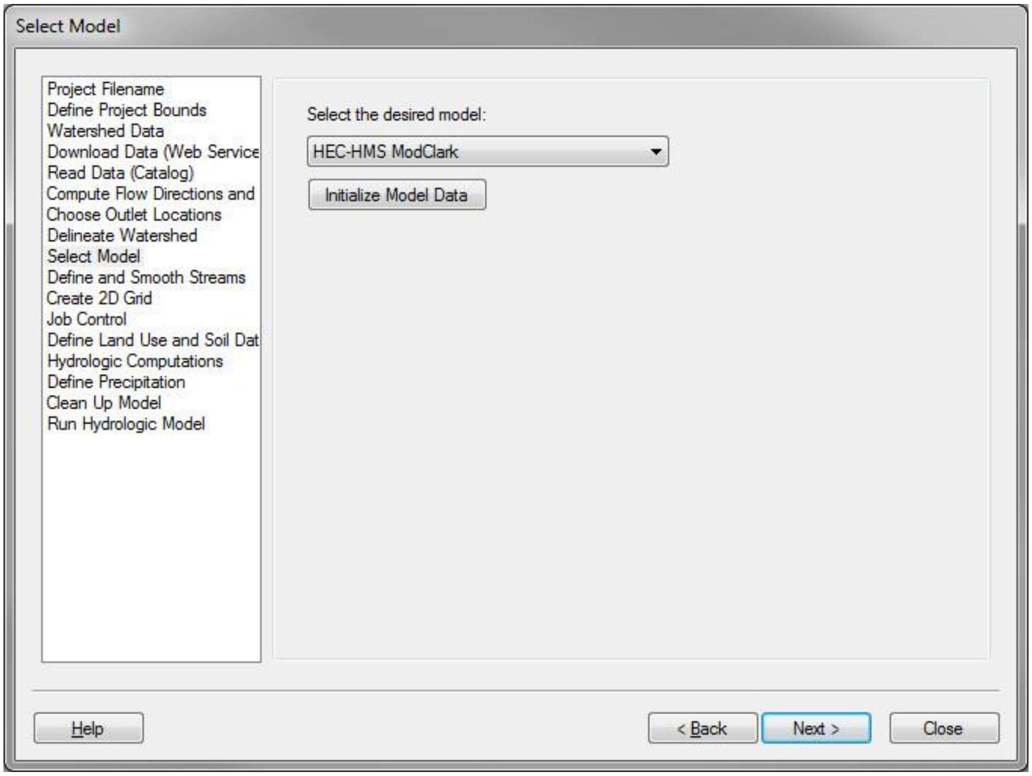
Units – This button is used to define the model and computation coordinates and units. Turn on the option to create a Tc coverage and change which data is computed by selecting the *Drain Data Compute Opts...* button in the *Units* dialog.

Sub-divide Watershed – This button subdivides the 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

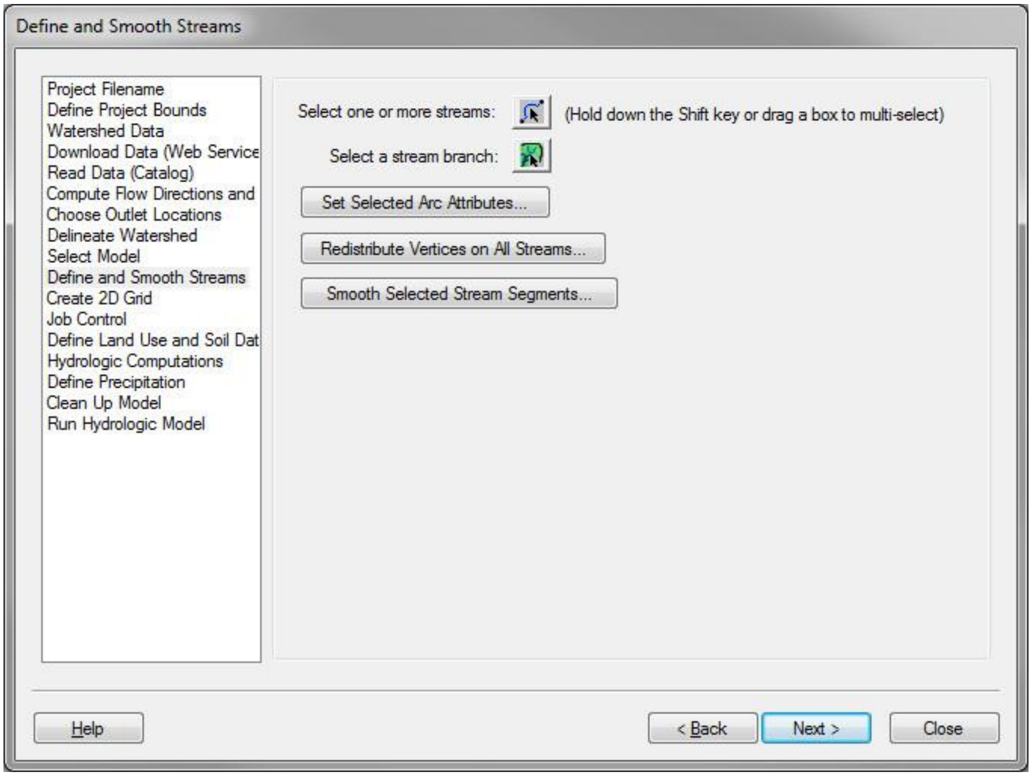


The Select Model step is used for defining which model to run with the delineated watershed.

Help

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

Define and Smooth Streams



When building a GSSHA model, define the parameters for and smooth all the streams in the 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, select one or more stream arcs and edit the attributes or smooth the selected stream arcs.

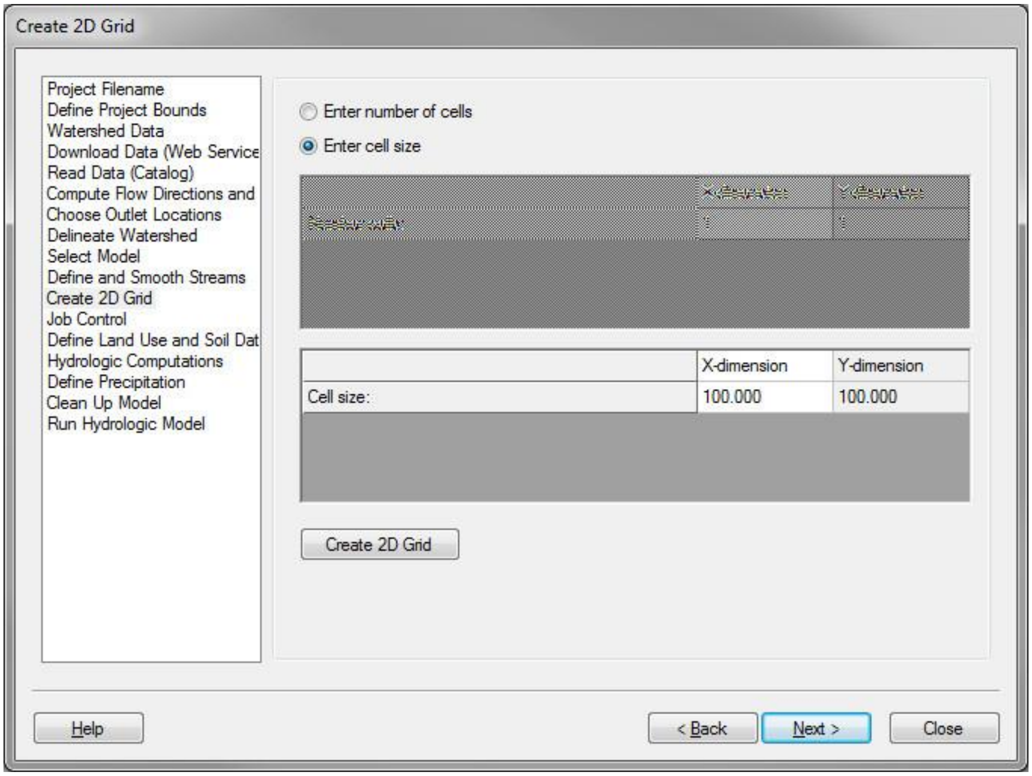
Select a stream branch – This button selects the **Select Stream Branch** tool . This is a specialized tool that selects a stream arc and any arcs upstream from the selected arc. This tool is useful for selecting an entire stream network or a branch of a stream network.

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

Redistribute Vertices on All Streams – Selecting this button will first switch the 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 which allows setting 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, 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

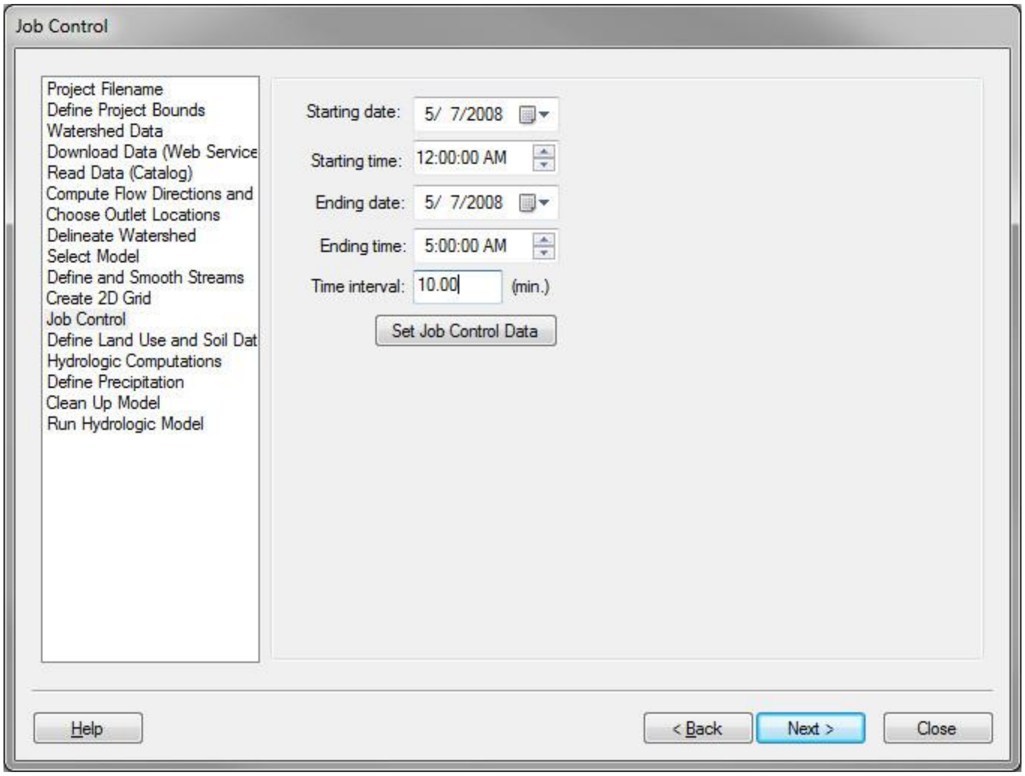


The Create 2D Grid step is used to create a 2D grid from the watershed boundary. Select a grid cell size or number of cells and select the **Create 2D Grid** button to create a 2D grid.

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

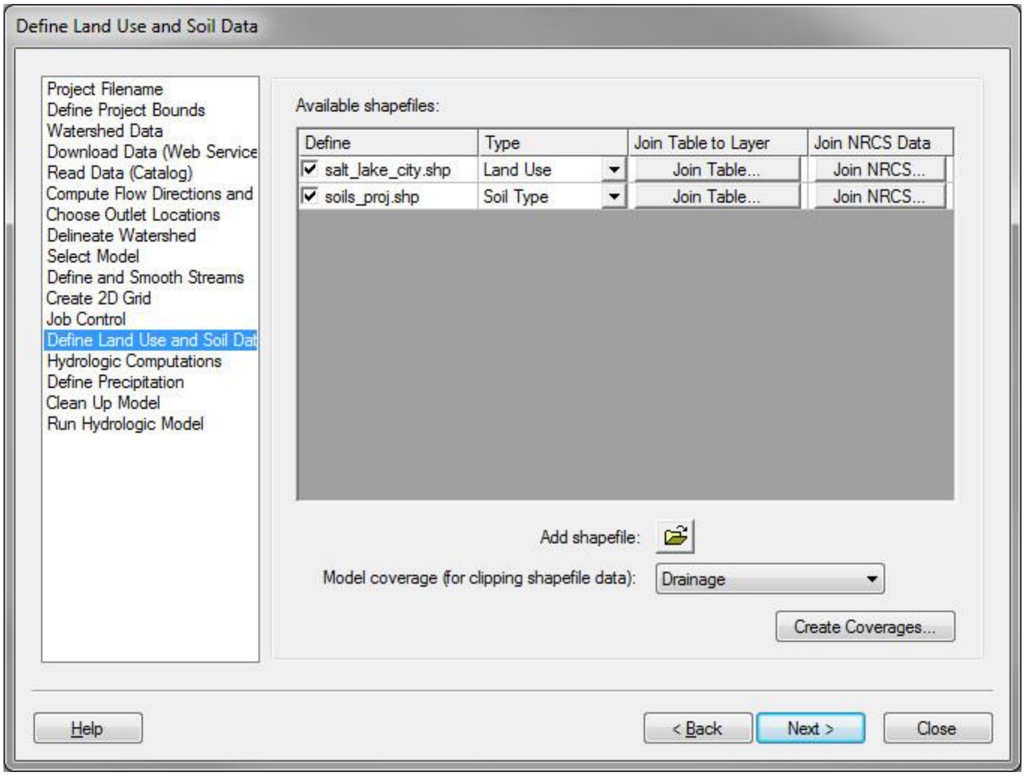


The job control step is used define the time parameters for running the model. Define the start and end time and date and select **Set Job Control Data** to set the job control parameters for the selected model.

Help

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

Define Land Use and Soil Data

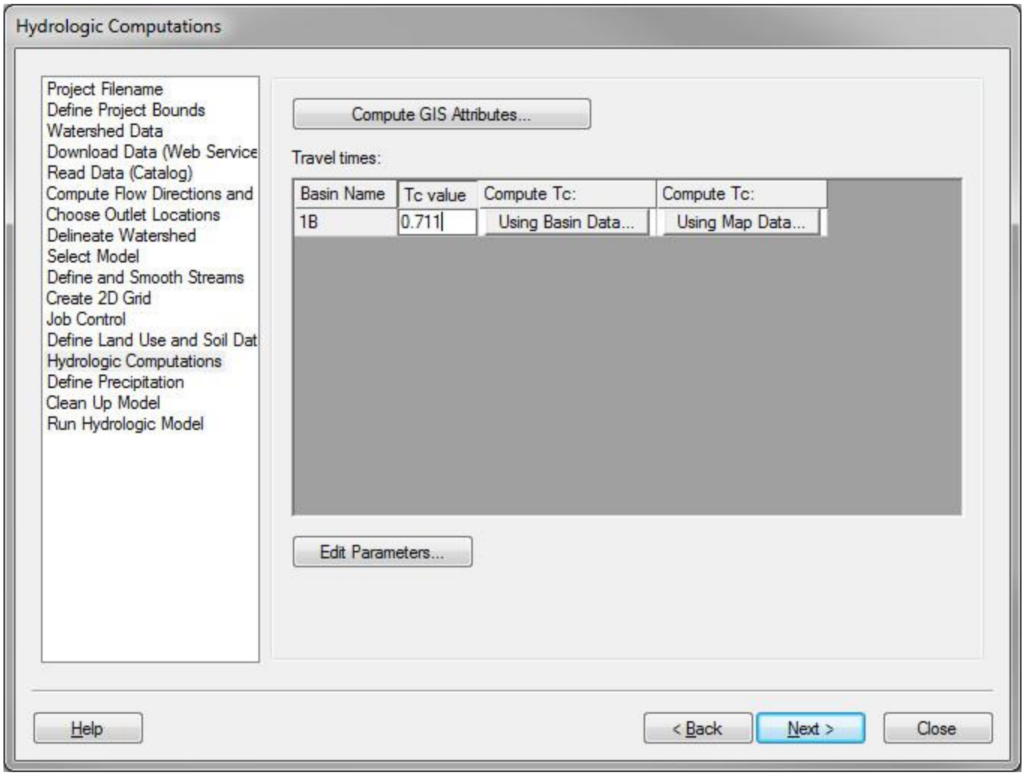


The Define Land Use and Soil Data step converts GIS Module shapefiles to data in the WMS Map module. WMS uses the boundary of the watershed to clip the shapefile data for the selected files.

Help

Create Coverages – Defines the land use and soil type shapefiles. A model coverage and the land use/soil data will be transferred to coverages in the map module then clipped to the watershed boundary.

Hydrologic Computations



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

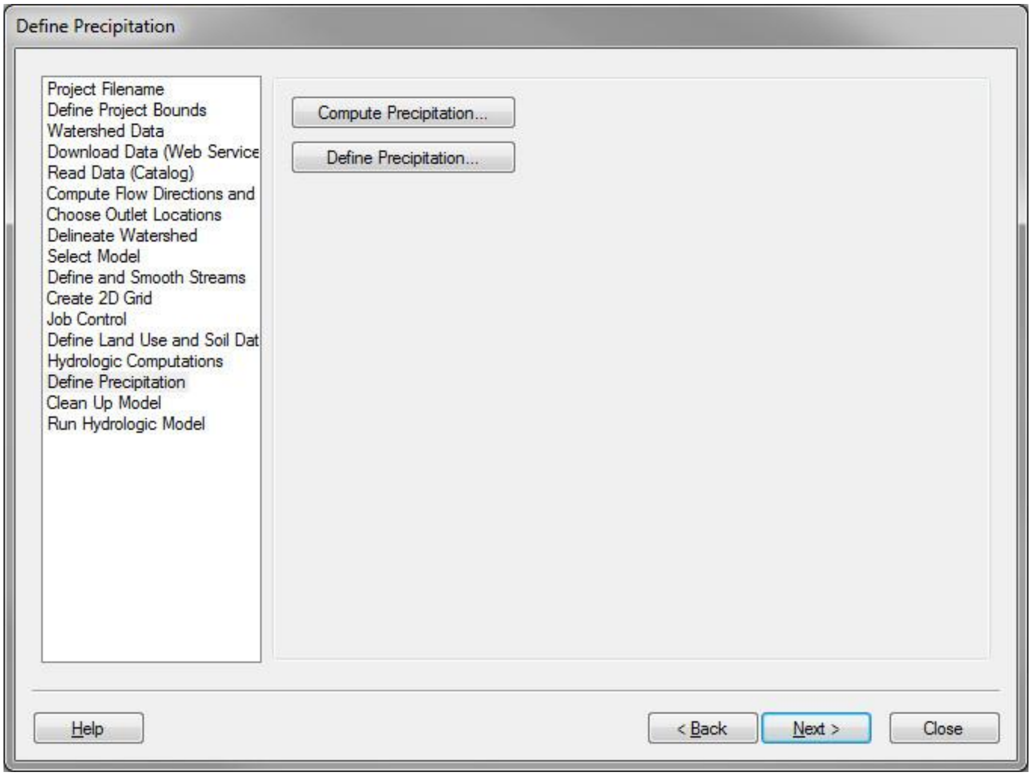
Help

Compute GIS Attributes – Depending on the model being used, this button brings up a dialog which can compute attributes from the land use and/or soil type data.

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

Edit Parameters – This button brings up the *Edit Parameters* dialog for the model to change modeling parameters for all the sub-basins in the watershed.

Define Precipitation



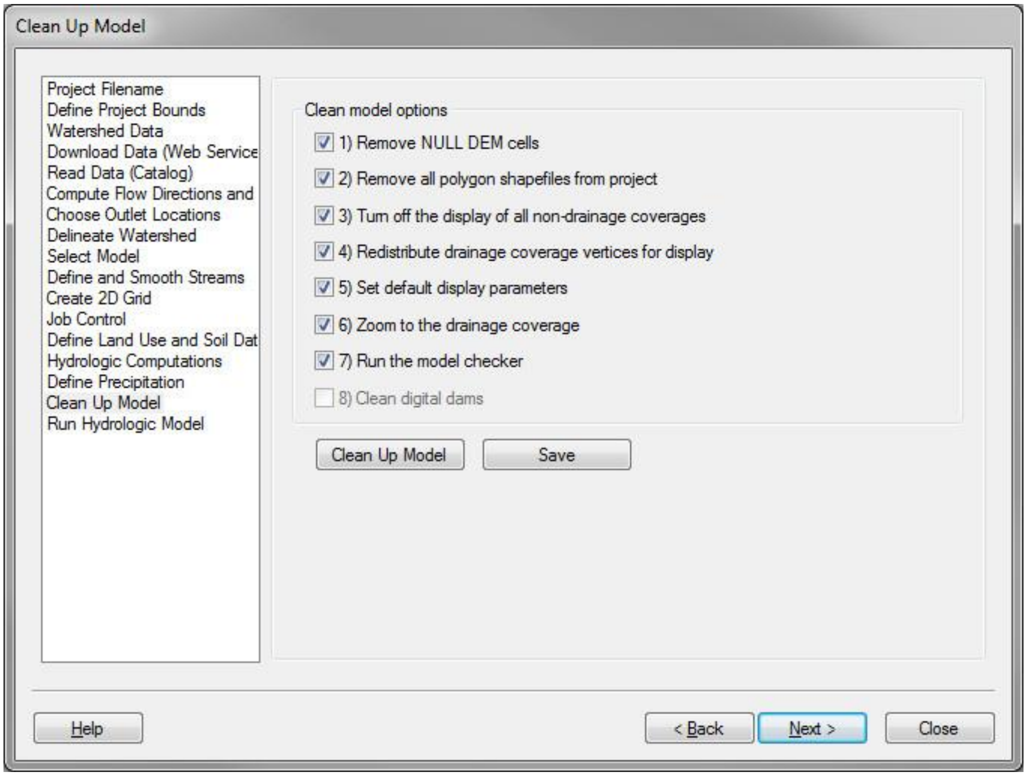
The Define Precipitation step is used to compute and define precipitation for the model.

Help

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

Define Precipitation – This button allows defining precipitation for the selected model.

Clean Up Model



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

5.4.1 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 being used for basin delineation and parameter estimation.

•HY8 Culvert Analysis – 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.

Related Topics

•Hydraulic Toolbox

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 being used 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 creating 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.

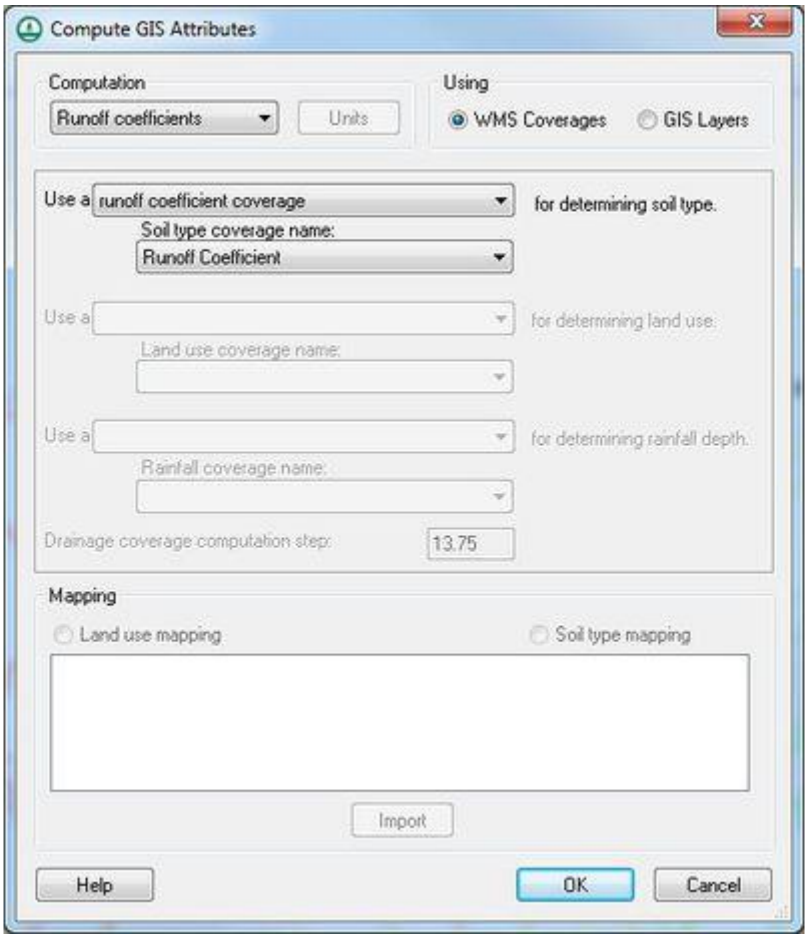
There is no distinct advantage of one method over the other. Each allows a certain amount of customization and the ability to generate a summary report in a text file or by copying to the clipboard so that these critical input data can be well documented. In general, if time of travel can be determined 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 selecting 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 being working on (i.e. TR-55 time of concentration, TR-20 lag time, etc.). It is also possible to compute travel times for selected basins or outlets using the **Compute Travel Time** command from the *Calculators* menu. This dialog will allow choosing 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 defining 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 joining tables or specific attributes from tables to the 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, the project is 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. Download one of the following tables and use them as a template for land use data.

- If using vector landuse data available as a shapefile from the USGS, download and use this table .
- If using NLCD 2006 land use data as a land use grid, download and use this table .
- If using Global Land Cover data as a shapefile, download and use this table .

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

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.

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 b Values

Required Inputs:

- Land use data
- Table relating land use IDs to and 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:

- 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, O .

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

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
- Hydraulic Toolbox

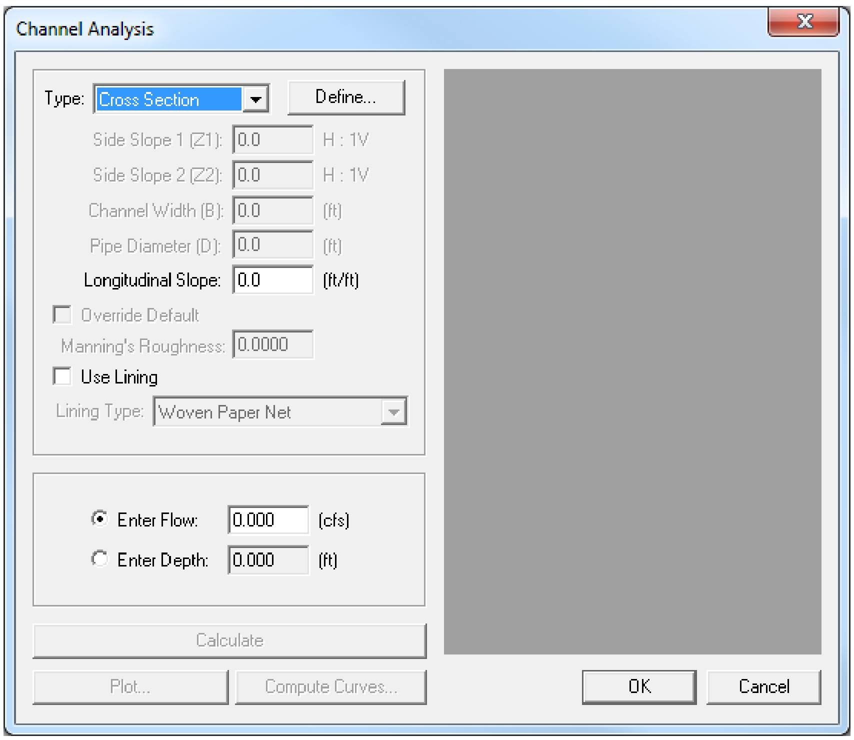
Channel Calculator

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.

Use the measure tool to determine the slope across a specified arc, as long as the user has elevation data loaded into WMS then turn on cross sections. 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 there are computed flowrates from one of the supported hydrologic models, estimate stage or water surface elevation using the channel calculator.

When computing the depth using a cross section arc, 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
- Hydraulic Toolbox

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

•Hydraulic Toolbox

5.5. River/Hydraulic Modeling Module

Hydraulic Modeling

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).

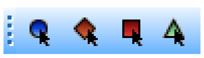
Default Hydraulic Model

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 selecting a command for another model, the default model will be updated to specify that model.

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 Modeling Tools



The toolbar for the WMS Hydraulic Modeling module has a variety of tools useful for editing and selecting items within a topological tree. The tools are described below.

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.

Select Hydraulic Node

The **Select Hydraulic Node** tool is used to select a hydraulic node and edit the node data from within the hydraulic modeling module.

Select Hydraulic Link

The **Select Hydraulic Link** tool is used to select a hydraulic link and edit the link data from within the hydraulic modeling module.

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 Toolbox

The Hydraulic Toolbox software was developed for the Federal Highway Administration by Aquaveo, LCC. This software is used in conjunction with WMS for a number of modeling processes.

More information can be found at the FHWA website .

The Hydrailic Toolbox is included will all paid editions of WMS.

Executable and Library Downloads

- Download Hydraulic Toolbox 4.0 Installation (.zip)
- Download Hydraulic Toolbox 2.1 Installation (.zip)
- Download Hydraulic Toolbox 1.0 Installation (.zip)
- Download C++ Library Redistributable files (.zip)

Release Notes

- Version 1.0 released January 21, 2010
- Version 2.1 released January 20, 2012
- Version 4.0 released December 11, 2012

Reference Papers

Brown, S.A., Schall, J.D., Morris, J.L., Doherty, C.L., Stein, S.M., Warner, J.C., September 2009, “Urban Drainage Design Manual, Hydraulic Engineering Circular No. 22, Third Edition”, FHWA-NHI-10-009, HEC-22. [29]

Kilgore, R.T., and Cotton, G.K., September 2005, "Design of Roadside Channels with Flexible Linings Hydraulic Engineering Circular No. 15, Third Edition , FHWA-NHI-05-114, HEC 15. [30]

McCuen, R.H., Johnson, P.A., Ragan, R.M., October 2002, “Highway Hydrology.” Hydraulic Design Series No. 2, FHWA-NHI-02-001 [31]

Miller, J.F., Frederick, F.H., Tracey, R.J., 1973. “Precipitation-Frequency Atlas of the Western United States.” NOAA Atlas 2, National Weather Service. [32]

Lagasse, P.F., et. al., 2009, “ Bridge Scour and Stream Instability Countermeasures—Experience, Selection, and Design Guidance.” Hydraulic Design Series No. 23, FHWA Publication No. FHWANHI-09-111 & 112. [33]

Strom, K.B.; Kuhns, R.D.; Lucas, H.J.; “Comparison of Automated Image-Based Grain Sizing to Standard Pebble-Count Methods.” ASCE Journal of Hydraulic Engineering, August 2010, pp. 461-473.

Wolman, M.G. (1954), “A Method of Sampling Coarse Bed Material.” American Geophysical Union, Transactions, 35: pp. 951-956.

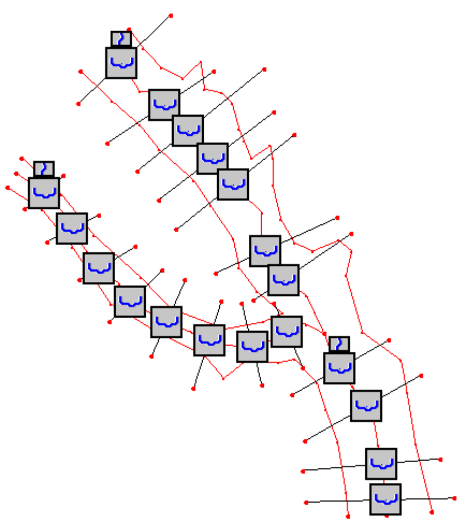
Hunt, J.H., Zerges, S.M., Roberts, B.C., Bergendahl, B., “Culvert Assessment Decision Making Procedures, FHWA-CFL/TD-10-005, September 2010. [34]

Related Topics

- Main WMS page
- Channel Calculator
- Detention Basin Calculator
- Rational Method Rainfall Intensity
- Weir Calculator

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]

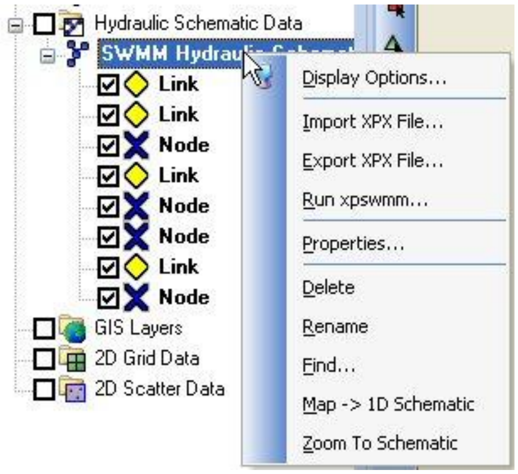
•[1-D Hydraulic Centerline]

•[1-D Hydraulic Cross-sections]

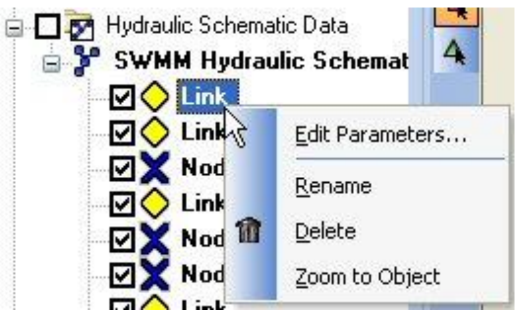
Project Explorer Contents for River Module

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 several options. The available options changes depending on the type of model being used. For example, right-clicking on a SWMM schematic brings up the following menu:



A user 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 making 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.

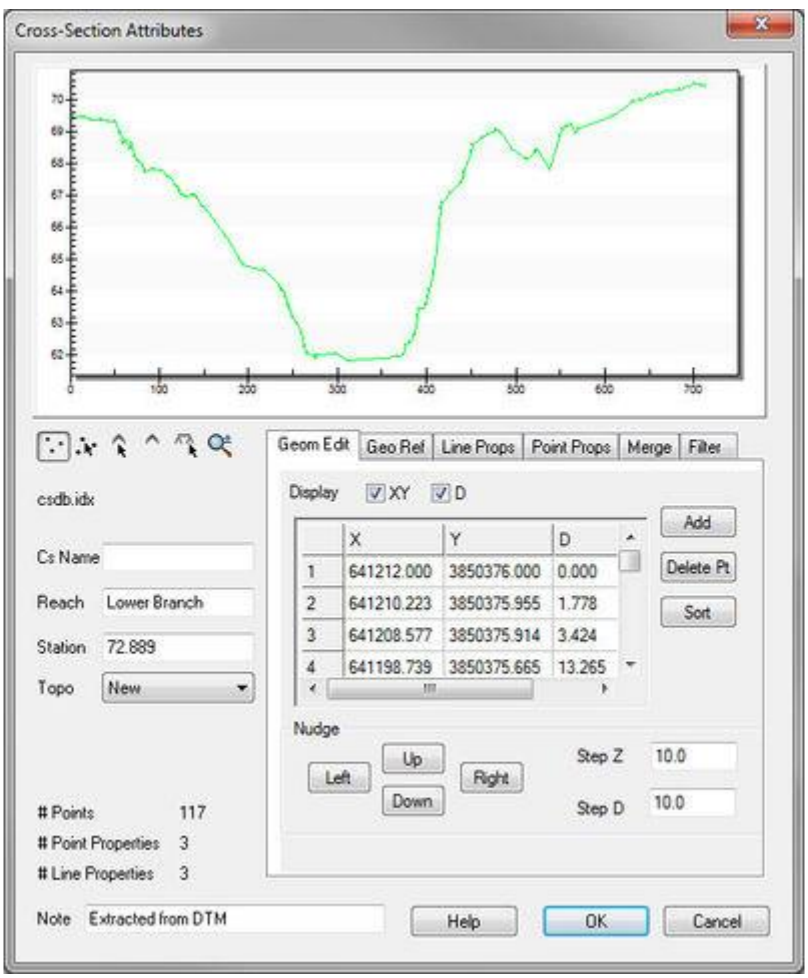
Edit cross sections in one of three ways:

1. 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, it also possible to enter the editor for that cross section.

2. Open a cross section database for editing (or create a new database) using the **Manage Cross Sections** command.

3. 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 Attributes* 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 Attributes* 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 Attributes* 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. WMS can also "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 Attributes* 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 Attributes* editor. Two different cross sections can be merged, with rules for locations and precedence defined in order to create a new cross section.

Filtering

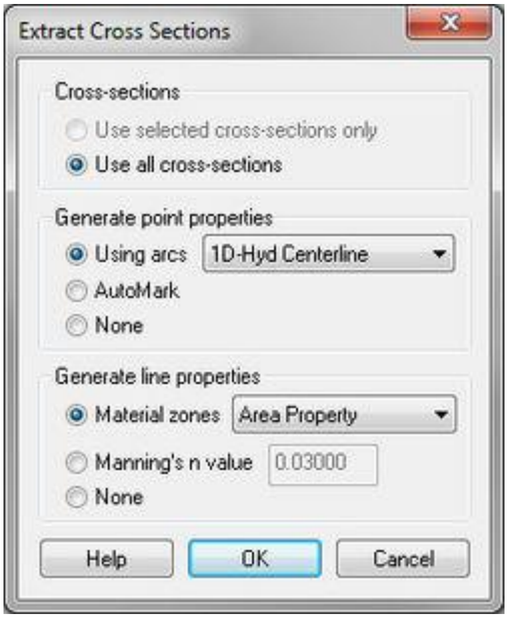
It may be that there are more points defining the cross section than are necessary (or that the hydraulic model is capable of processing). The *Filter* tab in the *Cross Section Attributes* editor allows specifying rules for filtering "insignificant" points along the cross section. This can be particularly 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, a prompt will ask 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

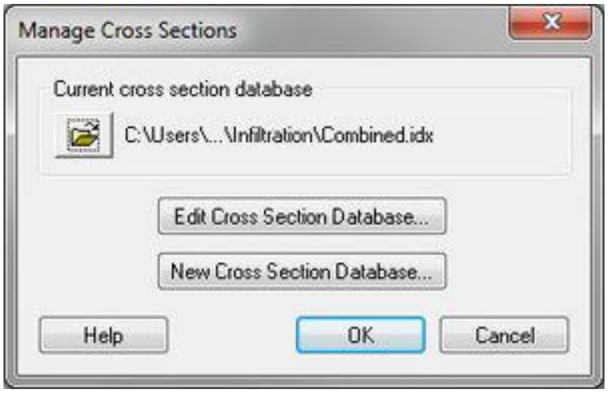
- Editing Cross Sections
- 1D-Hydraulic Centerline
- 1D-Hydraulic Cross-section
- Area Property

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 creating a new database or opening 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.



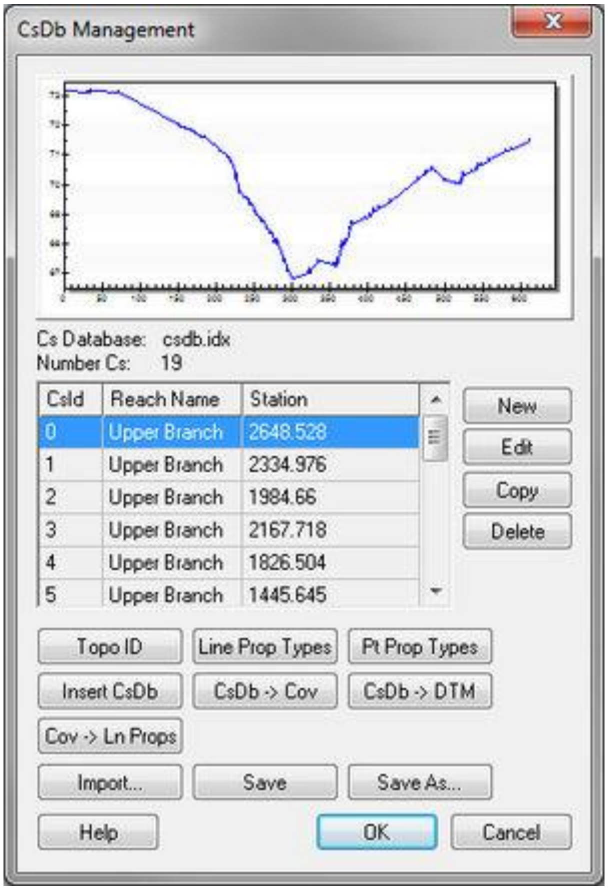
Cross Section Database Definition

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. 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 can also:

•Create a new cross section.

•Edit, copy, or delete an existing cross section.

•Insert an entire database or merge databases together,

•Convert a cross section database to a coverage. 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.

•Converting the coverage to line properties.

Related Topics

•Edit Cross Section Database

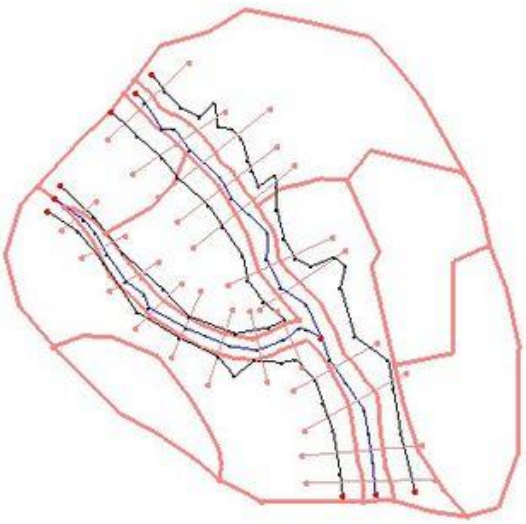
•Extract Cross Sections

•River Tools Overview (Hydraulic Modeling in WMS)

River Tools

The *River Tools* 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 allows interpolating 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

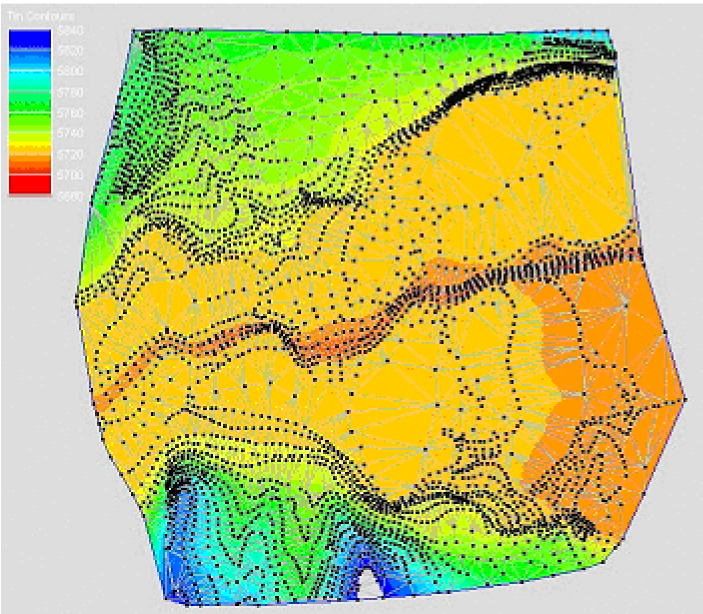
5.5.a. Flood Plain Delineation

Overview of Flood Plain Delineation

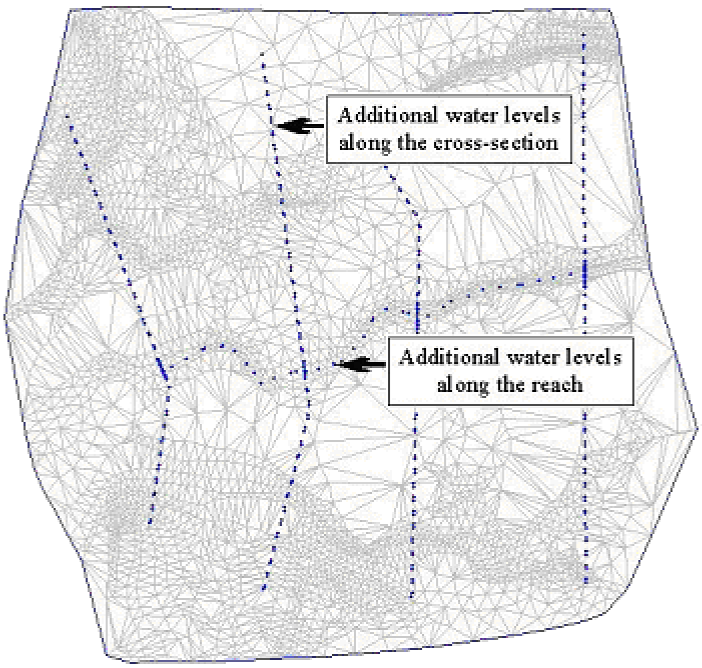
In addition to stream network and drainage basin delineation, WMS can also be used to perform floodplain delineation. Water levels simulated by a river hydraulic model or collected from different sources are read from a text file as a scatter dataset (see preparing stage data for more help). A smooth 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:

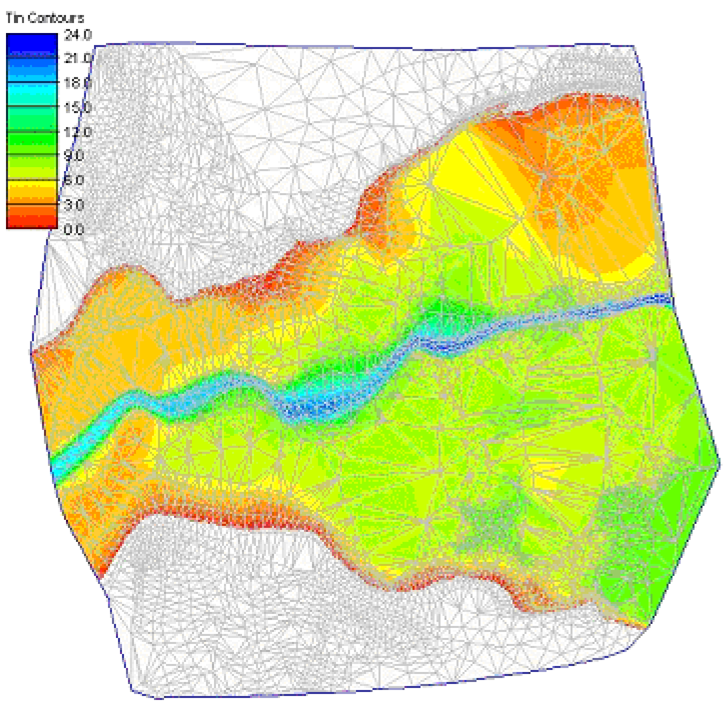
- Prepare a triangulated irregular network (TIN) surface of the area where the delineation is to be performed. This can be done by reading scattered elevation, converting from a DEM, or digitizing a contour map.



- Prepare the 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 are varied stochastically within a range of valid results.

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, also notice the following changes while using the new method:

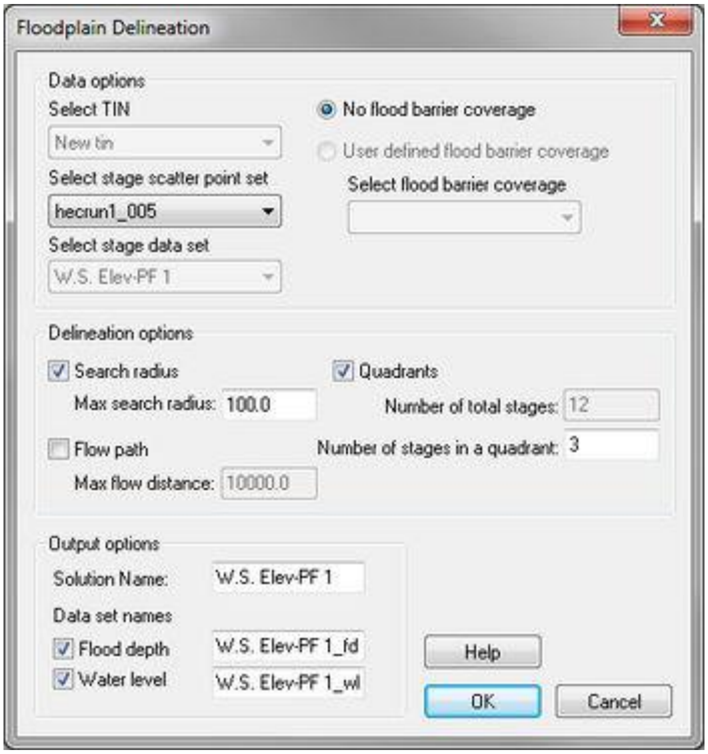
- Water levels are read as a scatter dataset as opposed to flood stages at TIN vertices.
- The method does not require “streams” in the TIN.
- Multiple events or water level time series can be read as oppose to a single event. Choose an event while delineating floodplain.
- Specify flood barriers as 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

- Delineate Flood Plain
- Preparing Stage Data
- Stochastic Modeling
- Hydraulic Modeling
- Interpolation Options for Floodplain Delineation
- Simplified Dam-Break Analysis
- Flood Barrier

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

When working with a single TIN this option will be dimmed out. However, it is possible to have more than one TIN in WMS. In such cases, 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 incorporating 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.

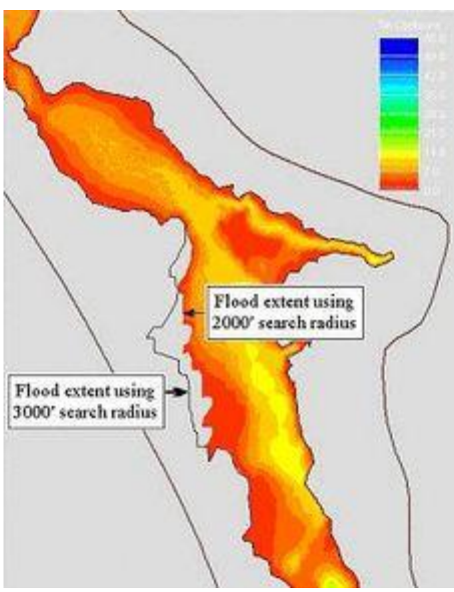
In order to delineate the floodplain properly, these barriers must be considered in a hydraulic model as well as in floodplain delineation process. This option not only incorporates such existing barriers in the process, it also provides the flexibility to professionals so that they can consider proposed 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.



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, it's necessary 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
- 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 *File Import 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 take advantage of the centerline and cross sections to develop a larger dataset for flood plain delineation. For example if having created a 1D-Hydraulic centerline to represent the stream, and optionally cross sections, it's possible to 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
- File Import Wizard
- Interpolating Hydraulic Model Results
- Floodplain Delineation
- Channel Calculator

5.6. 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 handling large files more efficiently when creating hydrologic models. For example the GIS module allows importing large files then selecting and converting 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 the GIS data in this fashion is more efficient (the equivalent GIS functionality is to clip out just the parts of the data necessary 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).

The GIS Data module is included will all paid editions of WMS.

Using the GIS Module with a License of ArcView®

WMS uses the ArcObjects to incorporate much of the ArcMap functionality directly. Open any file format (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) that is supported by ArcView® and 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 with licenses of ArcView® is available without a license. The primary differences are that only layers that are in the shapefile format can be imported, 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

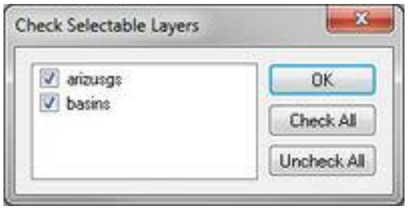
GIS Tools

The toolbar for the WMS GIS module has a variety of tools useful for selecting objects and obtaining information about them. The tools are described below.

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 checking or unchecking layers as being selectable.



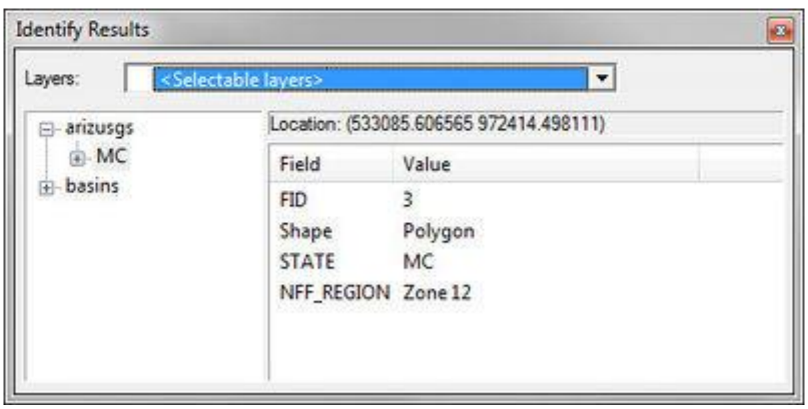
Individual features can be selected to create a new selection, added to the current selection, taken from the current selection, or selected from within features already selected as specified by the Interactive Selection method defined from the *Selection* menu (by default the selection method is create a new selection).

Multiple features may also be selected by dragging a box around all features that should be selected. In such cases all features that have any portion within the selection rectangle will be selected.

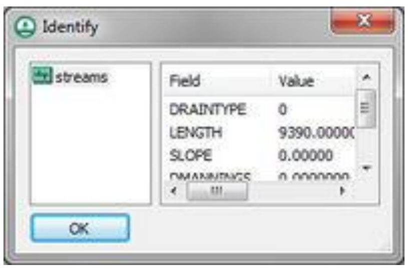
If ArcObjects® is not enabled, the icon will appear in its inactive state:

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, when holding the *SHIFT* key down, newly selected shapes can 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 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 the hydrologic modeling projects. When selecting 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 there is a current license to ArcMap that includes ArcObjects and ArcObjects have been enabled, 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 accessing 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:

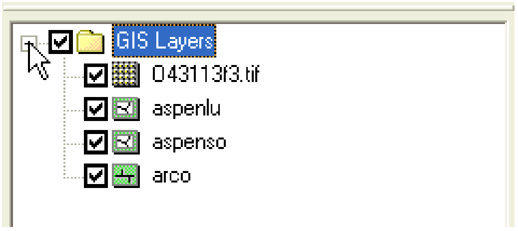
Description	Icon	GIS Data Type	Functionality	Licensing Requirements
WMS Point Shapefiles		GIS Vector Data	•Remove/Rename/Zoom To Layer •Convert to Map Module feature objects •View object and entire shapefile attributes	WMS License with Map Module

			•Join .dbf file with the vector layer using a common ID •Convert to Vector GIS Data (GlobalMapper)	
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

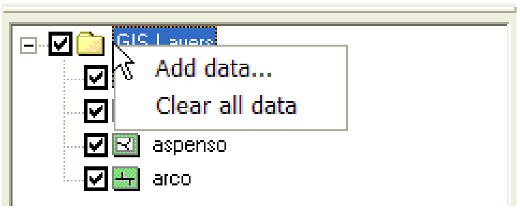
Generic GIS Raster Image		GIS Raster Data	•Remove/Rename/Set Transparency/Zoom To Extents/Open Containing Folder •Export to various supported raster formats •Automatic reprojection based on current projection •Export an image world file •2 or 3-point registration based on pixel/real-world coordinates •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	WMS License with Map Module

			•Dynamic layer-runs in a separate thread and changes with current view	
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	WMS License with Map Module, Current

			•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	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

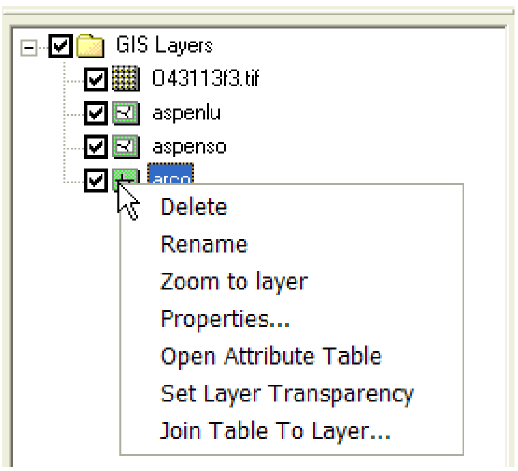


Right-clicking on the main GIS Layers folder allows adding the various types of GIS data (open new GIS layers) or clear all data. If ArcObjects is enabled WMS cannot read the GlobalMapper GIS data types. Right-click on the GIS Data folder and clear all the GIS data to disable ArcObjects and read GlobalMapper GIS data using the **Add GIS Data** button or menu. With ArcObjects enabled, access exists 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 has commands 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 there is an option to **Join Table to Layer** . If the layer is a grid, an options exist to **Write DEM ASCII File** or **Convert to DEM** .

The properties are the identical properties available from ArcGIS.



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

- GIS Module
- GIS Tables
- Coordinate Conversions

GIS Tables

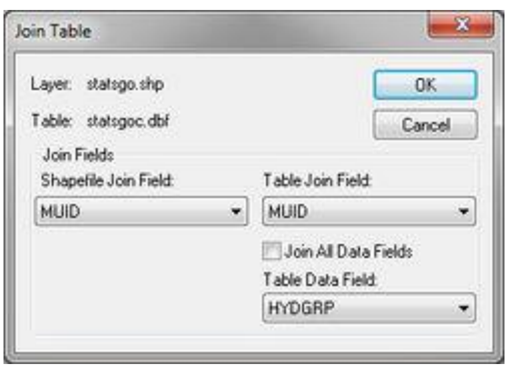
Open Attribute Table

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 joining 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. 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, a prompt will ask for the database file to join using the standard select file dialog. The *Join Table* dialog will then appear and ask to select the *Join Field* from the GIS data layer attributes and the *Join Field* from the table to be joined 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, 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 the 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

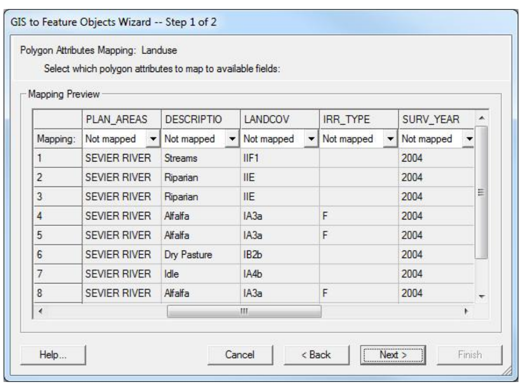
- Feature Objects
- Feature Object Guidelines
- Drainage Coverage
- Shapefiles

Mapping to Feature Objects

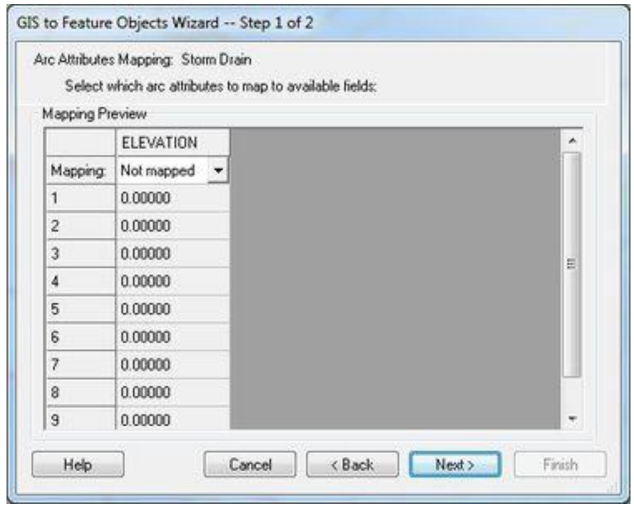
While future versions of WMS may be able to process some commands directly from the GIS data layers, currently map all features to use as part of model development to feature objects in a map coverage. One way to do this is to import an entire shapefile directly to map coverage (this is the only way 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 the study area and map those to feature objects in a map coverage.

A mapping wizard guides through the process of converting the GIS data layer features to feature objects in a map coverage. Before beginning the mapping process, first go to the map module and make sure that the currently active coverage is the coverage to map GIS data layer features to. For example, the default coverage in WMS is a drainage coverage and so, if about to map soil polygons, create a new coverage and make sure that it has an attribute set of Soil Type. After making sure WMS will be mapping to the correct coverage, select the polygons which overlay the study area to be mapped. (This is done with the selection tool(s) in the GIS module) If wanting to map all features, choose the **Select All** command from the *Edit Window* , or just move to the **Mapping** command (a prompt will ask if wanting to convert all features since none are selected).

If ArcObjects are enabled, the **ArcObjects→Feature Objects** command is activated. If ArcObjects is not enabled, the **Shapes→Feature Objects** command is activated. After choosing the appropriate mapping command the mapping wizard will appear as shown below. This wizard will guides 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

5.6.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 accesses some of the same functionality in WMS that is available in ArcView®, but this is only providing WMS is 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

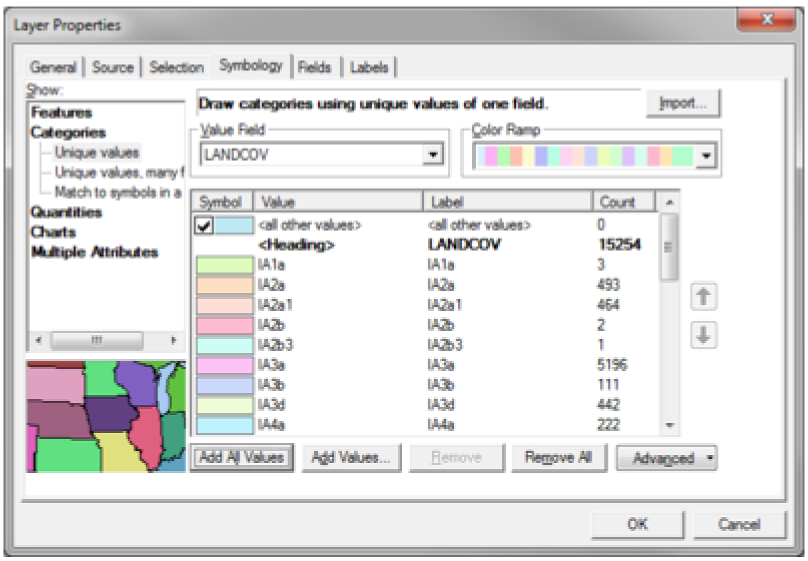
The *Data* | **Enable ArcObjects** command queries the ESRI license manager for ArcView/ArcGIS® to see if a license for version 8.x exists. If a valid license is found then the ArcView® functionality within WMS is enabled and access is allowed. If a license is not found then the ArcView® functions remain dimmed, but WMS is 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 ArcGIS®, the display capability of shapefile layers is minimal. The primary intent of using the shapefile layers is as a temporary holding place to convert the data to feature objects. As such, the full symbology capability of ArcGIS® has not been implemented within WMS. All shapes are displayed using black lines.

ArcObjects Display Options

With ArcObjects enabled, there is access to the ArcGIS® *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 ArcGIS®.

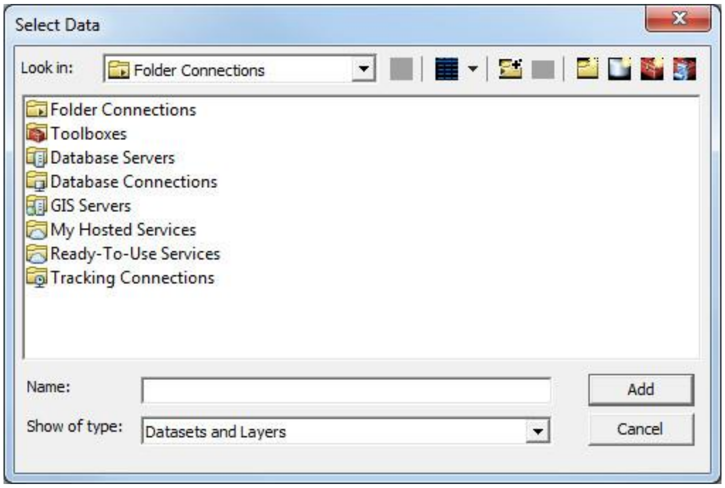
Transparency can be defined for a layer, but it will only affect the display that is rendered by ArcObjects, and not the rest of the data rendered by WMS.

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 ArcGIS®.



When ArcObjects® is enabled, it is possible to load any of the ESRI supported formats, including shapefiles, coverages, geodatabases, grids, images, CAD files and other formats, 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.

Shapefiles

There are nearly as many different ways to have stored watershed data in a GIS as there are watersheds stored. Therefore, what is done 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 shapefile 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 shapefile 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 the data is stored in the GIS, take advantage of as much of the predefined and stored hydrologically related parameters as possible.

Ideally the project 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 these three layers have been properly defined, import them and automatically create the topologic model used for hydrologic modeling in WMS. If one or more of these layers are not present, either create it in ArcView® or ARC/INFO® or define it in WMS after importing what is available. For example, if there is only a layer defining basin boundaries then 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 node (or from node) in an arc be the downstream node, while the second node (or to node) is the most upstream node. If the 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 to package the 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 having 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

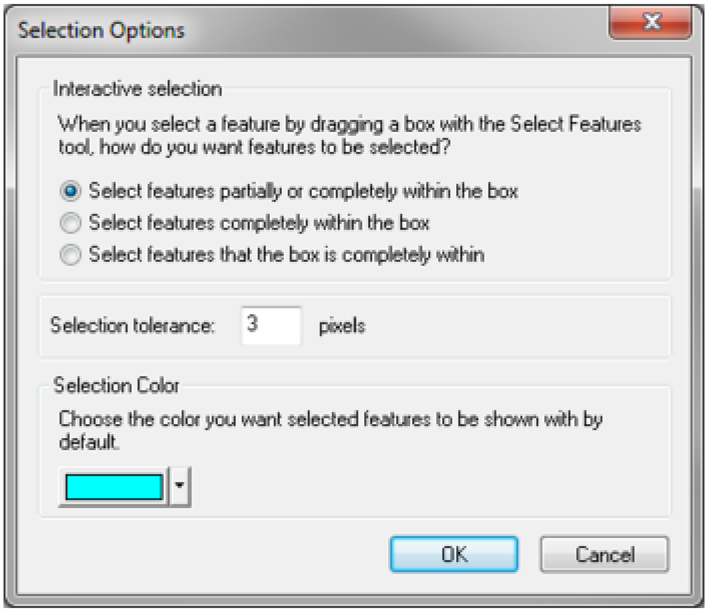
Feature Selection

With ArcObjects enabled three different selection methods are available. However, before attempting to select layers, make sure that the layer(s) to be used are selectable. By default a layer is selectable when opened, but the status can be changed to unselectable using the **Selectable Layers** command in the *Selection* menu.

Graphical Selection

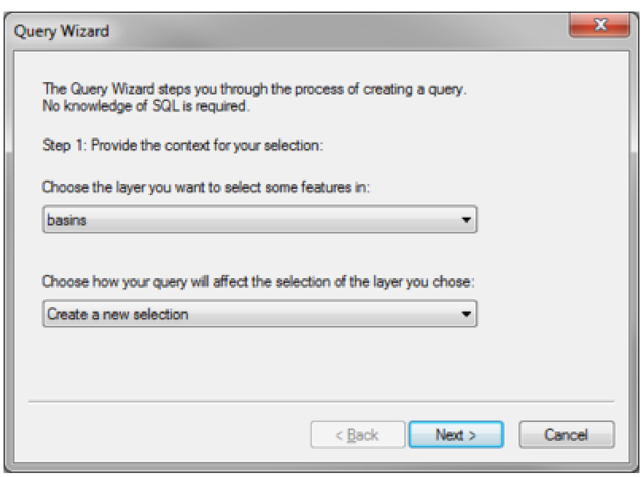
The **Select Features** tool can be used to interactively select features from GIS layers. When selecting using graphical selection all features from all selectable layers will be added to the selection list. A rectangular box can be dragged around a region of interest and then all features from all selectable layers that have in part within the rectangle will be added to the selection list.

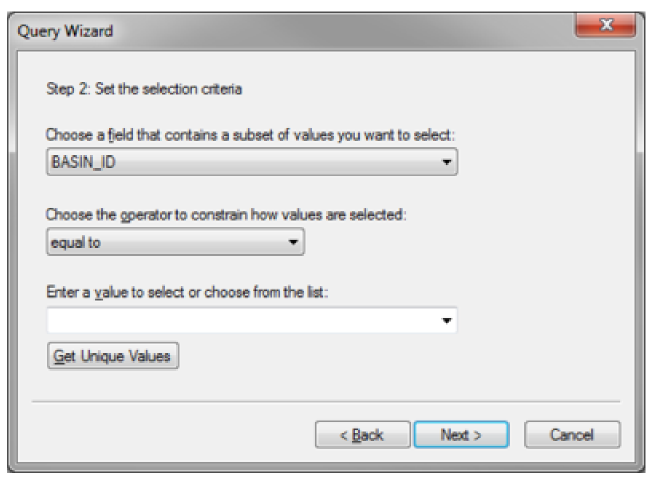
The **Options** command is used to specify how features are selected and the way they are displayed.

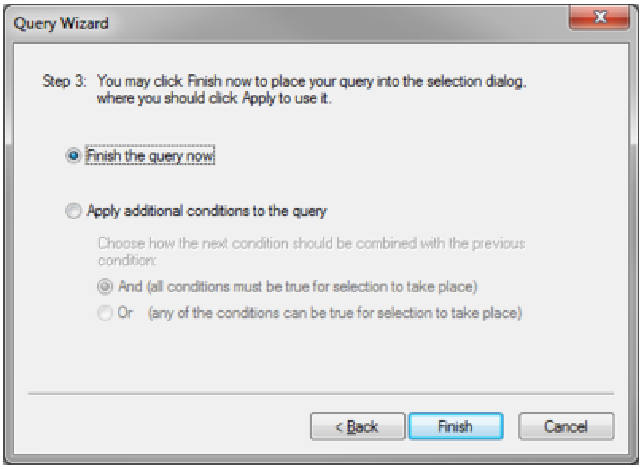


Selection by Attribute

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 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.

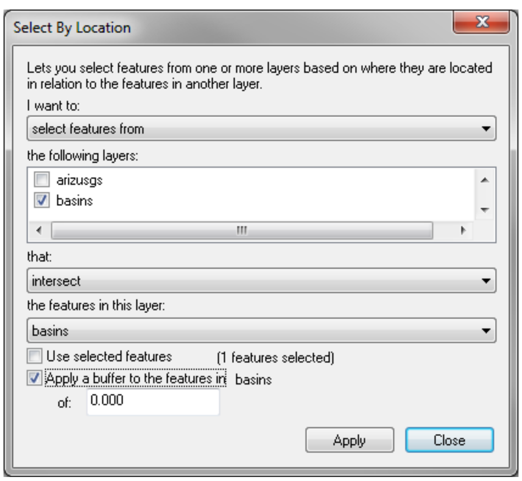






Selection by Location

Select features from one or more GIS layers based on their spatial relationship to another layer. The *Select By Location* dialog allows specifying whether to select new features, add to currently selected features, select from selected features, or delete from selected features. In addition, it specifies the layers to be selected from as well as the relationship with the selection layer. Finally, specify the layer used for selection and whether to 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 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 loading 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.

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. Then select only the polygons to convert to a coverage and map them. In this way WMS will only be building topology for the selected polygons.

Cleaning Imported Shapefile Data

If intending to use the data from the shape file in more than one session, save it as a WMS map file after importing/mapping the first time. Further, after importing the shape files, 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 5=Drainage outlet point 29=Route 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	Link id
Free flow coefficient	ffcoeff	Free flow coefficient
Crest width	crestwid	Crest width
Crest elevation	crestelev	Crest elevation
Flooded coefficient	floodcoef	Flooded coefficient
Gage depth	gagedepth	Rain gage depth for gage coverage
SWMM name	swmmname	SWMM node name
SWMM ponding type	swmmpondtype	SWMM ponding type options: "None" "Allowed" "Sealed" "Link Spill Crest to 2D" "Link Invert to 2D"
SWMM constant inflow	swmmconstinflow	SWMM node constant inflow
SWMM ground elevation	swmmgroundelev	SWMM node ground elevation
SWMM invert elevation	swmminvertelev	SWMM node invert elevation
Z coordinate	elevation	This changes the Z-coordinate of the point
Arc Attributes		
Drainage arc type *	draintype	0 = Generic 2 = Ridge 3 = Stream 32 = Lake 44 = Pipe
GSSHA arc type	casctype	GSSHA arc type: 0 = Generic 1 = Trapezoidal channel 3 = General stream 8 = Cross section channel 12 = Trapezoidal channel (Groundwater process enabled) 13 = Cross section channel (Groundwater process enabled) 30 = Trapezoidal channel (Sediment process

		enabled) 31 = Trapezoidal channel (Sediment and groundwater processes enabled) 42 = Embankment 45 = Pipe
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
SWMM name	swmmname	SWMM link name
SWMM shape	swmmshape	SWMM link shape options: "Circular" "Rectangular" "User defined" "Special" "Trapezoidal" "Power Fn." "Natural"
SWMM diameter	swmmdiameter	SWMM link height/diameter
SWMM length	swmmlength	SWMM link length
SWMM upstream invert	swmmupinvert	SWMM link upstream invert elevation
SWMM downstream invert	swmmdninvert	SWMM link downstream invert 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 browsing for and open shapefiles as GIS layers in WMS. Without a license of ArcView® on the 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

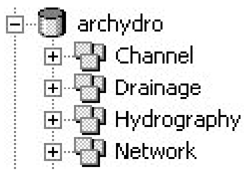
•Enabling ArcObjects

•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 the project. For more information, see Maidment (2002). These data are stored in four different feature datasets within the geodatabase.



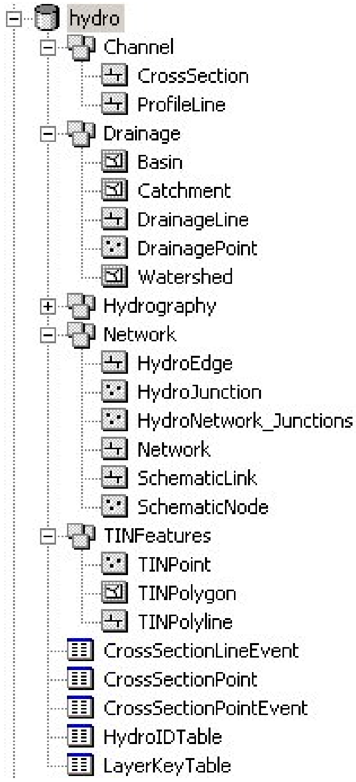
In order to create an ArcHydro geodatabase from WMS, first switch to the GIS module and enable ArcObjects. WMS uses the feature objects stored in each coverage to build the geodatabase.

Set up and build feature objects to add to the geodatabase, and define a coordinate system for the 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 there is 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 the data and spatial reference are defined, select **Feature Objects → ArcHydro Geodatabase** from the *Mapping* menu in the GIS module. At this point, a prompt will ask if wanting to create a raster grid from the DEM in WMS, if if there is one loaded. Then a prompt will ask whether to create a feature dataset consisting of a TIN boundary, TIN breaklines, and TIN vertices if there is a TIN in memory. WMS will build the ArcHydro geodatabase based on the coverages and data created. WMS will then create the raster grid and TIN dataset if there is this data in the WMS project and elected to build them.

Once WMS has finished building the geodatabase, 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 the 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

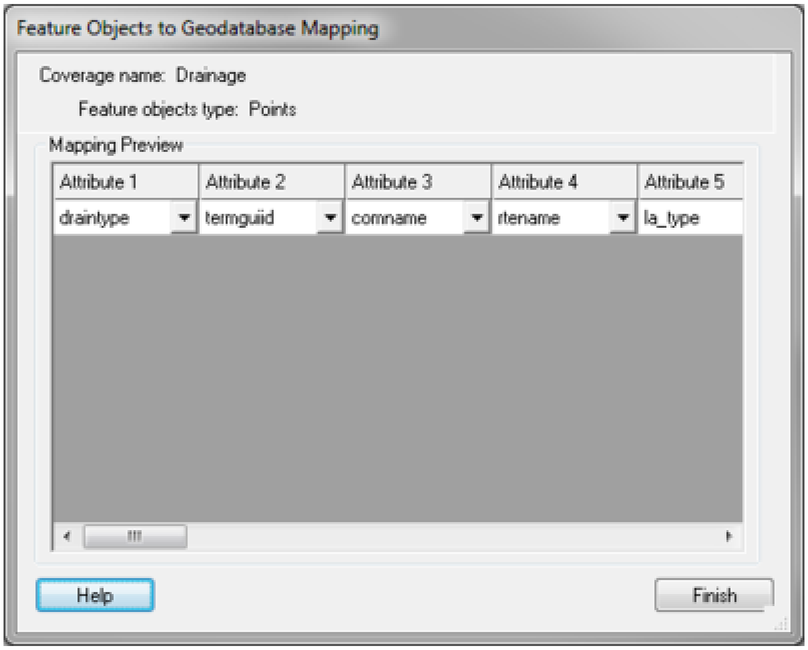
A geodatabase is a geographic database used to store GIS information, such as features and rasters. In addition to storing data, a geodatabase allows setting up relationships between data and create rules to validate data. Geodatabases are used in ArcGIS products.

In order to create a geodatabase from WMS, first switch to the GIS module and enable ArcObjects . WMS uses the feature objects stored in each coverage to build the geodatabase.

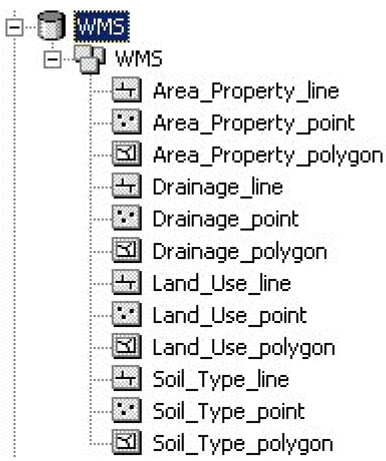
Set up and build feature objects to add to the geodatabase and define a coordinate system for the project. Vector data are stored in feature datasets and feature classes in the geodatabase which require a common spatial reference. The spatial reference in the geodatabase is defined from the current coordinate system in WMS.

Once the data and spatial reference are defined, select **Feature Objects → Geodatabase** from the *Mapping* menu in the GIS module. At this point, a prompt will ask whether to specify which coverage attributes will be mapped to the geodatabase. An option exists to specify which WMS coverage attributes to map over, or to let WMS map all of the appropriate attributes.

If choosing to map the WMS coverage attributes manually, a prompt will ask for each feature class created in the new geodatabase, which can take a while. If choosing to let WMS map the attributes, no prompt will appear and WMS will build the geodatabase automatically.



Once WMS has finished building the geodatabase, 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

5.7. 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, it is possible to 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.

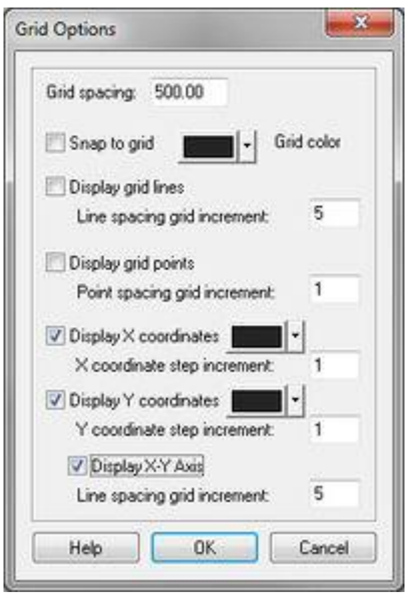
The 2D Grid module can be added to a paid edition of WMS.

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 dataset brings up a list of options that are available for that data type. In the 2D Grid module, when a dataset 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
- 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 dataset 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.

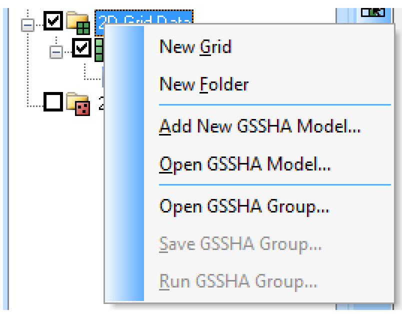
Cell filling the grid has been an option since WMS 6.1. However, WMS 7.0 allows much more flexibility in visualizing the output and input to GSSHA through cell filling by making it into part of the contour options. Simply ensure that the cell filled radio button is selected, and each cell will be filled 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.

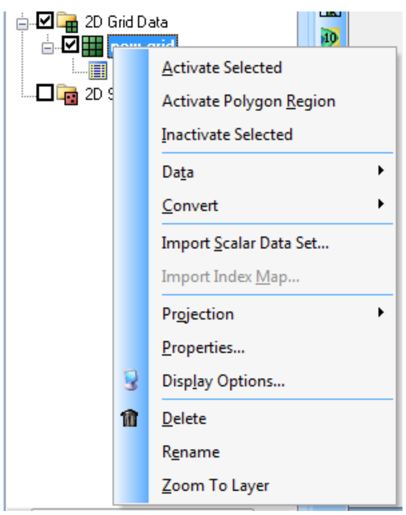
The Project Explorer allows access to individual contour options for each dataset. When a dataset is read in it is given the default set of contour options that can then be modified by right-clicking on the dataset in the Project Explorer. Whichever dataset is active is the only dataset contoured. The 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.

Right-Click Options

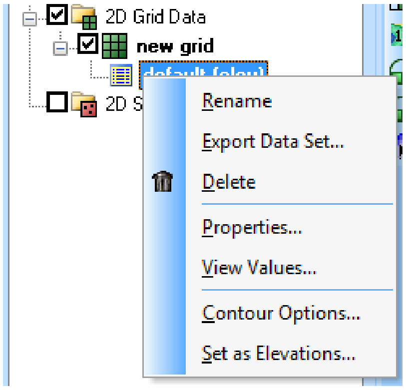
In the 2D Grid module, the grid and associated datasets are listed in the Project Explorer (WMS only allows working with one grid).

Right-clicking on the main 2D Grid Data folder has options to create a **New Grid** , **New folder** , or GSSHA Model options.





Right-clicking on the dataset for a grid has options to **Export** , **Rename** , view **Properties** , **View Values** , set **Contour Options** , and **Set as Elevations** .



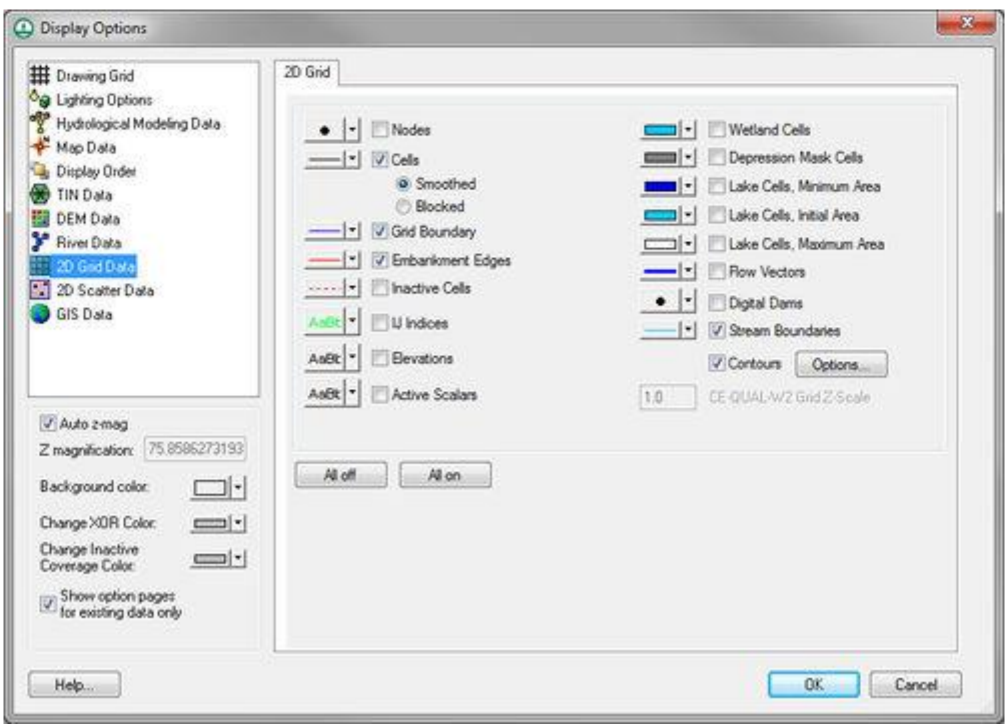
Related Topics

•[GSSHA Maps]

•[Project Explorer Overview]

•[Coordinate Conversions]

2D Grid Display Options



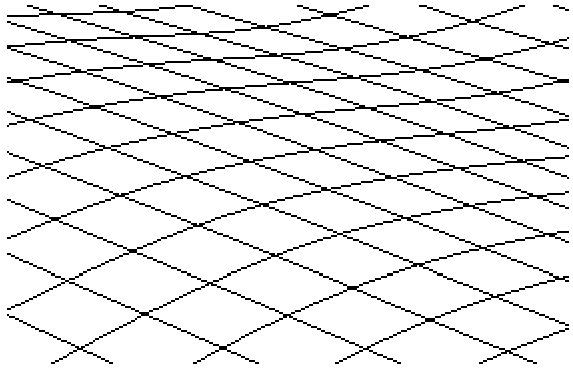
The display options control which components of the grid are displayed. The display options can be set by selecting the **Display Options** command in the *Display* menu. Most of the items in the dialog box are toggle boxes. If the toggle for a component of the grid is set, the component is displayed 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:

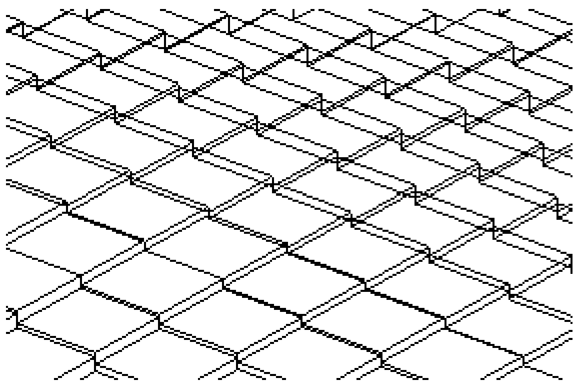
•The **Nodes** item is used to display grid nodes. Since all grids in WMS 7.0 are cell-centered, a dot is displayed at the cell centers.

•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.



300 px *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.

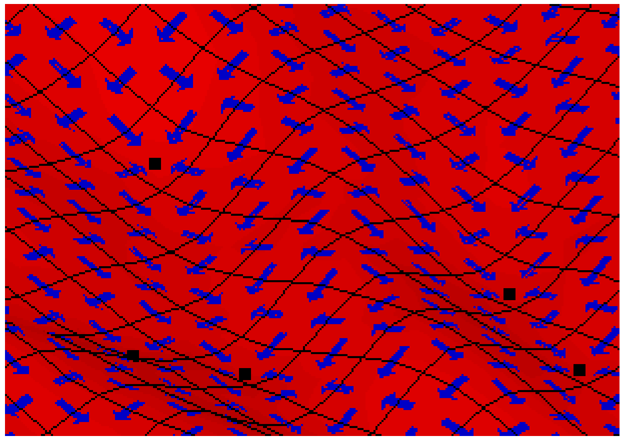
•The **Contours** item is used to display contours computed using the active scalar dataset.

- 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 toolbar for the WMS 2D Grid module has a variety of tools useful for editing and creating objects within that module. The tools are described below.

Select Grid 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 Grid Row

The **Select Grid Row**  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 Grid Column

The **Select Grid Column** 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.

Place Contour Labels

The **Place Contour Labels** 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.

Create Gages

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 a prompt will ask 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 .

Select Gages

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.

Select Hydrograph

The **Select Hydrograph** tool allows selecting 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 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

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 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 to specify a polygon file. If the *Select polygon interactively* option is selected, WMS prompts 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

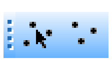
5.8. 2D Scatter Point Module

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 Scatter Point** tool. The most common use of scattered data are for creating water surface elevations to compute a floodplain delineation. Generally, in such cases, take advantage of a centerline and cross sections to develop a larger dataset for flood plain delineation.

2D Scatter Point Tools



The toolbar for the WMS 2D Scatter Point module has a variety of tools useful for editing and creating objects within that module. The tools are described below.

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* .

Create Scatter Data Point

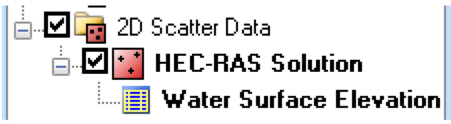
The **Create Scatter Data Point** tool is used to create new scatter points. The primary purpose of creating scatter points is to enter water surface elevations that can be used in a flood plain delineation. It can only be used in plan view.

Related Topics

- Interpolation
- Preparing Stages for Floodplain Delineation
- Interpolating Results Along a Centerline and Cross-sections
- Terrain Data Module

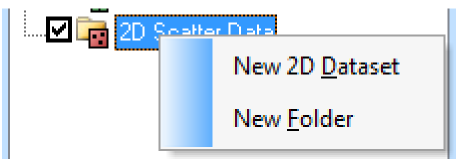
Project Explorer Contents for 2D Scatter Module

In the 2D Scatter Point 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.

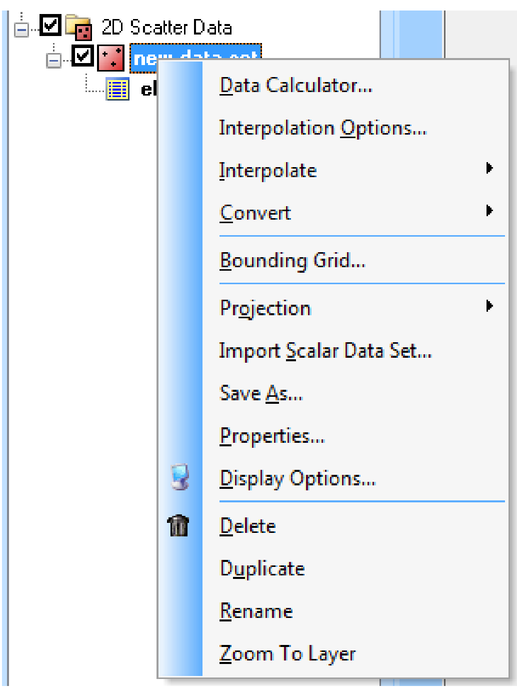


The active data is determined by selecting it from the Project Explorer.

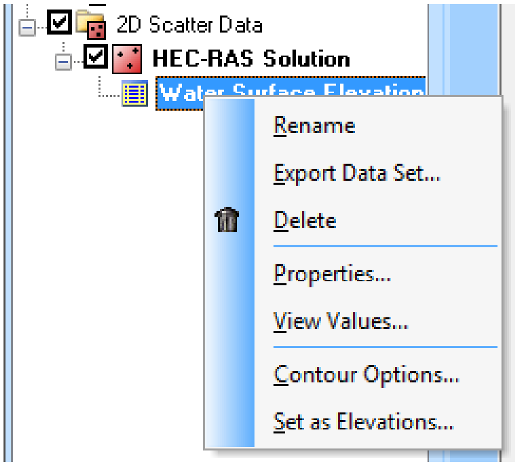
Right-clicking on the main 2D Scatter Data folderhas options to create a **New 2D dataset** or a **New** folder to organize scatter datasets in.



Right-clicking on a scattered dataset has options 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 has options 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 dialogs.

- Hydrologic Modeling

Related Topics

- Hydraulic Modeling
- Hydrologic Modeling

Models Available in WMS

- CE-QUAL-W2
- GSSHA
- HEC-1
- HEC-HMS
- HEC-RAS
- HSPF
- HY-12
- MODRAT
- NSS (was NFF)
- OC Hydrograph
- OC Rational
- Rational
- River Tools

- Storm Drain and 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.

The model selection menu

•The Models available in the Map Module are:

- CE-QUAL-W2
- GSSHA
- Storm Drain
- Storm Drain-FHWA
- River Tools

•The Models available in the Hydrologic Modeling Module are:

- HEC-1
- HEC-HMS
- TR-20
- TR-55
- NSS
- Rational
- MODRAT
- OC Hydrograph
- OC Rational
- HSPF
- SWMM

•The Models available in the River/Hydraulic Module are:

- HEC-RAS
- SMPDBK
- SWMM

- HY-12

How to select a model

Do I want to do hydrology or hydraulics?

1. Hydraulics:
   a) Do I want to do a simple dam break simulation?
      - SMPDBK (Simplified Dam Break)

   b) Do I want to do floodplain delineation, hydraulics, or a more complicated dam break simulation?
      - HEC-RAS

   c) Do I want to model storm drains?
      1. Do I need additional capability only available in xpswmm such as FHWA hydraulic, etc.?
         - xpswmm
      2. Do I not need that additional capability?
         - EPA-SWMM
2. Hydrology:
   a) Am I doing a small urban model?
      - TR-55
      - Rational
      - MODRAT (LA County)
      - OC Rational (Orange County, CA)
      - NSS Urban (Statistical model)
      - HEC-1 (OC Hydrograph)
      - HEC-HMS
      - SWMM
      - TR-20

   b) Am I doing a rural model?
      - NSS (Statistical model)
      - HEC-1 (OC Hydrograph)
      - HEC-HMS
      - SWMM
      - TR-20

   c) Am I interested in water quality, sediment transport, or the effects of wetlands on a hydrograph
      - GSSHA

- HSPF (Water Quality only)

Related Topics

•Models

6.1. CE-QUAL-W2

CE-QUAL-W2

WMS is a very effective aid for setting up the CE-QUAL-W2 bathymetry and control files. The CE-QUAL-W2 model can be added to a paid edition of WMS.

Creating and Running a CE-QUAL-W2 Model using WMS

In order to create the bathymetry file, 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, create a branch coverage then trace an outline of the water body using the **Create Feature Arc** tool. Then can divide the water body into separate branches accordingly.

Next, the branch coverage is duplicated and renamed as segment coverage. Segments are created using the **Create Feature Arc** tool accordingly.

From here appropriately number 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 create a storage-capacity curve. Options exist for plotting this curve as well as exporting the data that created this curve to 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 refer to Creating a CE-QUAL-W2 bathymetry file from a TIN .

If requiring the WMS to generate a storage capacity curve to be very accurate when compared to a given storage capacity curve then it is recommended that to 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 then proceed to make a detailed bathymetry representation of the model.

With a completed bathymetry file, proceed to begin editing the control file. In the *Job Control* dialog 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 there is knowledge or experience that would dictate to 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 to 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 project. 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 to compare the 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. If none of this works there is a forum that can be used to contact the developers or ask questions of other users (w2forum.cee.pdx.edu/).

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, some amount of calibration is required for the model to resemble the observed data. The first calibration that should be performed is a water balance. Then look at calibrating the temperature. After a thermal calibration has been reached, proceed to calibrate any constituents that are modeled. For help on calibration, refer to the CE-QUAL-W2 user's manual [35] 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

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 creates the input files that are used in the CE-QUAL-W2 model. 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, the simulation should have been defined using WMS. If the simulation was not defined using WMS, define branch and segment coverages representing the branches and segments 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 reading 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.

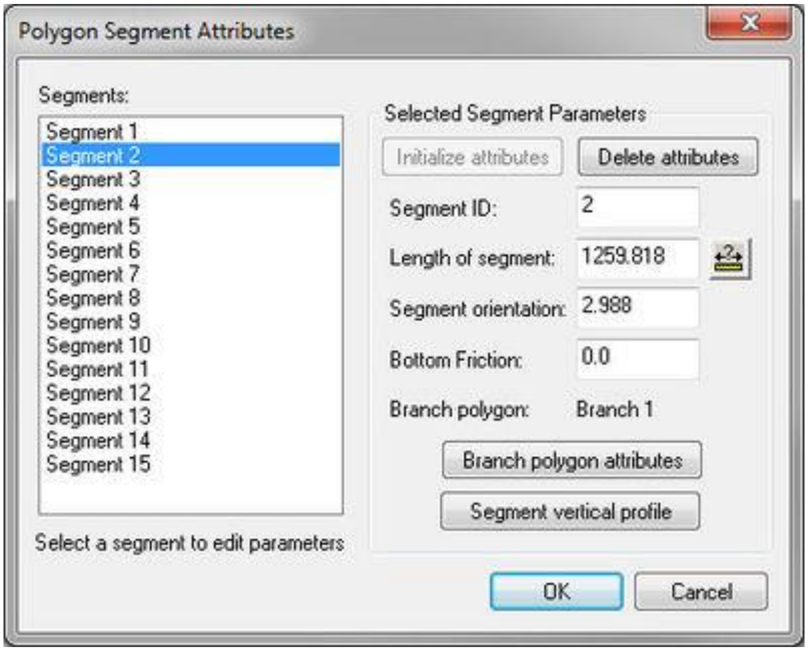
This menu item can be used to run a CE-QUAL-W2 simulation from the WMS model wrapper given the control filename. If the model runs to completion, the results can be read and analyzed in WMS using the **Read Solution** menu item.

Branches

This feature allows editing the features of a branch. See CE-QUAL-W2 Branches for more information.

Segments

The **Segments** command in the *CE-QUAL-W2* menu allows editing 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 viewing and editing 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 where it is possible to 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. Refer to the segment tab of a branch polygon editor to see the segments that are assigned to a branch.

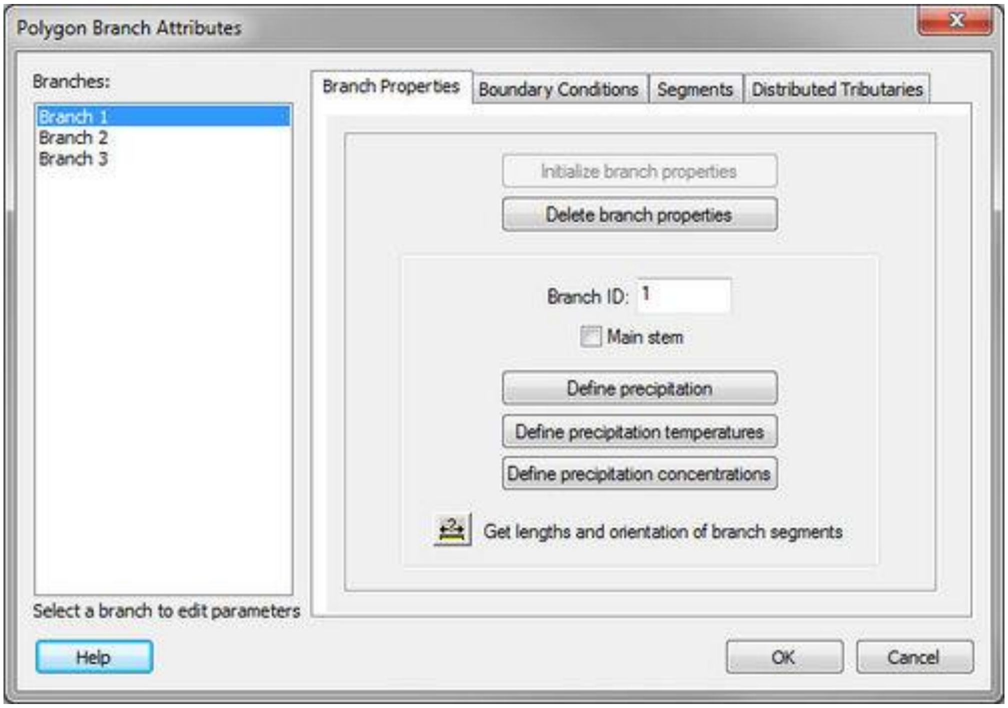
Related Topics

•CE-QUAL-W2

•CE-QUAL-W2 Branches

CE-QUAL-W2 Branches

This feature allows editing 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 to 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 tracing 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 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 specify any inflows and the temperature and constituent concentration of the inflow.

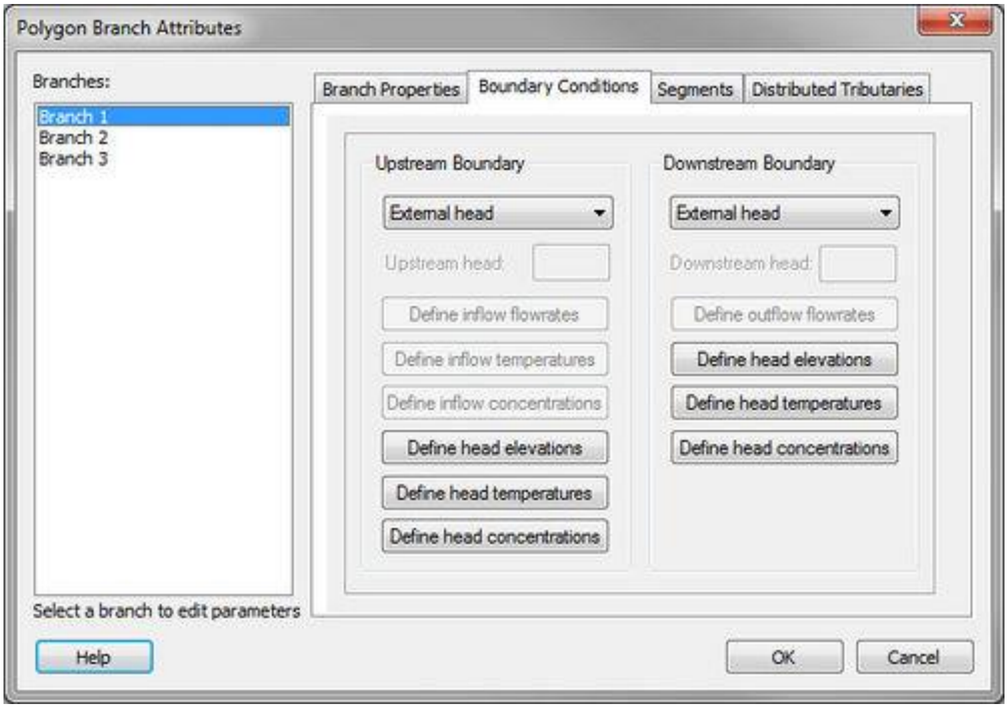
Selecting *internal head* specifies that the branch is attached to another branch. Nothing else is required when this option is selected.

The downstream boundary condition has three options to choose from: external head, external flow and internal head.

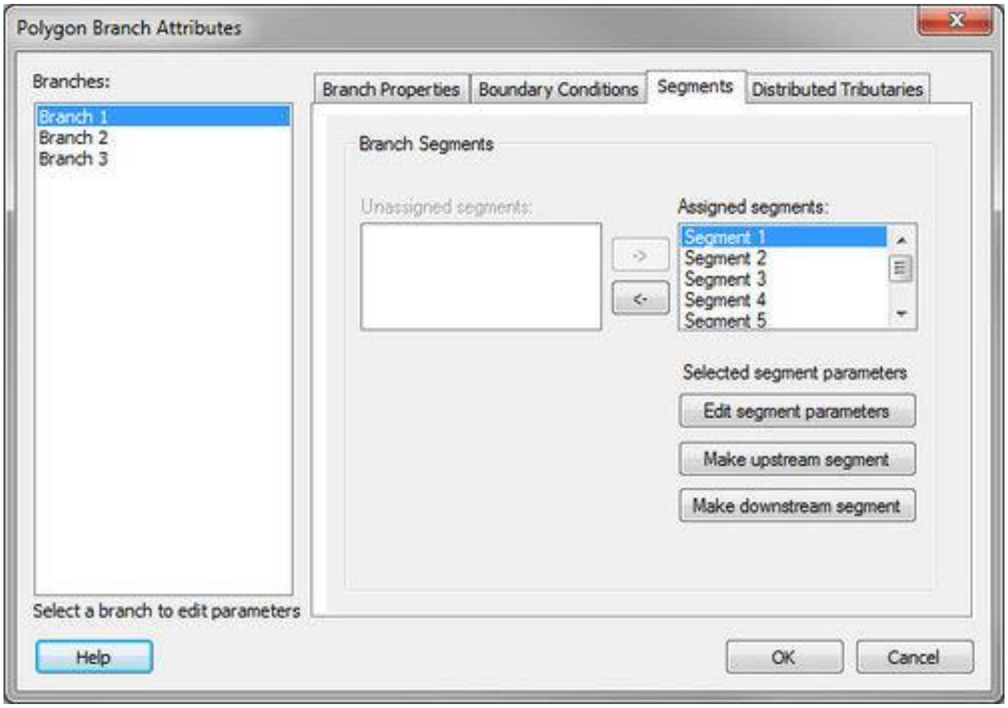
When *external head* is selected the **Define head elevations** , **Define head temperatures** , and **Define head concentrations** buttons become active. Selecting these buttons opens dialogs that specify the elevations, temperatures, and concentrations of the external head.

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.

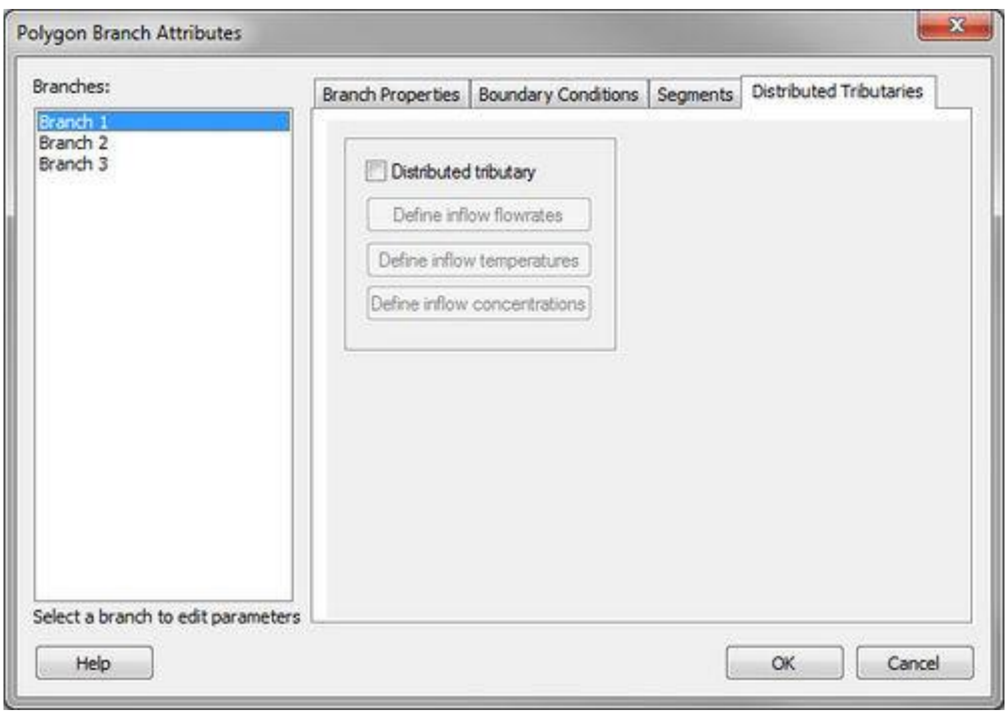
Selecting *internal head* specifies that the branch is attached to another branch. Nothing else is required when this option is selected.



In the *segment* tab of the *branch attributes* dialog (shown below), 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, check the box which then activates the **Define inflow flowrates** , **Define inflow temperatures** , and **Define inflow concentrations** buttons. These buttons open dialogs to 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

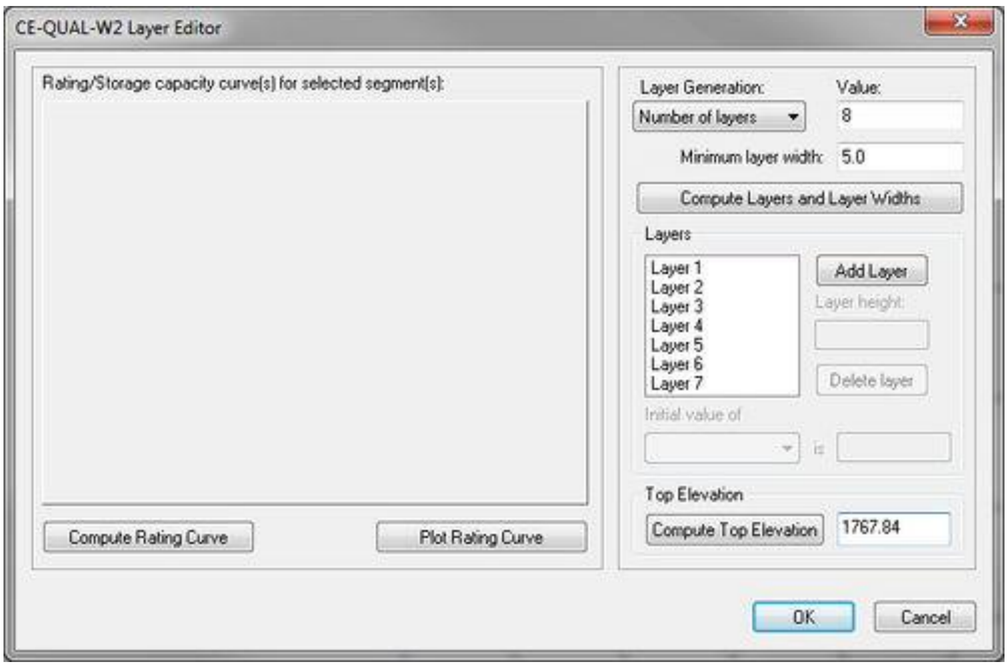
The layer editor is used for generating a storage capacity curve for a CE-QUAL-W2 application. The curve is computed from selected segment(s) of a representative waterbody and is generated by selecting the **Compute storage-capacity curve** button. Generation of the curve may take some time, especially 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 two options for the number of layers in the bathymetry file. If wanting to prescribe the number of layers, the *Number of layers* option should appear in the box and the desired number of layers entered in the text box titled value. If wanting a certain layer height, 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 accordingly 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

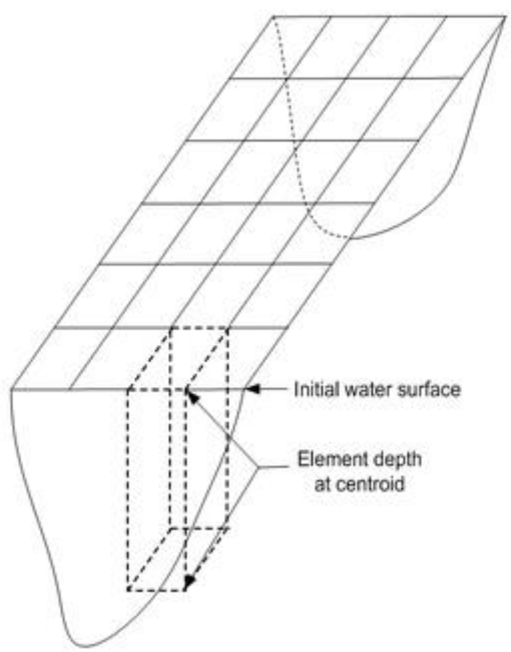
Introduction

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).

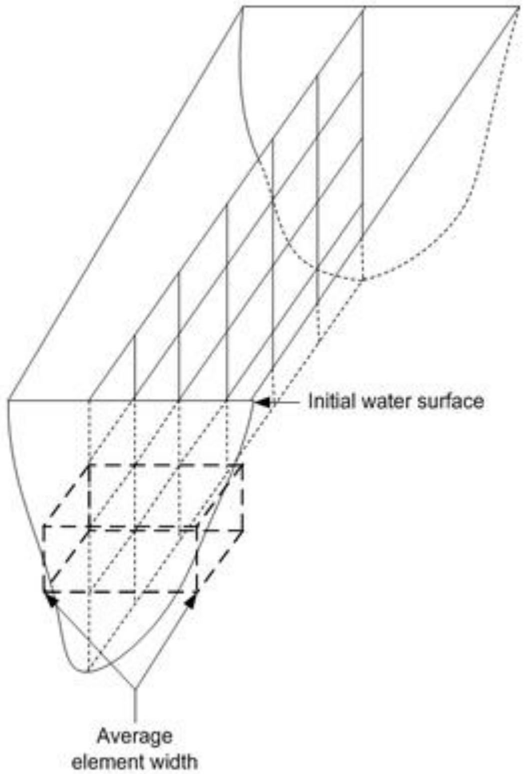
This page discusses the process WMS uses to compute bathymetric layer widths for CE-QUAL-W2 simulations. WMS is extremely useful for setting up CE-QUAL-W2 bathymetry files. WMS can also be used to setup CE-QUAL-W2 modeling parameters, but this page focuses only on the bathymetry file setup.

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 defining a spatial domain of the model and then appropriately filling 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).

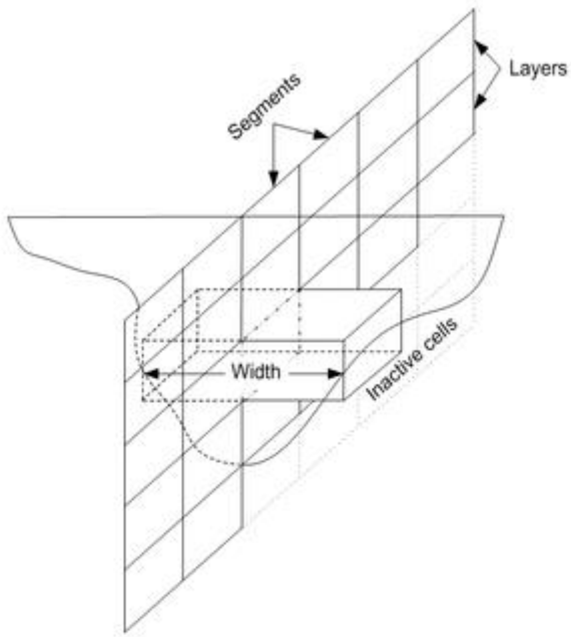


A major limitation of depth-averaged models is that they do not represent vertical variations well. For example, velocities computed using such a model do not account for variations that may occur between the top and bottom of the model since a single, average, value for each vertical column is computed. 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.



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.

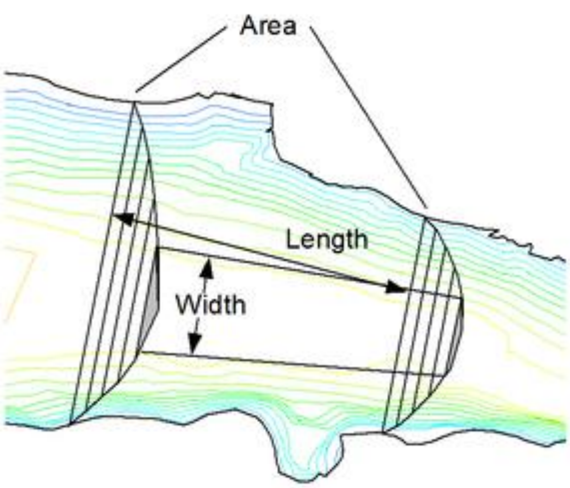
Layer depths are generally a function of the sensitivity, or gradient of the vertical variations of the variables being analyzed (i.e. temperature, phosphorous, dissolved oxygen, etc.). The higher the gradient the smaller the element heights should be. Once lengths and heights are determined a finite 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.



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.



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.

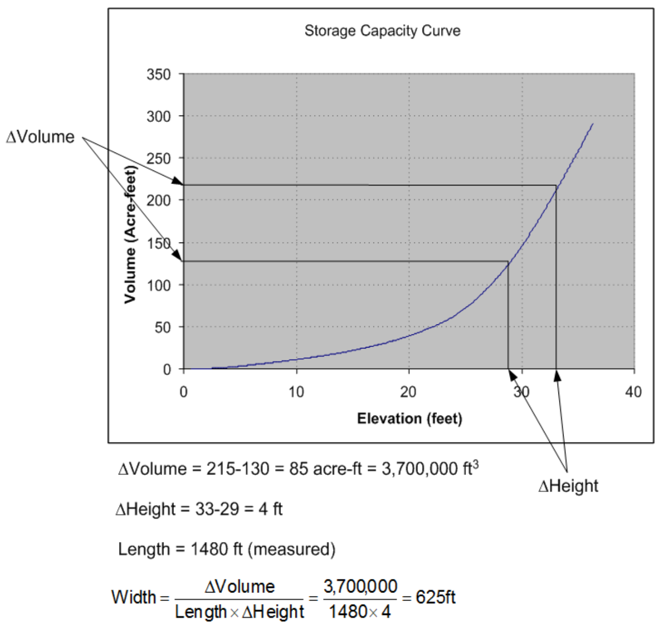
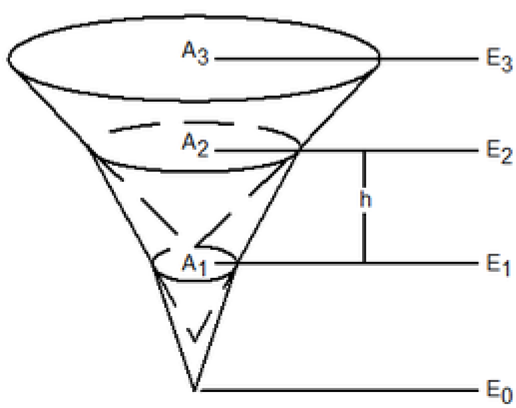


Figure 5: Calculation of width from incremental volumes between two reservoir heights.

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.

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).

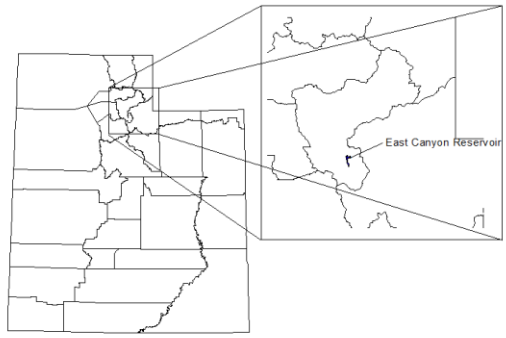


Figure 7: Location of the East Canyon Reservoir.

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.

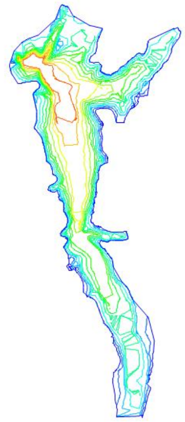


Figure 8: Bathymetric data for the East Canyon Reservoir.

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.

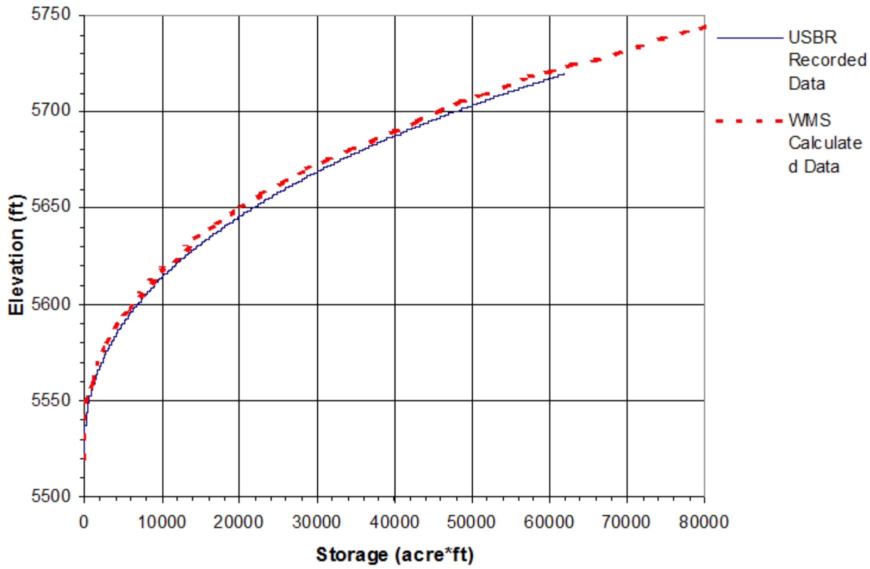


Figure 10: Comparison of original storage capacity curve with one computed by WMS.

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.

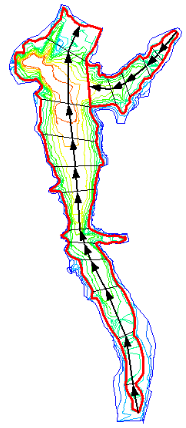


Figure 11: East Canyon conceptual model of branches and segments.

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. [36]

•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. [37]

•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. [38]

•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. [39]

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

The GSSHA model can be added to a paid edition of WMS.

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.

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.

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 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

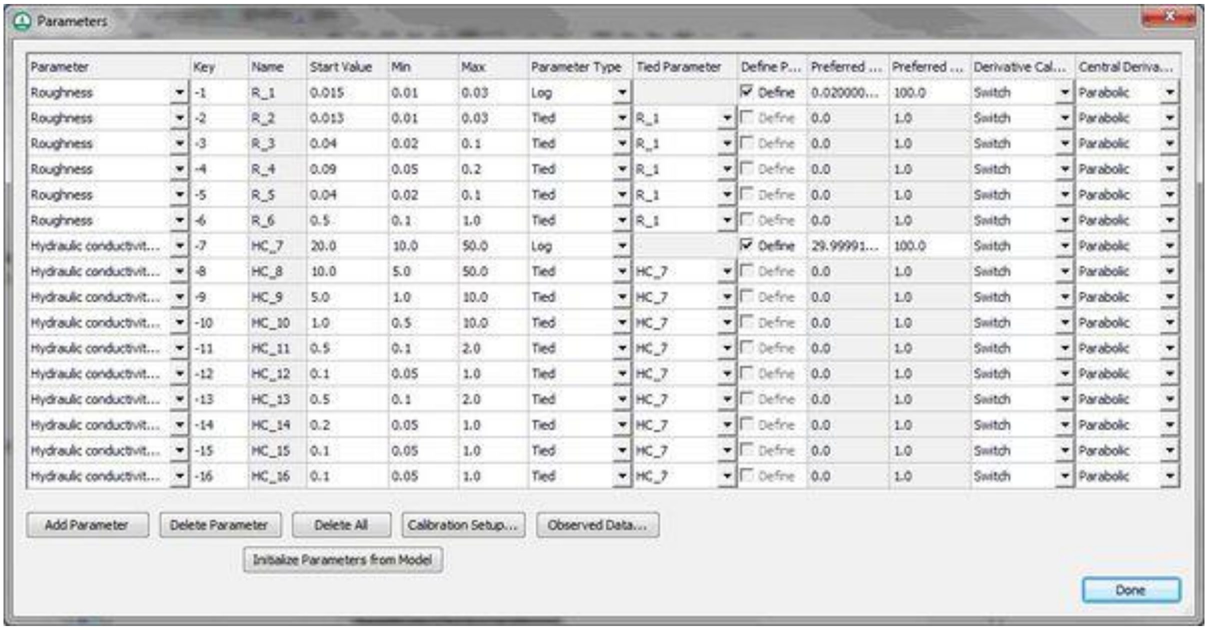
- GSSHA Calibration
- GSSHA Channel Routing

GSSHA Calibration

The GSSHA model allows for both automatic (auto) calibration and manual calibration.

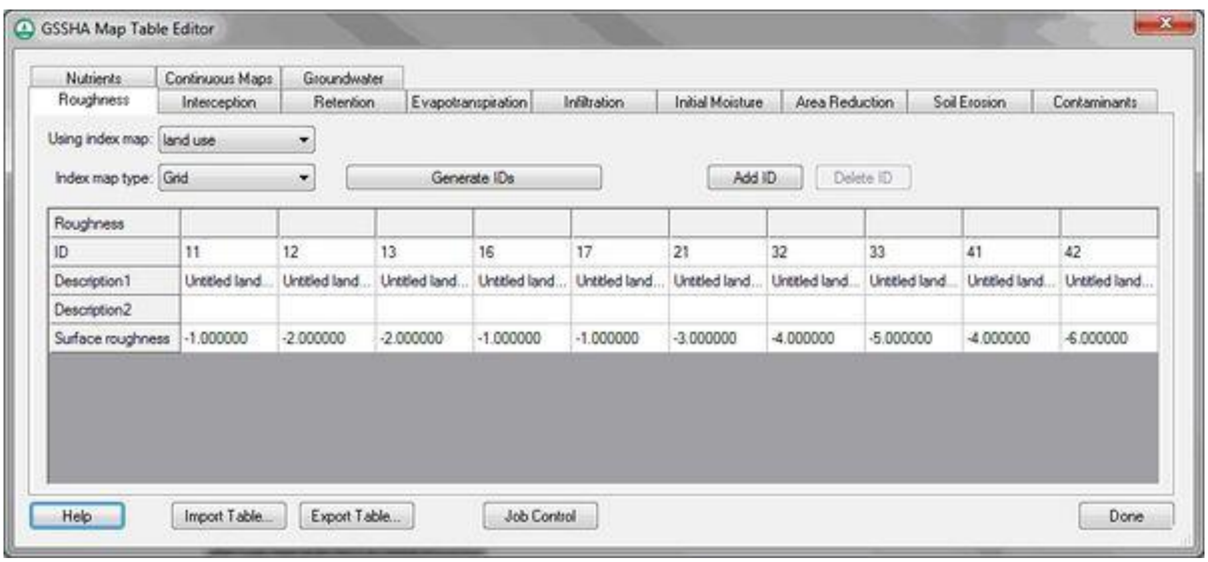
Auto Calibration

GSSHA has a PEST-based automated calibration routine that is described on the GSSHA wiki. WMS writes the parameters, observed data, and calibration files necessary to run GSSHA in any of the supported GSSHA calibration modes. There is a tutorial included in the WMS Tutorials that describes how to setup a basic calibration model. Enter in the needed data by toggling *Calibration* in the *Job Control* dialog. Clicking on the **Edit Parameters** button allows selection of which parameters are to be calibrated and edit the calibration settings associated with each of the parameters. The dialog shown below appears. Here enter the maximum, minimum, and initial values for each parameter.

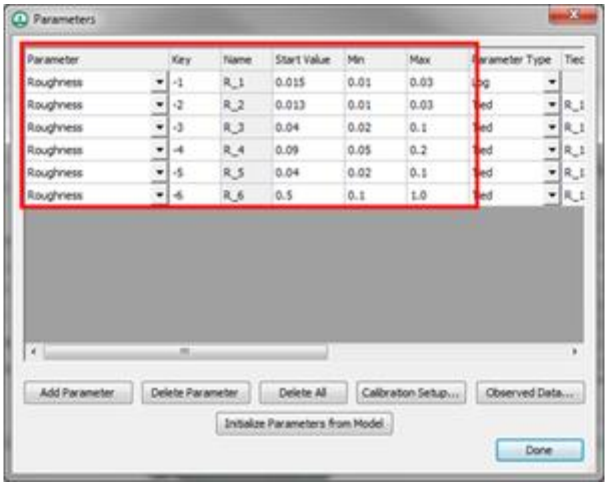


Defining Key Values

Select adjustable parameters by defining *key* values as negative numbers in the regular GSSHA interface and then assigning starting, min, and max values to each of these parameters in the calibration interface. For example, if wanting to set overland Manning’s roughness values as an adjustable model parameter, specify negative integers for the values in the mapping table dialog as shown below:



Then define the key values in the calibration *Parameters* dialog as shown below:

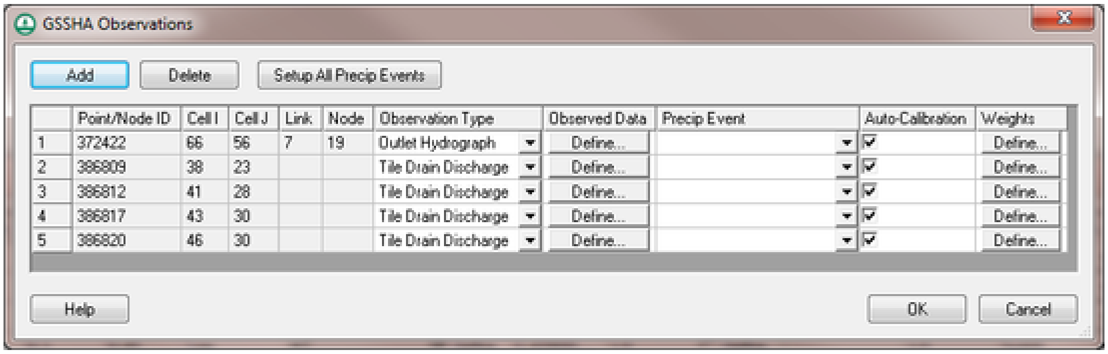


The **Initialize Parameters from Model** button makes defining key values easy. This button searches the model for any parameters that are defined as negative integers and assigns these parameters to the dialog. For each key value, define a start, min, and max value for the calibration engine. Also define other parameter information, such as whether parameters are fixed or tied to other parameters and the methods used to calculate derivatives for each parameter. Options exist to define regularization parameters that define prior information for the calibration run. There are options to define preferred values, where a specific value is set as an optimal value for the calibration parameter, or homogeneous values, where one or more calibration parameters are linked to another calibration parameter in an attempt to get these values as close as possible.

Defining Observed Data

Clicking the **Observed Data...** button in the *Calibration Parameters* dialog opens another dialog, which is shown below. In this dialog, enter in various types of observed data (including time series hydrograph data) for each event of the simulation. It is not necessary to associate a rainfall event with each observed data time series unless running an SCE-type simulation. This dialog shows all the feature points with observed data and allows turning automated calibration on or off for each of these observations. An SCE-type automated calibration simulation can only be used to calibrate the hydrograph at the outlet point and has been deprecated in the current version of WMS. All other automated calibration methods support automated calibration of the following data types at any computation point in the watershed model:

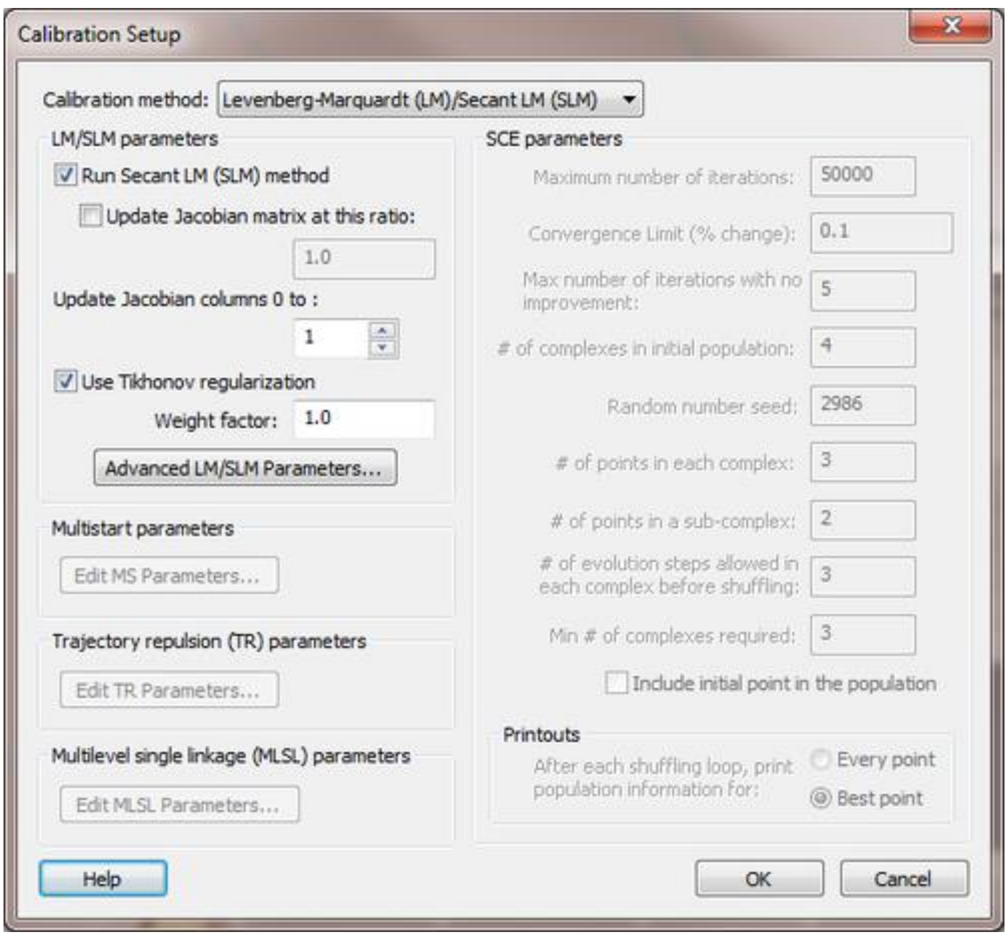
- Overland Depth
- Infiltration Depth
- Surface Moisture
- Grid Suspended Sediment (TSS) Concentration
- Channel Depth
- Channel Flow
- Channel Total Suspended Sediment (TSS) Concentration
- Groundwater Head
- Outlet Hydrograph (Only at the watershed outlet point)
- Snow Water Equivalent
- Tile Drain Discharge (In a GSSHA Storm Drain coverage)



The *Observation* dialog can be accessed from the feature point/node attribute dialog in the GSSHA and the GSSHA Storm Drain coverages. If WMS is running the default LM/SLM-based calibration, associate weights with each observation value in the XY series by clicking on the **Weights** button in the observation window and defining weights for each value. WMS defaults all the weights for observed data points to 1.0, but this value could be modified to give higher weights to certain observed values.

Defining Calibration Setup Parameters

Lastly, set the calibration setup parameters, which can be accessed by clicking the **Calibration Setup...** button. The following dialog will appear:



The parameters in this dialog are used for the calibration control file. This file, along with the parameters and observed data files are written out by WMS when saving a GSSHA project.

The following options are of note in this dialog:

Calibration method

The following calibration methods are available: Levenberg-Marquardt (LM)/Secant LM (SLM), Multistart (MS), Trajectory Repulsion (TR), Multilevel Single Linkage (MLSL), and Shuffled Complex Evolution (SCE). The LM and SLM methods use a local search to optimize model parameters while the other methods are global search methods. You can find more detailed information about each of these optimization methods on the GSSHA wiki . The SCE method can be considered a deprecated optimization method in GSSHA. Use one of the other optimization methods to optimize your model.

Run Secant LM (SLM) method

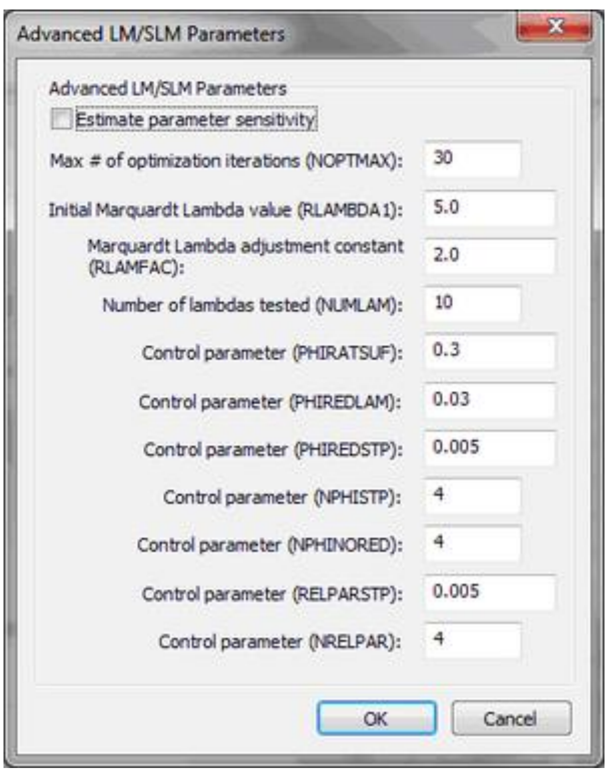
Turning on the option to run the SLM method sets the input file flag to run the SLM method instead of the LM (Levenberg-Marquardt) local search method. The SLM method is an efficiency enhancement to the LM method and is the default local search optimization method for a GSSHA calibration in the WMS interface.

Use Tikhonov regularization

Often, when you define preferred values or homogeneous parameter values in a calibration model using the PEST-based prior information (regularization) option, it is desirable to find a balance between fitting the solution to the observed data and fitting to the regularization relationships. The Tikhonov regularization option provides a way to adjust this balance by defining a regularization weight factor and running Tikhonov regularization using this weight factor. This weight factor balances finding a solution that matches observed data versus fitting prior information (regularization) relationships. See the GSSHA wiki for more information about Tikhonov regularization with GSSHA.

Advanced Calibration Parameters

Sometimes, it's desired to set more advanced calibration settings when running an LM/SLM calibration. These settings can be set in the **Advanced LM/SLM Parameters** dialog and are described below:



Estimate parameter sensitivity

Turn this option on to write a -1 to the NOPTMAX parameter in the PEST control file. This tells PEST to only run a sensitivity analysis instead of a full parameter optimization run. The sensitivity file is read into WMS if reading the calibration solution.

(The descriptions below are taken from the PEST User Manual, Copyright 2013)

NOPTMAX

This sets the maximum number of optimisation iterations that PEST is permitted to undertake on a particular parameter estimation run. If wanting to ensure that PEST termination is triggered by other criteria more indicative of parameter convergence to an optimal set or of the futility of further processing, set this variable very high. A value of 20 to 30 is often appropriate.

If NOPTMAX is set to zero, PEST will not calculate the Jacobian matrix. Instead it will terminate execution after just one model run. This setting can thus be used when wanting to calculate the objective function corresponding to a particular parameter set and/or to inspect observation residuals corresponding to that parameter set.

RLAMBDA1

This real variable is the initial Marquardt lambda. PEST attempts parameter improvement using a number of different Marquardt lambdas during any one optimization iteration; however, in the course of the overall parameter estimation process, the Marquardt lambda generally gets smaller. An initial value of 1.0 to 10.0 is appropriate for most models, though provide a higher initial Marquardt lambda if PEST complains that the normal matrix is not positive definite .

For high values of the Marquardt lambda the parameter estimation process approximates the steepest-descent method of optimization. While the latter method is inefficient and slow if used for the entirety of the optimization process, it often helps in getting the process started, especially if initial parameter estimates are poor.

RLAMFAC

RLAMFAC, a real variable, is the factor by which the Marquardt lambda is adjusted. RLAMFAC must be greater than 1.0. When PEST reduces lambda it divides by RLAMFAC; when it increases lambda it multiplies by RLAMFAC. PEST reduces lambda if it can. However if the normal matrix is not positive definite or if a reduction in lambda does not lower the objective function, PEST has no choice but to increase lambda.

NUMLAM

This integer variable places an upper limit on the number of lambdas that PEST can test during any one optimization iteration. It should normally be set between 5 and 10. For cases where parameters are being adjusted near their upper or lower limits, and for which some parameters are consequently being frozen (thus reducing the dimension of the problem in parameter space) experience has shown that a value closer to 10 may be more appropriate than one closer to 5; this gives PEST a greater chance of adjusting to the reduced problem dimension as parameters are frozen.

PHIRATSUF

During any one optimization iteration, PEST may calculate a parameter upgrade vector using a number of different Marquardt lambdas. First it lowers lambda and, if this is unsuccessful in lowering the objective function, it then raises lambda. If, at any stage, it calculates an objective function which is a fraction PHIRATSUF or less of the starting objective function for that iteration, PEST considers that the goal of the current iteration has been achieved and moves on to the next optimization iteration.

PHIRATSUF (which stands for “phi ratio sufficient”) is a real variable for which a value of 0.3 is often appropriate. If it is set too low, model runs may be wasted in search of an objective function reduction which it is not possible to achieve, given the linear approximation upon which the optimization equations are based. If it is set too high, PEST may not be given the opportunity of refining lambda in order that its value continues to be optimal as the parameter estimation process progresses.

PHIREDLAM

If a new/old objective function ratio of PHIRATSUF or less is not achieved as the effectiveness of different Marquardt lambdas in lowering the objective function are tested PEST must use some other criterion in deciding when it should move on to the next optimization iteration. This criterion is partly provided by the real variable PHIREDLAM. The first lambda that PEST employs in calculating the parameter upgrade vector during any one optimization iteration is the lambda inherited from the previous iteration, possibly reduced by a factor of RLAMFAC (unless it is the first iteration, in which case RLAMBDA1 is used). Unless, through the use of this lambda, the objective function is reduced to less than PHIRATSUF of its value at the beginning of the iteration, PEST then tries another lambda, less by a factor of RLAMFAC than the first. If the objective function is lower than for the first lambda (and still above PHIRATSUF of the starting objective function), PEST reduces lambda yet again; otherwise it increases lambda to a value greater by a factor of RLAMFAC than the first lambda for the iteration. If, in its attempts to find a more effective lambda by lowering and/or raising lambda in this fashion, the objective function begins to rise, PEST accepts the lambda and the corresponding parameter set giving rise to the lowest objective function for that iteration, and moves on to the next iteration. Alternatively if the relative reduction in the objective function between the use of two consecutive lambdas is less than PHIREDLAM, PEST takes this as an indication that it is probably more efficient to begin the next optimization iteration than to continue testing the effect of new Marquardt lambdas.

A suitable value for PHIREDLAM is often around 0.01. If it is set too large, the criterion for moving on to the next optimisation iteration is too easily met and PEST is not given the opportunity of adjusting lambda to its optimal value for that particular stage of the parameter estimation process. On the other hand if PHIREDLAM is set too low, PEST will test too many Marquardt lambdas on each optimization iteration when it would be better off starting on a new iteration.

PHIREDSTP, NPHISTP

PHIREDSTP is a real variable whereas NPHISTP is an integer variable that are used to tell PEST that the optimization process is at an end. For many cases 0.01 and 4 are suitable values for PHIREDSTP and NPHISTP respectively. However, be careful not to set NPHISTP too low if the optimal values for some parameters are near or at their upper or lower bounds. In this case it is possible that the magnitude of the parameter upgrade vector may be curtailed over one or a number of optimization iterations to ensure that no parameter value overshoots its bound. The result may be smaller reductions in the objective function than would otherwise occur. It would be a shame if these reduced reductions were mistaken for the onset of parameter convergence to the optimal set.

NPHINORED

If PEST has failed to lower the objective function over NPHINORED successive iterations, it will terminate execution. NPHINORED is an integer variable; a value of 3 or 4 is often suitable.

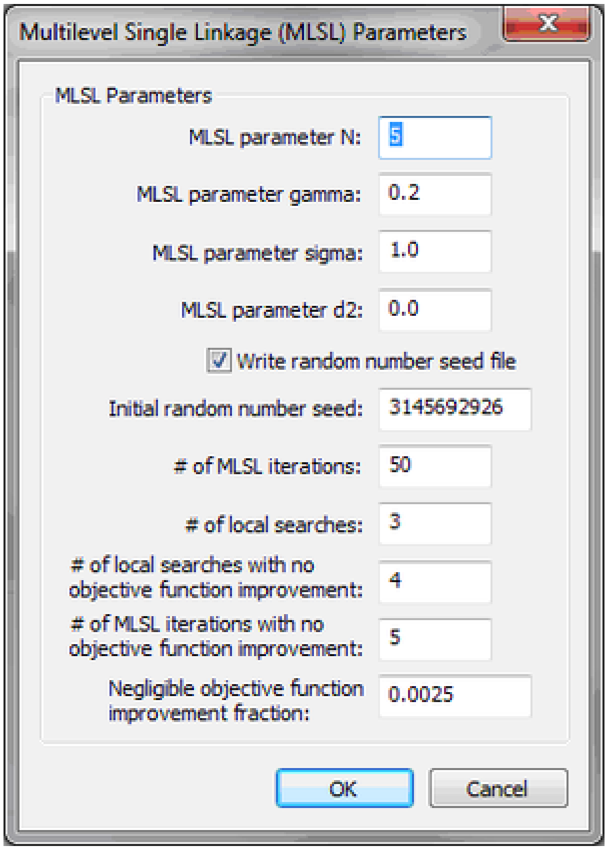
RELPARSTP, NRELPAR

If the magnitude of the maximum relative parameter change between optimization iterations is less than RELPARSTP over NRELPAR successive iterations, PEST will cease execution. PEST evaluates this change for all adjustable parameters at the end of each optimisation iteration, and determines the relative parameter change with the highest magnitude. If this maximum relative change is less than RELPARSTP, a counter is advanced by one; if it is greater than RELPARSTP, the counter is zeroed.

All adjustable parameters, whether they are relative-limited or factor-limited, are involved in the calculation of the maximum relative parameter change. RELPARSTP is a real variable for which a value of 0.01 is often suitable. NRELPAR is an integer variable; a value of 2 or 3 is normally satisfactory.

Calibration Model-Specific Parameters

Each calibration model has model-specific calibration parameters. The LM/SLM parameters are local calibration methods and are used with any of the PEST-based global calibration methods as well. The Multistart, TR, and MLSL parameters can be edited by selecting the desired calibration type and then setting the parameters for the calibration in the calibration model dialog, like the MLSL dialog shown below:



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

•WMS Tutorials

GSSHAWiki

Primer

•Watershed Delinieation and Grid Construction

Tutorials

•11 Manual Calibration

User's Manual

•Building a GSSHA Model

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

Primer

•Overview

Tutorials

User's Manual

•Preface

•Introduction

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:

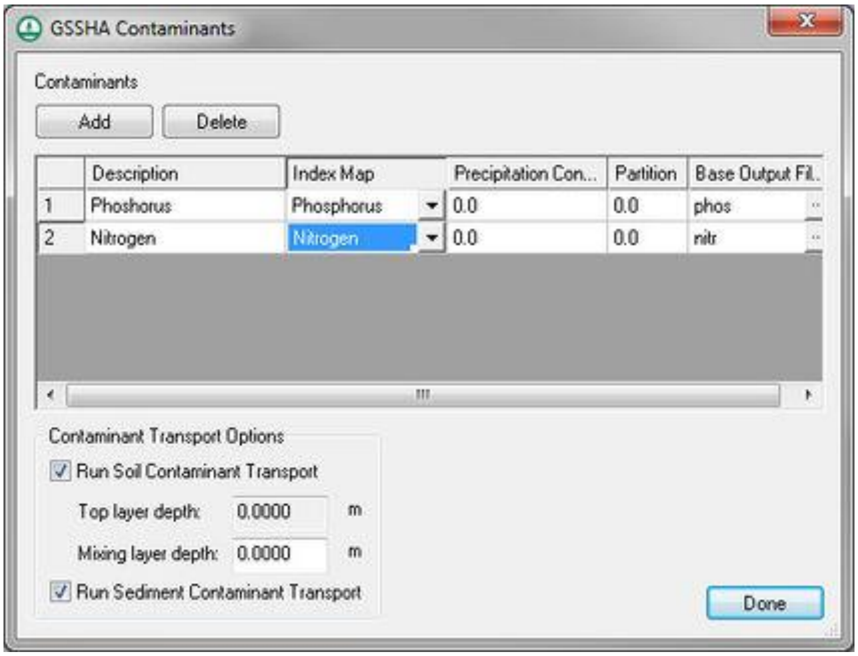
1. Set up the contaminant types.
2. Create the index maps for the contaminants.
3. Set up the contaminant parameters in the mapping table.

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

Primer

- Mapping

User's Manual

- Mapping Table File
  - Index Maps
  - Mapping Tables

Tutorials

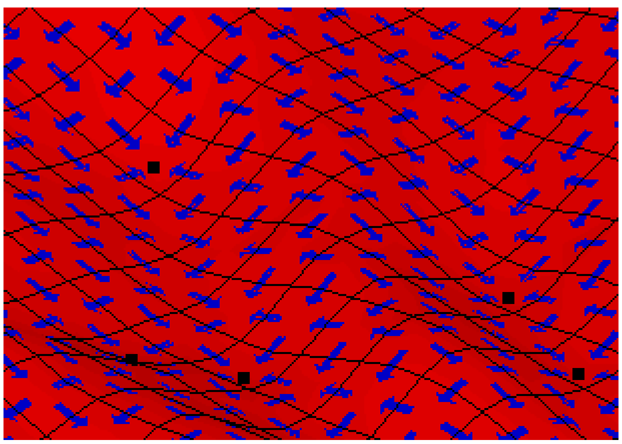
GSSHA Digital Dams

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

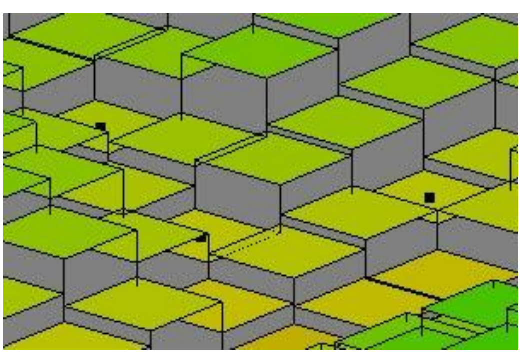
WMS now has two ways to visualize digital dams. These options are accessible on the *2D Grid* tab of the *Display Options* dialog. Toggle on the *Digital Dams* option to have WMS identify digital dams is by looking for cells that are lower than their four neighbors. The cells that meet this criterion 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.

WMS 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

Primer

- Watershed Delinieation and Grid Construction
  - Editing the Grid to Correct Elevation Errors

Tutorials

- Fixing Digital Dams

User's Manual

- General Considerations
  - Elevation Map

GSSHA Embankment Arcs

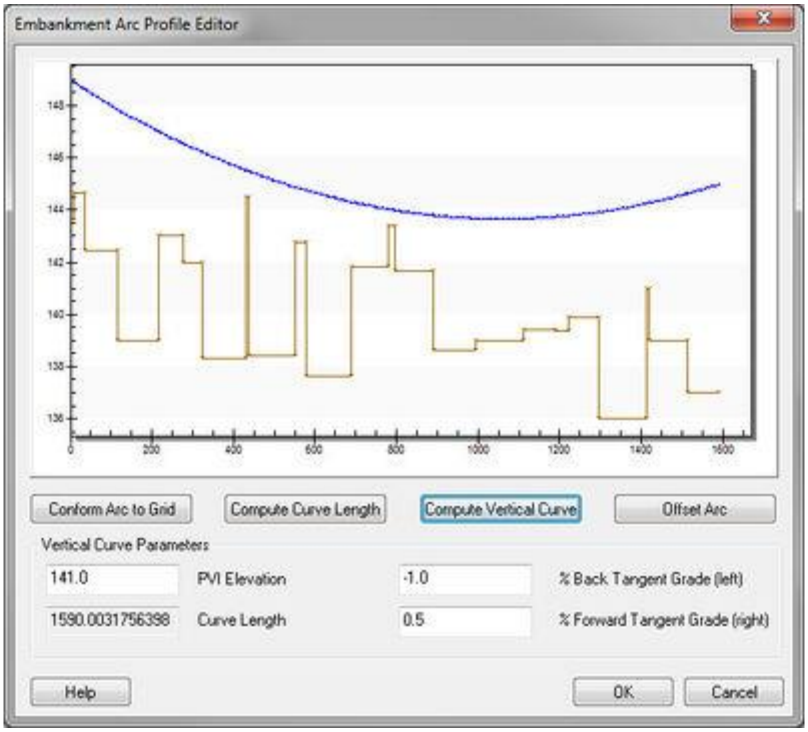
Embankment Arcs in WMS

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 user specified constant.

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

Primer

•Watershed Delinieation and Grid Construction

•Lake and Channel Routing

Tutorials

User's Manual

•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

See GSSHA Embankment Arcs for more information.

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

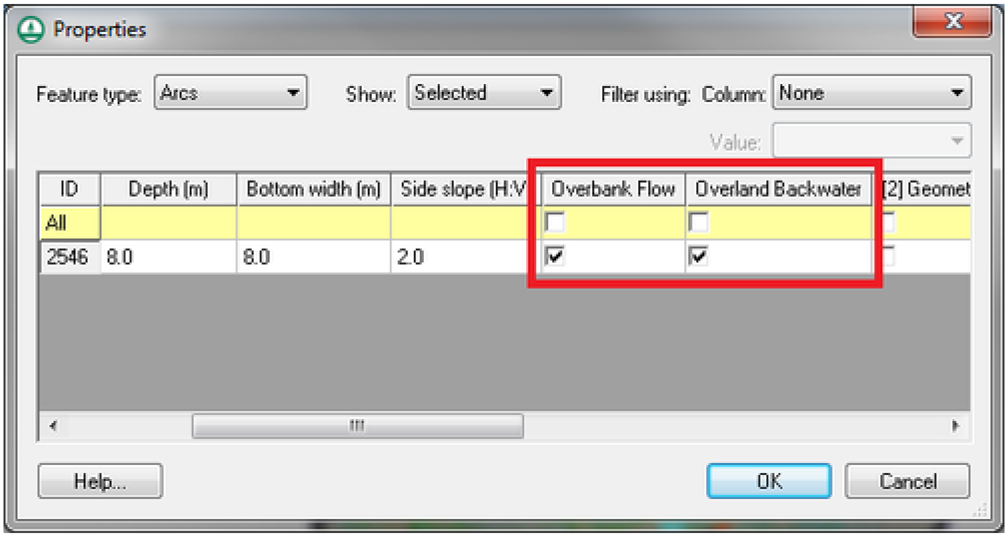
Assign one of the following groundwater boundary conditions to the 2D grid cells intersecting the feature arc: Generic, No Flow, Constant Head, Flux River, or Head River. For more information, see GSSHA Groundwater .

Solution Results

Click on the button to view results at all node locations for the link in the stream/channel network. See GSSHA Solution Results for information.

Specified Overbank and Backwater Channel Links

GSSHA has two options that could be defined globally. The first option, the overbank flow option (OVERBANK_FLOW), increases the level of connection between the overland flow and the channel in the 1D hydraulic model by allowing water to spill from the channel back onto the overland flow plane. If the second option, the overland backwater option (OVERLAND_BACKWATER), is turned on, flow from the overland flow model to the channel in the 1D hydraulic model is restricted if the elevation of the water in the channel exceeds the overland cell elevation. This option can be defined at each of the arcs (links) in the GSSHA model in the GSSHA arc properties dialog. If this option is defined for one of the arcs, the global option is turned off.



Related Topics

- [GSSHA Embankment Arcs]
- [GSSHA Feature Nodes]
- [GSSHA Feature Polygons]
- [GSSHA Groundwater]
- [GSSHA Job Control]
- [GSSHA Solution Results]
- [GSSHA Stream Arcs]

GSSHAWiki

Primer

- Watershed Delinieation and Grid Construction
- Lake and Channel Routing

Tutorials

User's Manual

- Surface Water Routing

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

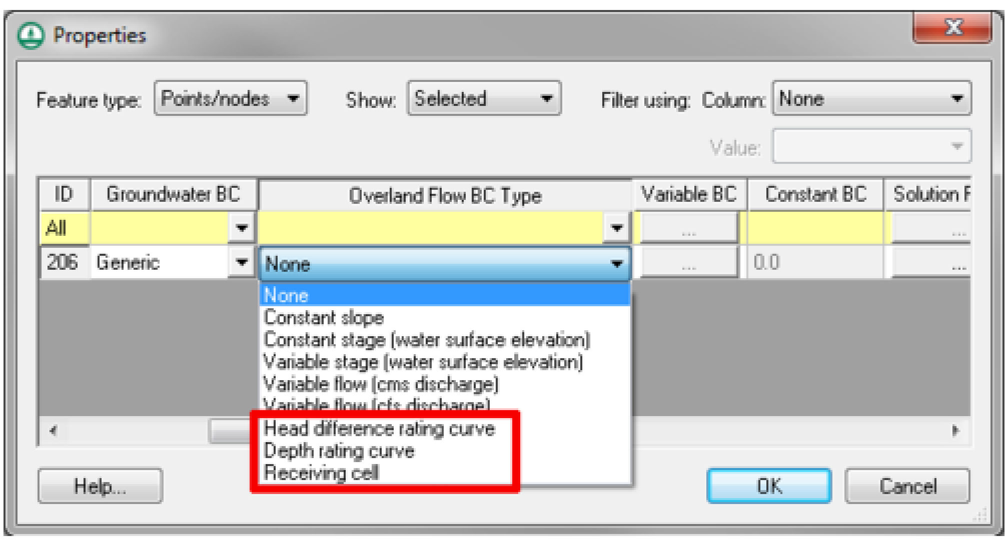
The link number is assigned by WMS to nodes that have hydraulic structures defined.

Overland Flow BC Type

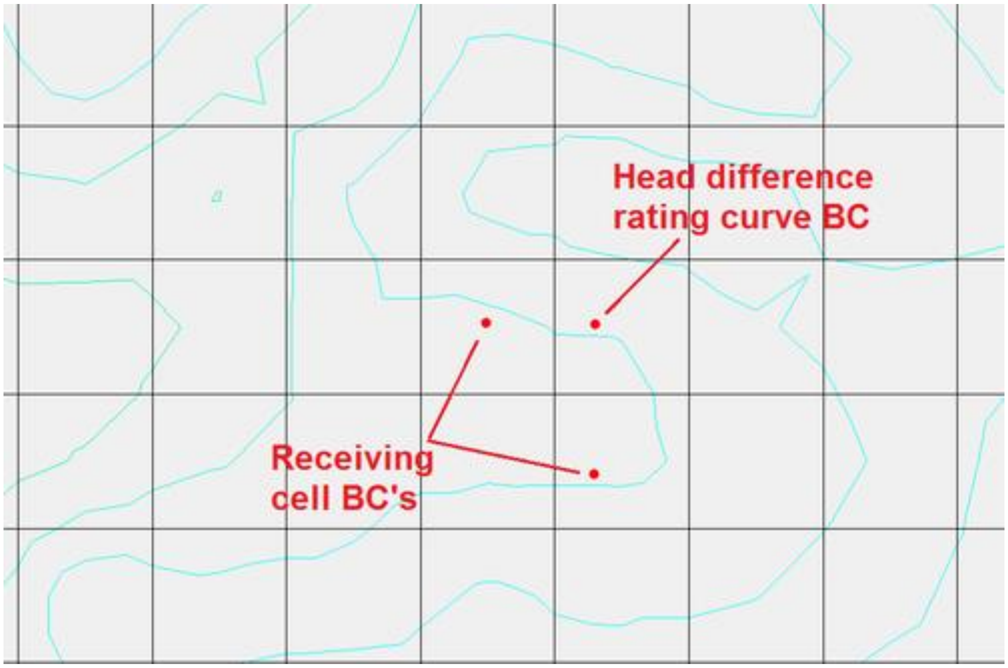
There are several overland flow BC options that can be specified for points and nodes in the GSSHA coverage. These include the following:

- Constant slope
- Constant stage
- Variable stage
- Variable flow (cms)
- Variable flow (cfs)
- Head difference rating curve
- Depth rating curve
- Receiving cell

If the *Head difference rating curve* option is selected, define a relationship between the BC cell/receiving cell head difference and the discharge to the receiving cell. If the *depth rating curve* option is selected, its necessary to define a relationship between the depth in the BC cell and the discharge to the receiving cell. The relationship is defined in the WMS *XY Series Editor* . In either case, define one or more receiving cells.



For example, consider the following diagram where there is one cell with a head difference rating curve BC and two receiving cells:



This example shows a correct use of the new point boundary conditions in WMS. If one of the rating curve BC's does not have an adjacent receiving cell, WMS shows a message that says something like the following when running the GSSHA model checker: *WARNING: The grid cell at I=34 and J=73 does not have an adjacent receiving cell.*

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 four main types of hydraulic structures: detention basins (lakes), weirs, culverts, and implicit defined structures. Any number of lakes, weirs, culverts, and/or curves may be specified together, but with only one instance of each curve type allowed.

Detention Basins/Lakes

See the page describing how to define lakes for more information about defining detention basins and lakes in a GSSHA model. It is simpler and recommended that the user define a lake as a detention basin on a hydraulic structure at a point in a GSSHA model instead of using a polygon.

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. See GSSHA Groundwater for more information.

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

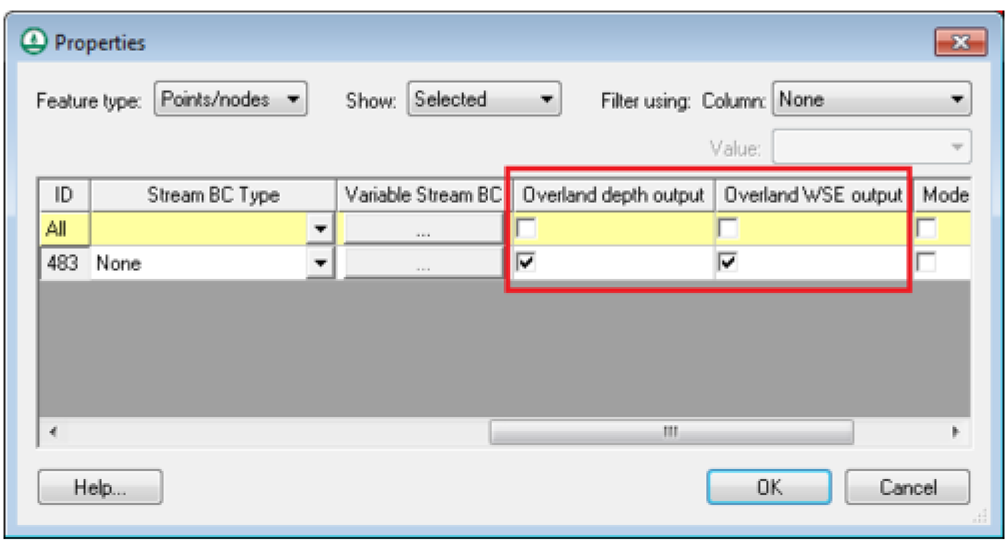
Click on this button to visualize grid or link/node datasets for this node. See GSSHA Solution Analysis for more information.

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. See GSSHA Observation Points for more information.

Point Overland and Stage Output

Point stage and depth time series information can be output from any cell on the overland flow plane. To get the output, define feature points in the GSSHA coverage and turn on the *Overland depth output* and/or the *Overland WSE output* options, as shown below:



Related Topics

•GSSHA Feature Arcs

•GSSHA Groundwater

•GSSHA Calibration

•GSSHA Observation Points

•GSSHA Solution Results

GSSHAWiki

Primer

•Watershed Delinieation and Grid Construction

•Lake and Channel Routing

Tutorials

User's Manual

•Surface Water Routing

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

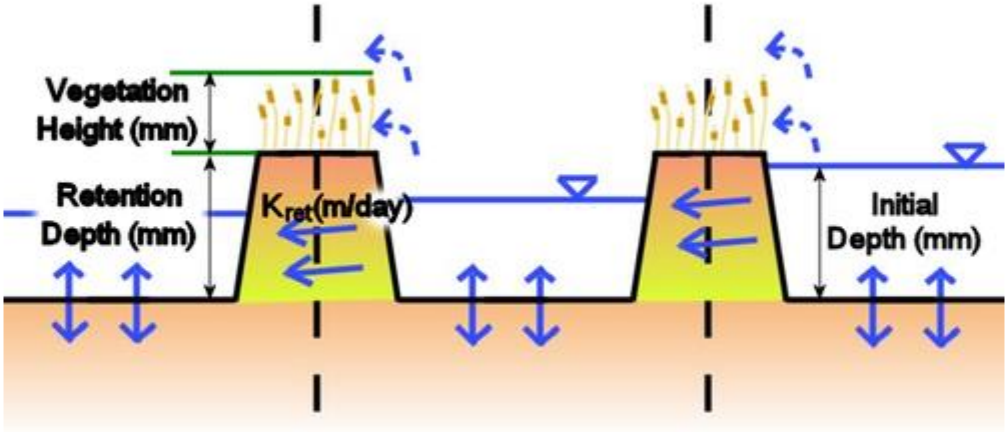
Boundary polygons are used to define the perimeter of the watershed and are used for creating grids from an already known watershed boundary.

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



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

Primer

- Watershed Delinieation and Grid Construction
- Lake and Channel Routing

Tutorials

User's Manual

- Surface Water Routing

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 regenerating 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

Primer

- Groundwater
- Richards' Equation

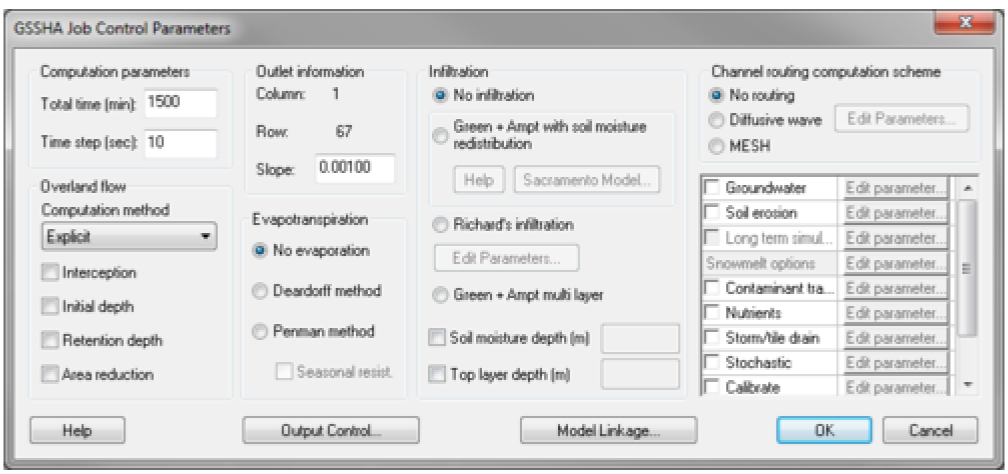
Tutorials

User's Manual

- Groundwater
- Infiltration

GSSHA Job Control

The *GSSHA Job Control Parameters* dialog contains 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.



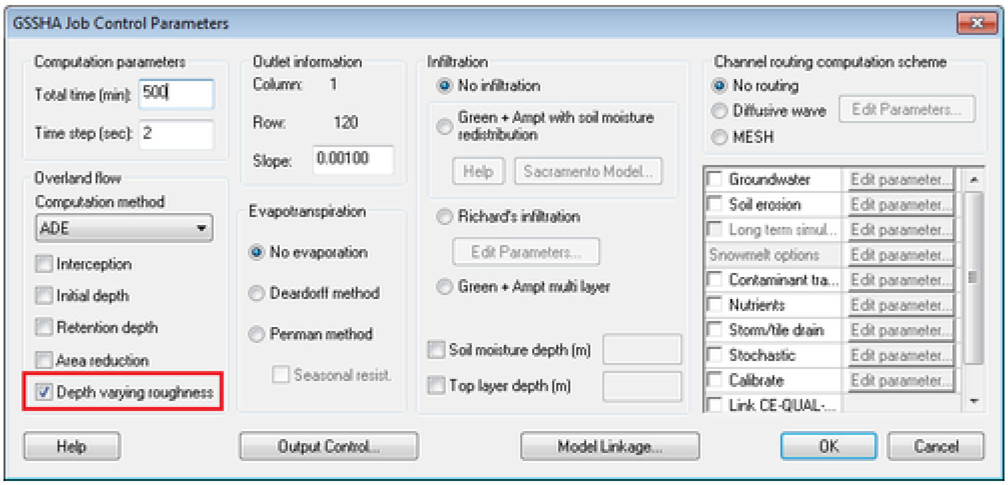
Computation Parameters (required)

- 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)

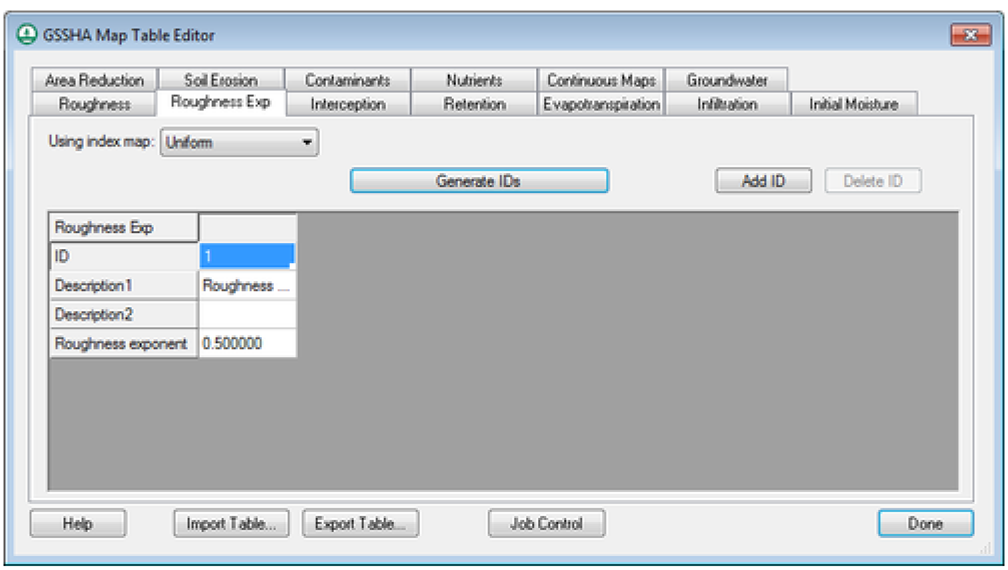
- Choosing an appropriate time step is critical to the success of the simulation. If the time step used is too large then excessive numerical averaging will take place, delaying simulation flows and possibly causing oscillating results. Too small of a time step will take an inordinate amount of time 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.

Depth Varying Roughness

There is an option in the GSSHA Job Control dialog to define a depth-varying overland flow roughness mapping table, as shown below:



If this option is defined, define a mapping table specifying a depth-varying overland flow roughness exponent. This parameter is used in addition to the roughness mapping table, as shown below:



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 – Used to manage the paths and filenames of all GSSHA input/output files

Related Topics

•GSSHA Overview

•GSSHA Output Control

•GSSHA Precipitation

GSSHAWiki

Primer

•Overview

- Modeling Process

Tutorials

User's Manual

•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. Use the **Join NRCS Data** command to join more attributes which can then be mapped to the soil coverage.

The **Join NRCS 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

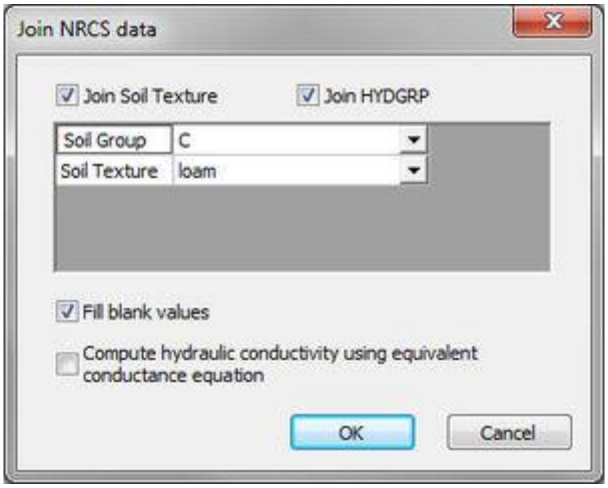
•**chtextur.txt** : TEXTURE

•**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. 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, select the **Join NRCS Data** command again in order to join the data.

Related Topics

•GSSHA Overview

GSSHAWiki

Primer

•Overview

User's Manual

•Preface

•Introduction

Tutorials

GSSHA Lakes and Wetlands

Defining Wetlands and Lakes

Wetlands can be created as feature polygons in a GSSHA coverage. Lakes and detention basins should be defined at a point by defining a detention basin hydraulic structure at a point attribute on a stream in a GSSHA coverage.

Follow these additional steps after assigning a detention basin as a node attribute in a GSSHA coverage:

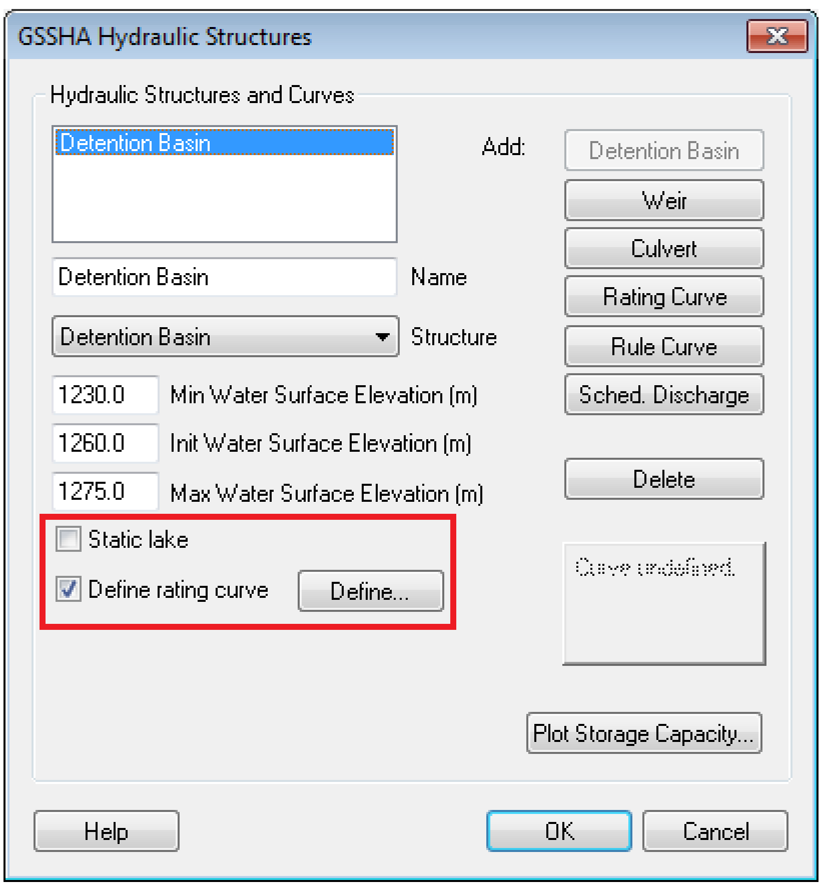
1. Create an embankment arc downstream from the lake that represents the dam elevation for the lake. Make sure the type of arc is set to be an embankment arc and that the elevations of the arc are set in the embankment arc editor. When creating the embankment arc, the grid edges defined by the embankment arc must be downstream from the outlet point/detention basin hydraulic structure. If the edges are not downstream from the outlet (display the edges to make sure they are outside), an error will appear after defining the lake water surface elevations.

2. Go to the detention basin node hydraulic structure attributes and set the minimum water surface elevation, initial water surface elevation, and maximum water surface elevation in the *GSSHA Hydraulic Structures* dialog.

3. Define any outlet structures, such as culverts, weirs, rating curves, scheduled discharges or rule curves, for this feature node with the detention basin attributes.

Static Lakes

One option of note is the static lake option. Lakes can be specified as static features on the overland and channel. This can be computationally expedient because the calculations required for lakes taking and releasing overland cells and channel nodes is removed. The lake is assumed to occupy the maximum lake extent specified. To make a lake static, turn on the *Static lake* option in the *GSSHA Hydraulic Structures* dialog. The Min/Max/Init WSE must still be specified. All cells below the Max WSE become lake cells and any water that enters the cell through any mechanism will be added to the lake. If the lake area is actually highly variable, then the accuracy of this approximation may suffer.



Lake Rating Curves

A lake rating curve option is available. A stage/volume/area curve can be specified for a lake by toggling the *Define rating curve* option and defining the curve. The rating curve stage values must go up to at least the Max WSE. Rating curves can be used with or without the *Static Lake* option turned on.

If the lake rating curve is specified, the lake stage/area/volume calculations will be performed using the rating curve. The lake will still interact with the stream and the overland based on the stage of the lake unless the *Static lake* option is toggled on.

Wetlands

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 in GSSHA

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

Primer

•Wetlands

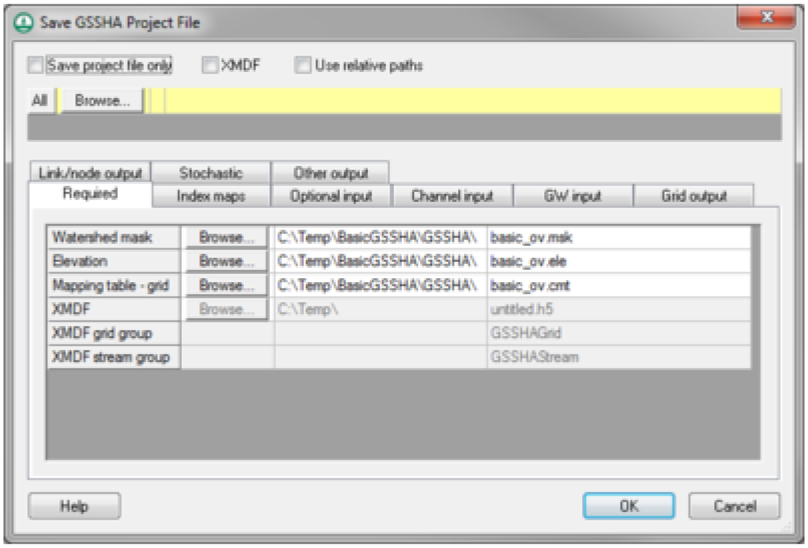
- Wetlands Conceptual Model
- Wetlands Setup

Tutorials

User's Manual

•Surface Water Routing

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. This option is not normally used if setting up a basic GSSHA model or if new to the GSSHA model.

There are several files that are included as a part of the GSSHA project. If building a basic GSSHA model using the WMS interface, the individual file names associated with the GSSHA project are not normally edited. However, there might be a need to rename a specific index map, other project file, or 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, the name of the infiltration depth file name cannot be edited. Turn on the option to export the infiltration depth from GSSHA to define an infiltration depth filename in the *GSSHA Project File* editor.

There is also 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 wanting to turn on that process in the *Job Control* and will bring up the *Job Control* dialog. If an index map has been assigned to a process and the IDs used in the index map increases, 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

Primer

•Mapping Tables

Tutorials

User's Manual

•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, a project might have a stream-based index map instead of a grid-based index map. When defining a stream index map, define an index value greater than zero to each link in the model. There is not currently an option to automatically define stream index map values from grid or map module data. Turn on the stream link IDs in the *Display Options* dialog and define an index for each of these IDs. Assign index values to each of the stream links in the model. 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

- Assigning Parameter Values to Individual Grid Cells
  - Index Maps
- Mapping Table File
  - Index Maps

Tutorials

GSSHA Multiple Scenarios

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 it is possible to 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 is to be assigned.

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 is to be assigned. 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 is to be assigned. 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 prompting to select which scenarios to save. Also 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 are to be opened. This will open each project that corresponds with this group. There is also 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 to select which projects to run. These projects will be saved as a group and then WMS will run each GSSHA scenario in the group.

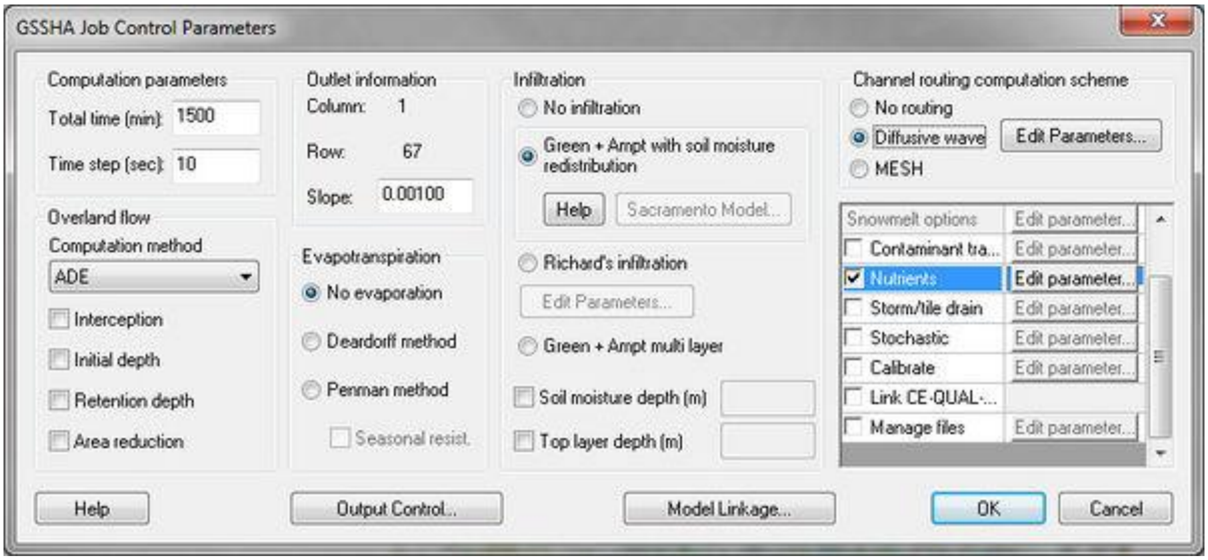
Related Topics

•GSSHA Overview

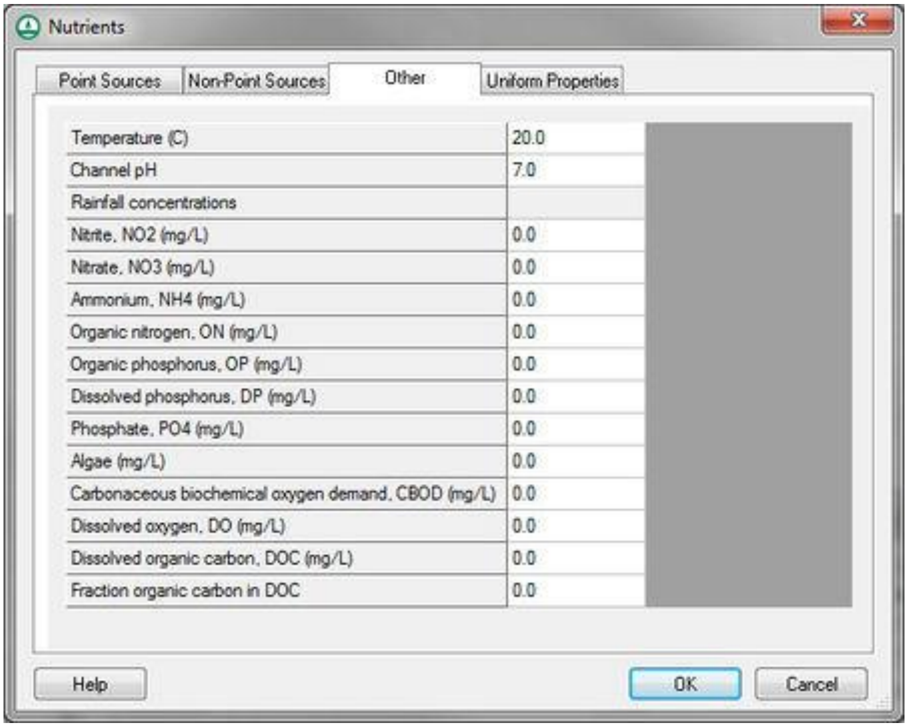
GSSHA Nutrients

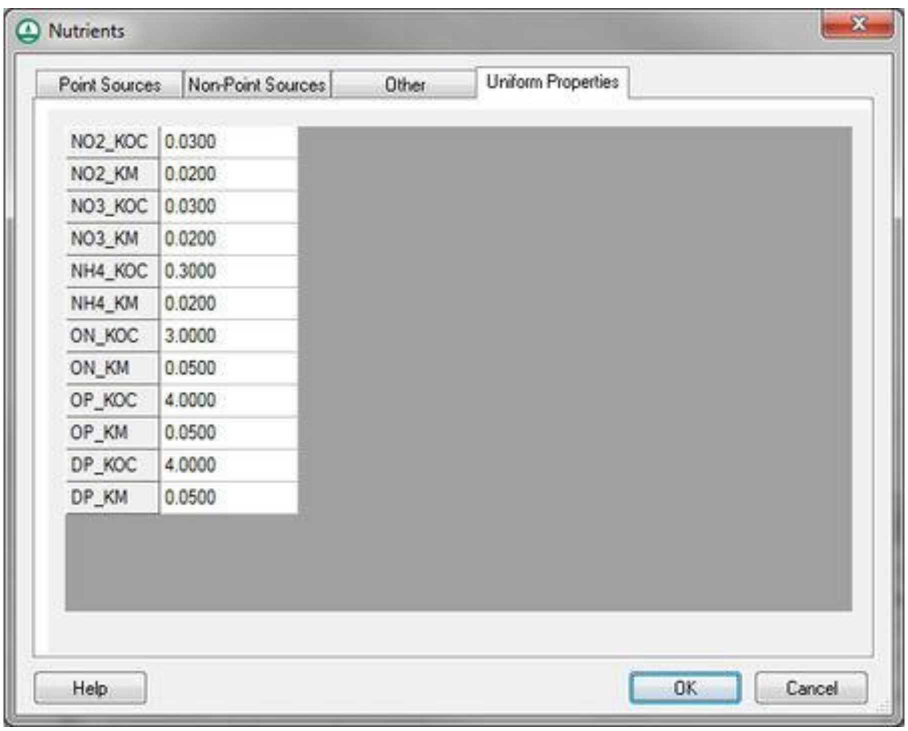
To model nutrients using GSSHA:

1. Toggle on the *Nutrients* option in the *Job Control* .

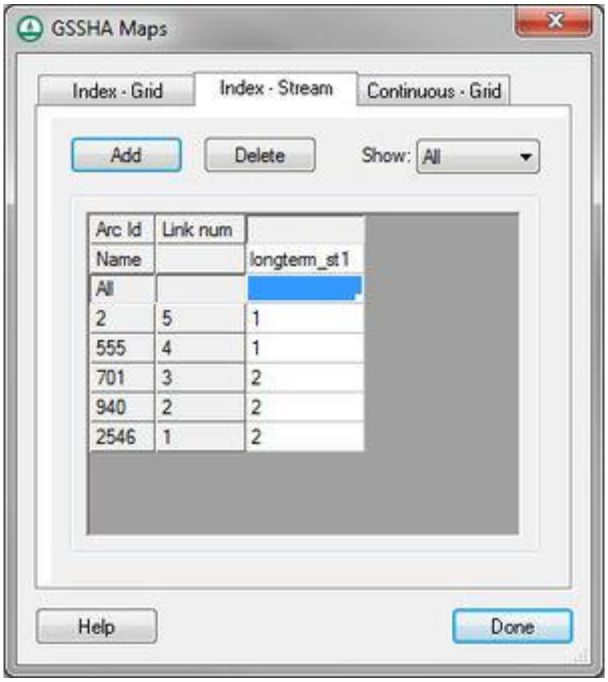


2. Open the *Nutrients* dialog to view the four nutrient tabs: *Point Sources* , *Non Point Sources* , *Other* , and *Uniform Properties.* Enter nutrient concentrations in the Other tab. These concentrations will be applied to rainfall input. The *Uniform* properties tab contains default values for uniform nutrient properties. It's possible to adjust these values.





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 the desired datasets.

6. If wanting to define Point Sources or Non-Point Sources, select the *Nutrients* button in the **GSSHA Job Control** dialog. If defining non-point sources, define a grid-based or a stream-based index map associated with the non-point source. If defining a point source, 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, define a mass curve or a concentration curve for any contaminant source to be modeled.

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.

•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

•A project 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

Primer

Tutorials

User's Manual

•NSM Project Cards

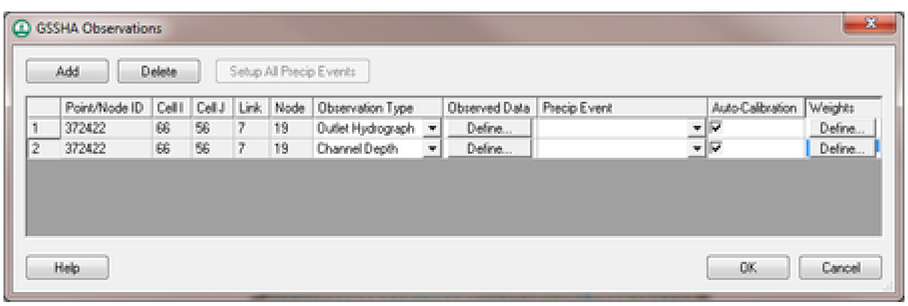
•NSM Aquatic Kinetic Constants

•Point/Non-point Sources

•NSM Map Tables

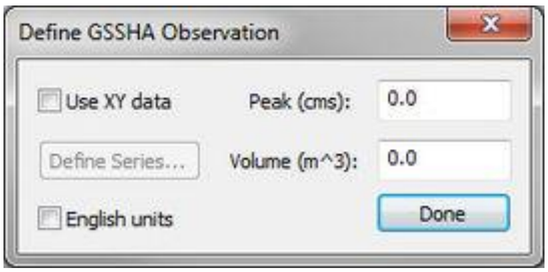
GSSHA Observation Points

Observed data are stored as part of the attributes of a feature point in the GSSHA and the GSSHA Storm Drain coverage types. 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 can be stored for data comparison or can be used with the GSSHA automated calibration algorithms. If you are not running an SCE-based calibration, observation points can be defined anywhere in your model. GSSHA will run an automated calibration using your observed time series data if you have the Auto-calibration toggle box turned on for a calibration. The auto-calibration option is only available for certain types of observation types. Channel-based observations can only use the auto-calibration option at nodes on streams in your GSSHA coverage. Tile drain discharges can be calibrated by defining the observed discharges at junction nodes in the GSSHA Storm Drain coverage. All the observations in the GSSHA and the GSSHA Storm Drain coverage assigned to your model are listed in the Observation dialog when it is accessed from the Calibration Parameters 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 in the *Define GSSHA Observation* dialog. The dialog gives the option to define 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. The point/node ID is also displayed so you can relate each line in the dialog to a feature point. The point/node ID is useful to identify points that correspond with observations if you are displaying all the observations from the GSSHA calibration parameters dialog.



Related Topics

- GSSHA Calibration
- GSSHA Feature Nodes

GSSHAWiki

Primer

- Watershed Delinieation and Grid Construction
- Lake and Channel Routing

Tutorials

User's Manual

- Surface Water Routing

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 datasets 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

Primer

•Post-Processing

- Output Control

Tutorials

User's Manual

•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:

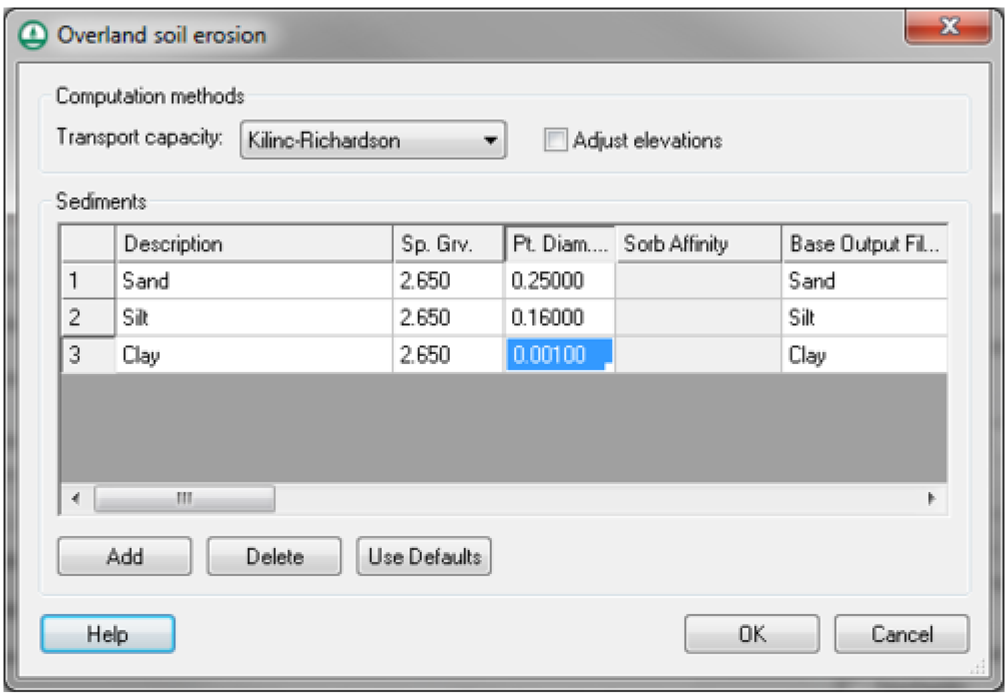
1. Create the sediment types,
2. Create a soil type index map,
3. Define sediment proportions and other relevant parameters for each soil type.

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.

Creating 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

Primer

- Mapping

Tutorials

User's Manual

- Mapping Table File
    - Index Maps
    - Mapping Tables

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 specifying 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
    a) Year
    b) Month
    c) Day
    d) Hour
    e) 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

Primer

- Spatially and Temporally Varying Rainfall
  - Temporally Varying - Spatially Uniform Rainfall
  - Temporally Varying Multiple Gage Rainfall

Tutorials

User's Manual

- Precipitation
  - Spatially and Temporally Uniform Precipitation
  - Spatially and Temporally Varied Precipitation
  - Interpolation Between Gages
  - Interception

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. 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

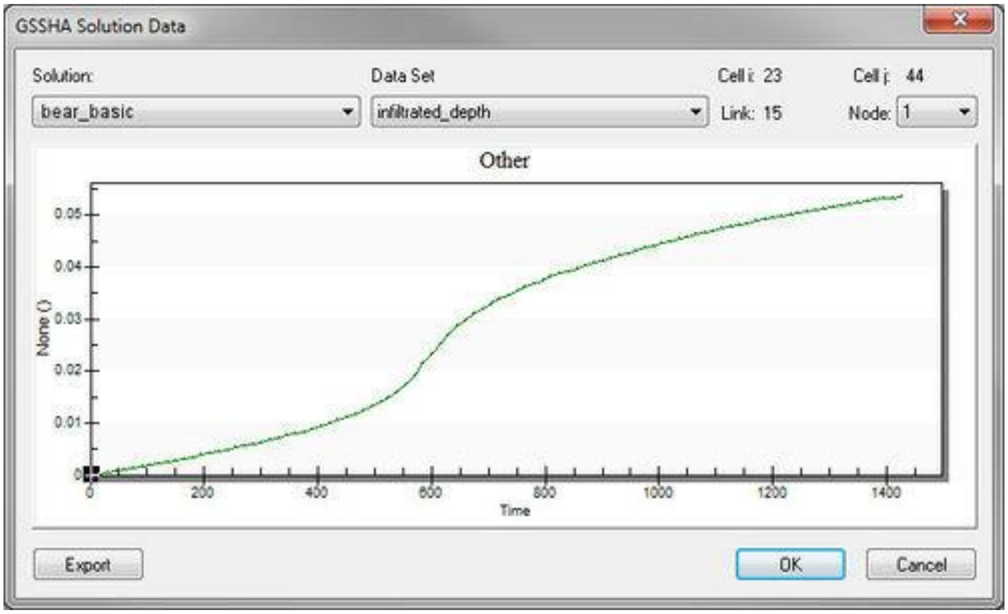
GSSHAWiki

Primer User's Manual

Tutorials

GSSHA Solution Results

The *Solution Results* dialog, accessible from the feature arc *Properties* 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 Arcs

GSSHAWiki

Primer

•Running GSSHA

- Running GSSHA from WMS
- Project File

Tutorials

User's Manual

•Output

- Run Summary File

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

Primer

•Watershed Delinieation and Grid Construction

•Lake and Channel Routing

Tutorials

User's Manual

•Surface Water Routing

Pipe and Node Parameters

Overview

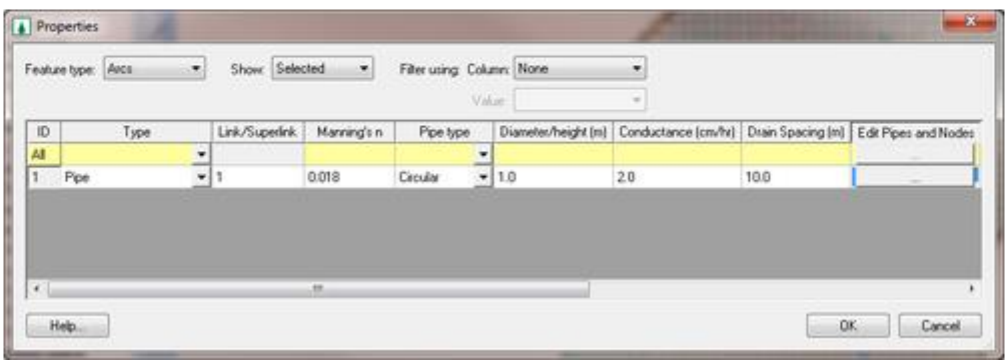
The *GSSHA Pipe and Node Parameters* dialog is accessed by selecting one or more feature arcs, going to the *GSSHA Feature Arc Properties* dialog, and changing the arc type to Pipe. This dialog is used for building and editing the pipes and nodes associated with Super-Links when running a GSSHA Storm and Tile drain simulation.

Help

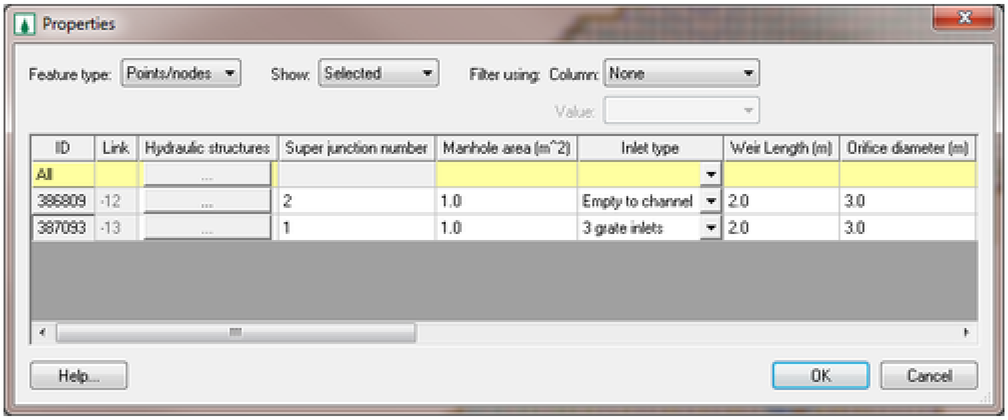
When running a GSSHA Storm and Tile drain simulation, it is a good idea to add more than one pipe (2 nodes) between Super-Junctions. In WMS, Super-Links are represented as arcs and Super-Junctions are represented by the 2 nodes at the end of each arc. Each link may contain 1 or more pipes and 2 or more nodes. Many of the attributes between Super-Junctions and nodes between Super-Junctions overlap. In WMS, define nodes along a Super-Link (between Super-Junctions) by adding vertices along an arc. Redistribute the vertices on the arc using the **Redistribute** command or can manually add vertices along an arc using the **Create Vertex** tool. Normally, use the following steps to define storm or tile drains and their attributes in WMS:

•Create a new GSSHA storm drain coverage and define the 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, the is the option to **Initialize the Pipes from the Arc Geometry** . Clicking on this button creates a pipe for each arc segment on the 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 these transferred data values can be edited. 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 the storm or tile drain network. Deleting the pipes will delete the pipes and nodes from the arc, but 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.

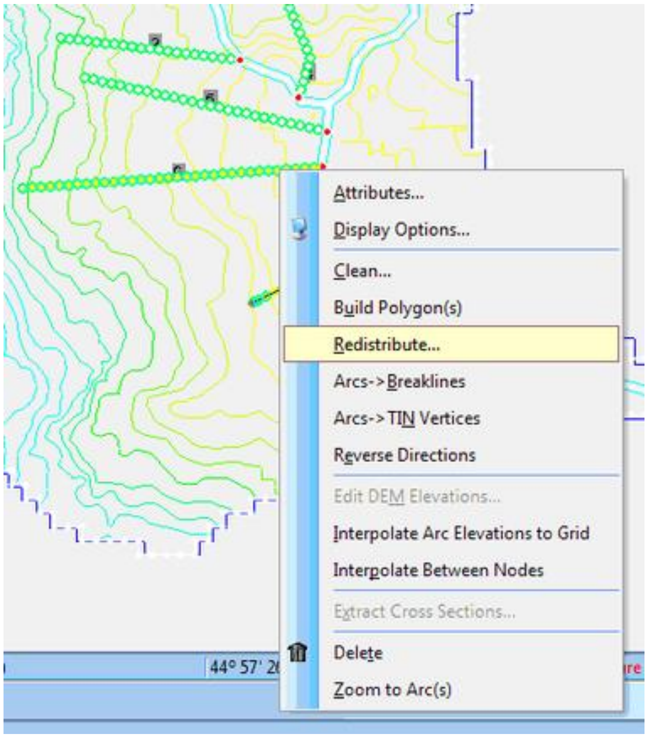
Refer to the GSSHA Subsurface Tile and Storm Drain tutorial for more information about how to setup a storm or tile drain model.

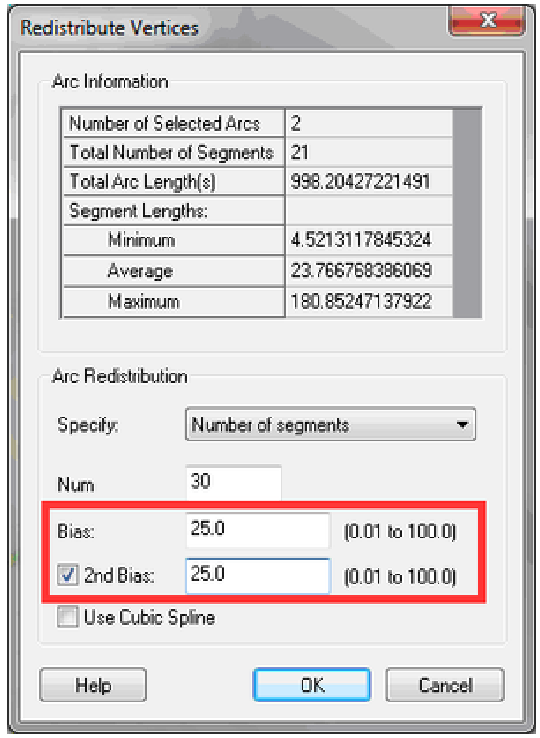
Tools for Editing GSSHA Storm and Tile Drain Data

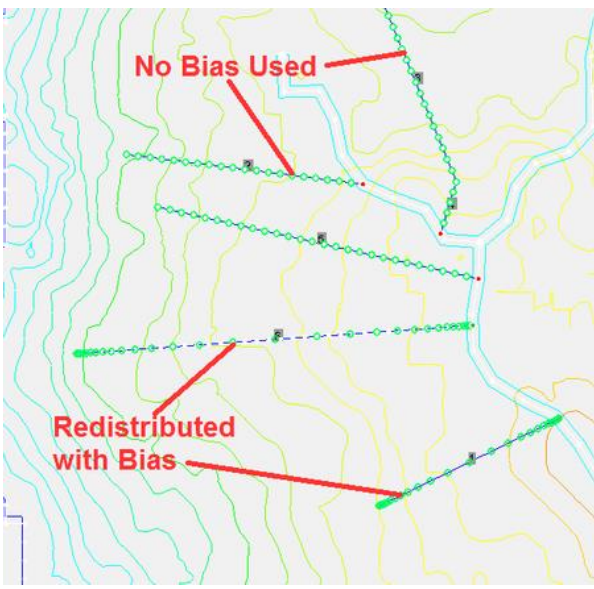
WMS 10.0 and later versions have several specialized tools for editing storm and tile drain data. These tools are described in this section.

Varied node distribution on an arc

The distribution of vertices (nodes) along an arc (superlink) in the GSSHA Storm Drain coverage is important in the storm/tile drain model used by GSSHA. Space nodes closely near junctions and farther apart toward the center of the superlink. WMS has an option under the **Redistribute** option for feature arcs that allows for defining a second bias. Each of the bias values defines the ratio of spacing between vertices at the center of the arc and at the middle of the arc. Use the same bias value along with a specified vertex spacing or a number of vertex segments to define vertices close together at the ends and far apart at the center of the superlink. See the images below.

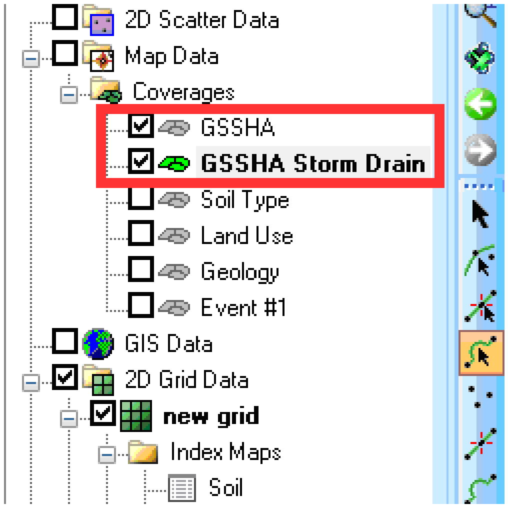






GSSHA Storm Drain coverage

In WMS 10.0 and later, **Pipe** arc types are no longer defined in a GSSHA coverage. They are defined in a *GSSHA Storm Drain* coverage so pipes drawn inside of WMS do not impact stream elevations in the GSSHA coverage.



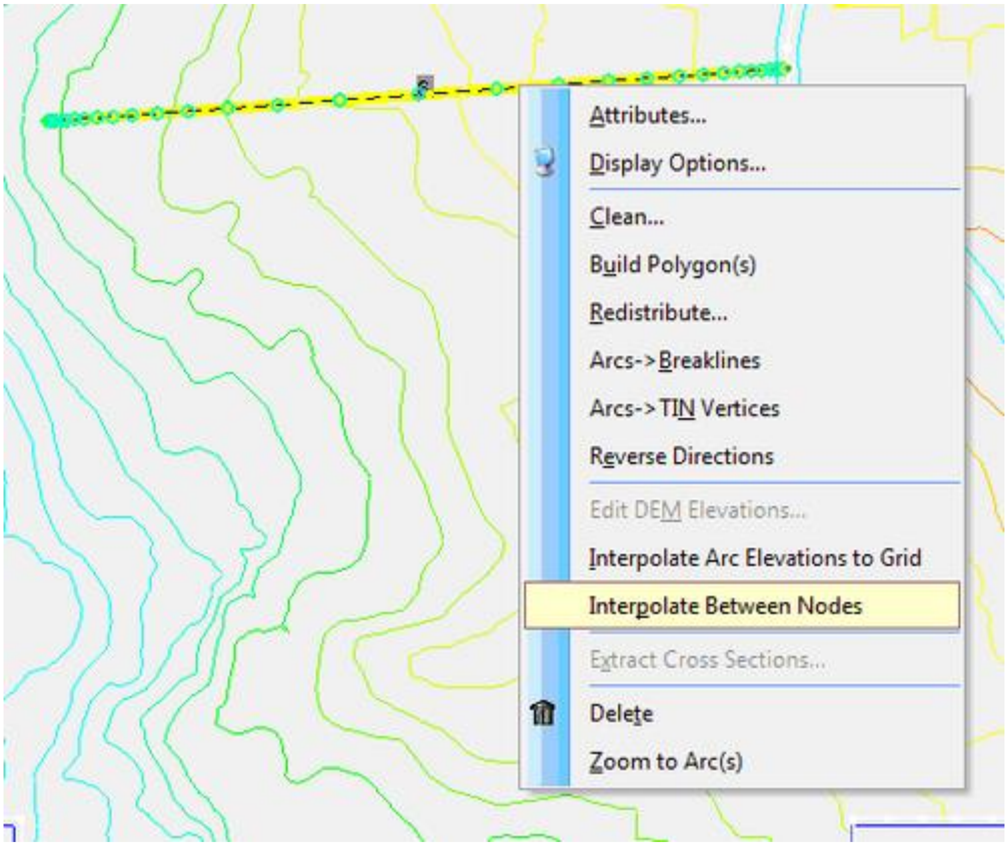
If there is an existing GSSHA project with pipe arcs in a GSSHA coverage, WMS converts these arcs to generic arcs in the GSSHA coverage and copies the pipes into a GSSHA Storm Drain coverage. This transition should work seamlessly, without needing to do any work. Existing files with a GSSHA storm drain coverage can be read and should be able to be re-written and run in GSSHA from the WMS interface. It may be desirable to delete the generic arcs that were pipes and are left in the GSSHA coverage, or leave them there. WMS does not put up any messages when converting a file from the single coverage to the GSSHA and GSSHA Storm Drain coverage model.

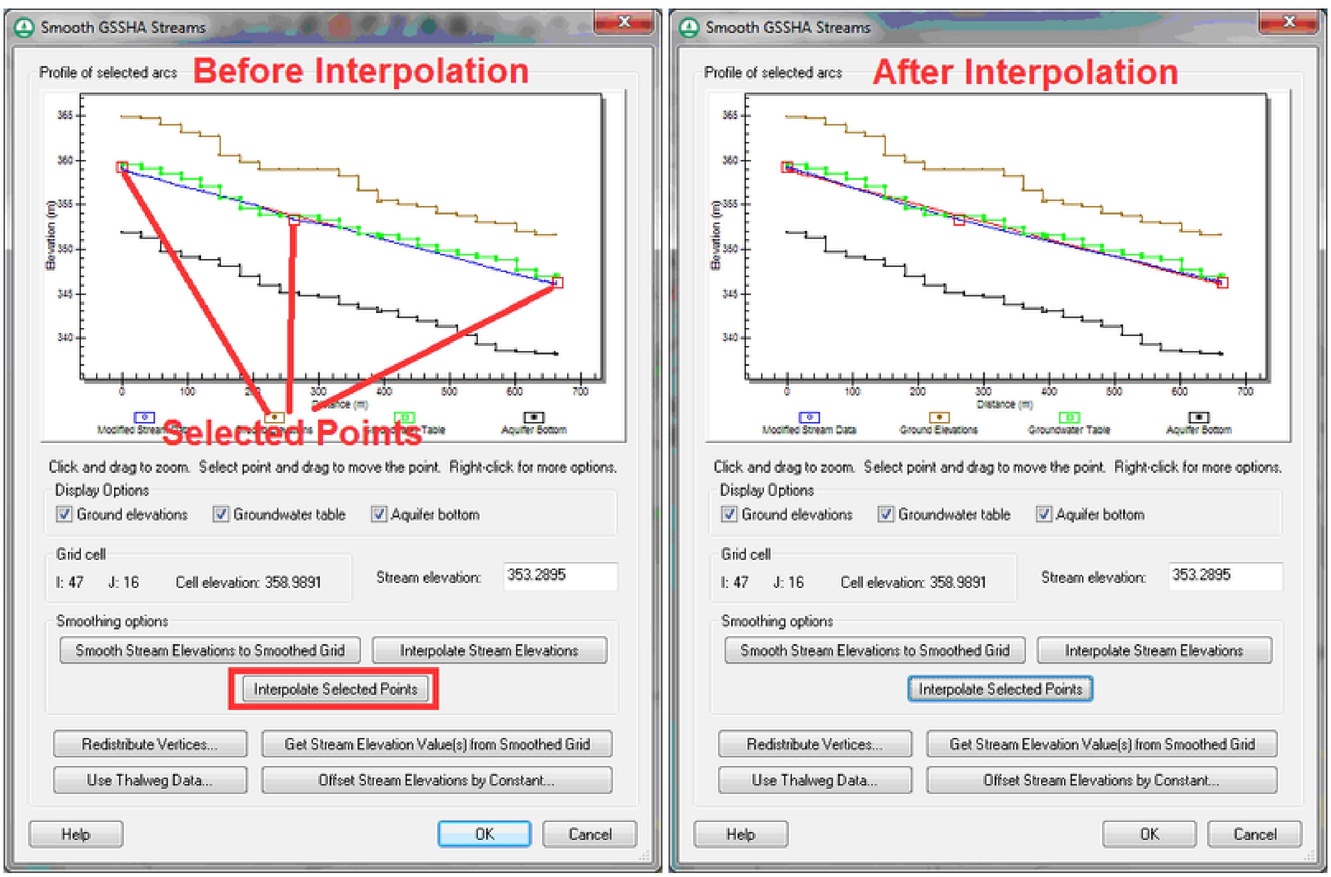
Auto-Updates of Pipes and Nodes and Attributes

WMS 10.0 and later versions automatically update the pipes and nodes for a superlink when a vertex is added or deleted along an arc. When redistributing vertices on an arc, WMS updates the pipes and nodes on the arc. The attributes for the new pipes and nodes are assigned based on the attributes assigned to the arc and the upstream superjunction point before adding vertices to the arc.

Elevation Interpolation Tools

WMS 10.0 and later versions have an **Interpolate Between Nodes** option when right-clicking on arcs. With this command, define an elevation at one or both of the nodes at the ends of the arc and then select this command. This command uses linear interpolation to assign the vertex elevations between the nodes at the ends of the arc. WMS also has a **Smooth Stream/Pipe Arcs** menu command that allows selecting points along a single arc and use linear interpolation between to assign elevations between the points. See the images below.


Attributes...
Display Options...
Clean...
Build Polygon(s)
Redistribute...
Arcs->Breaklines
Arcs->TIN Vertices
Reverse Directions
Edit DEM Elevations...
Interpolate Arc Elevations to Grid
Interpolate Between Nodes
Extract Cross Sections...
Delete
Zoom to Arc(s)

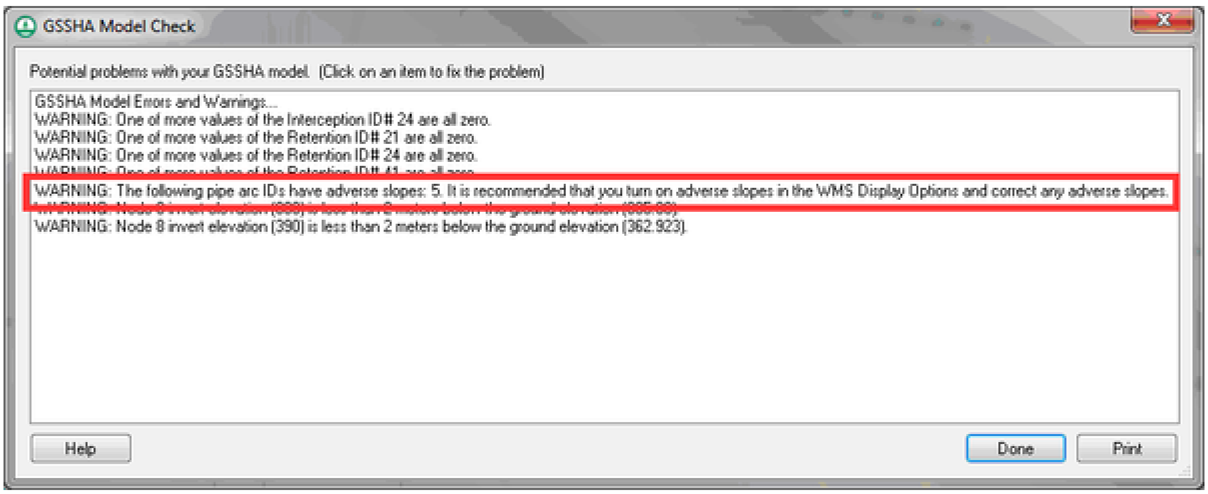

Smooth GSSHA Streams
Profile of selected arcs
Before Interpolation
Selected Points
Click and drag to zoom. Select point and drag to move the point. Right-click for more options.
Display Options
Ground elevations
Groundwater table
Aquifer bottom
Grid cell
I: 47 J: 16 Cell elevation: 358.9891
Stream elevation: 353.2895
Smoothing options
Smooth Stream Elevations to Smoothed Grid
Interpolate Stream Elevations
Interpolate Selected Points
Redistribute Vertices...
Get Stream Elevation Value(s) from Smoothed Grid
Use Thalweg Data...
Offset Stream Elevations by Constant...
Help
OK
Cancel
Smooth GSSHA Streams
Profile of selected arcs
After Interpolation
Modified Stream Data
Ground Elevations
Groundwater Table
Aquifer Bottom
Click and drag to zoom. Select point and drag to move the point. Right-click for more options.
Display Options
Ground elevations
Groundwater table
Aquifer bottom
Grid cell
I: 47 J: 16 Cell elevation: 358.9891
Stream elevation: 353.2895
Smoothing options
Smooth Stream Elevations to Smoothed Grid
Interpolate Stream Elevations
Interpolate Selected Points
Redistribute Vertices...
Get Stream Elevation Value(s) from Smoothed Grid
Use Thalweg Data...
Offset Stream Elevations by Constant...
Help
OK
Cancel

Storm/Tile Drain Adverse Slope Detection

If a segment on the arc has an adverse slope and the option to display adverse slopes is turned on, WMS displays the adverse slope in red. If any of the stream or pipe arcs in a GSSHA model have adverse slopes, the following error message is displayed when running the **Model Check** command under the GSSHA menu:

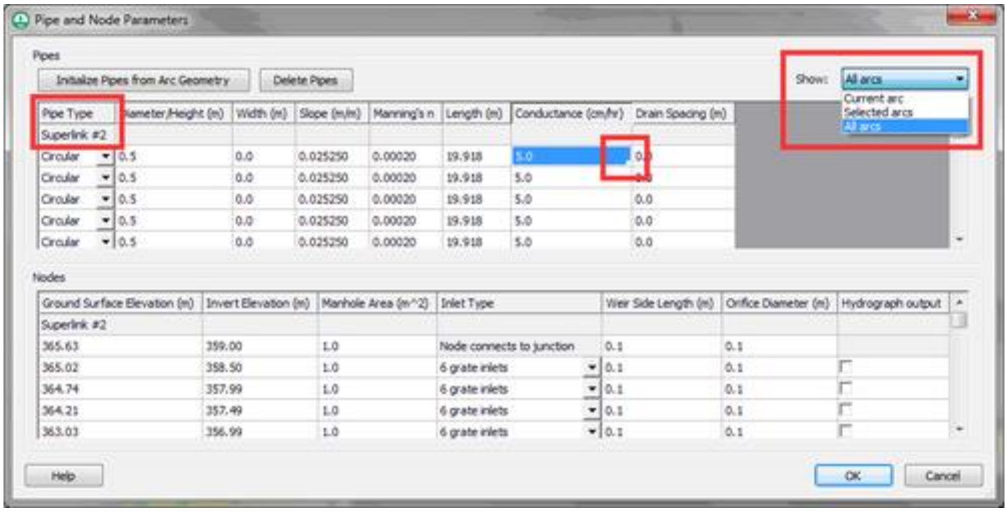
WARNING: The following pipe arc IDs have adverse slopes: <arc number>. It is recommended that to turn on adverse slopes in the WMS display options and correct any adverse slopes.

Then turn on adverse slopes in the display options and correct any adverse slopes. See the images below:



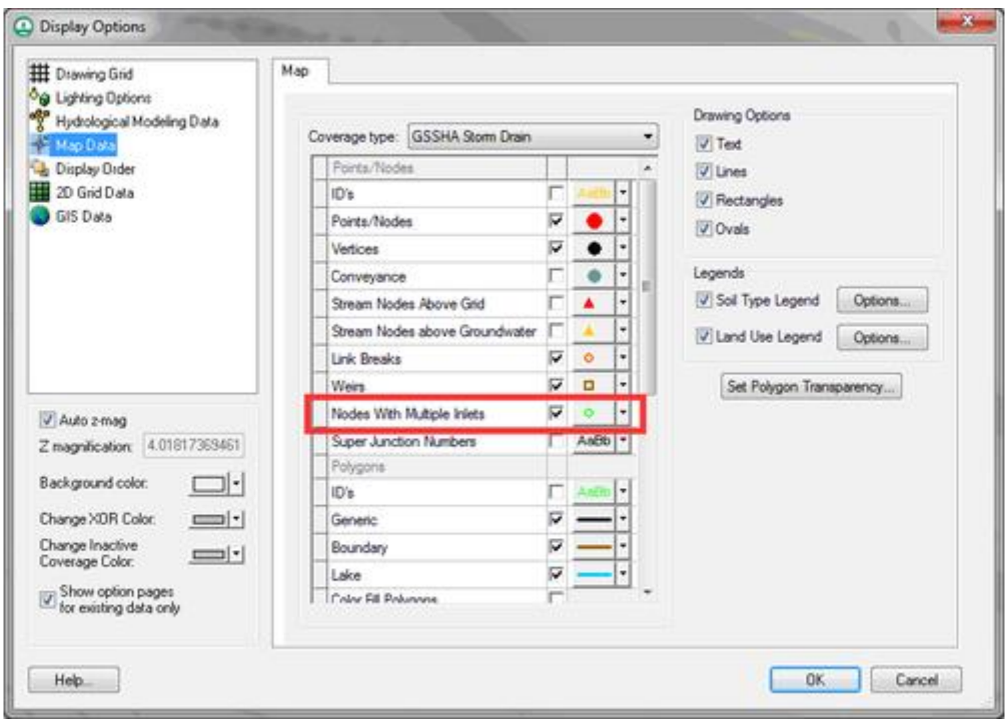
Editing Pipe and Node Parameters for Multiple Arcs

WMS 10.0 and later versions have a combo box at the top of the *Pipe and Node Parameters* dialog that allows displaying only the current arc, selected arcs, or all arcs. If the *Current arc* option is selected, only the pipes and nodes for the current arc are displayed, and so on. If wanting to re-initialize all the pipes and nodes from the arc geometry, select the option to show *All arcs* and select the **Initialize Pipes from Arc Geometry** button. This will re-initialize all the pipes and nodes for all the arcs in the current GSSHA Storm Drain coverage. Note that there's also an option to only do this for the selected arcs. If wanting all the attributes for the displayed arcs to be the same, set the value in the top cell of the dialog. There is a small square in the lower right section of the cell. Double-click on this small square and the attribute will be copied to all the other pipes or nodes that are displayed. The superlink number is displayed at the top of each set of superlink attributes.



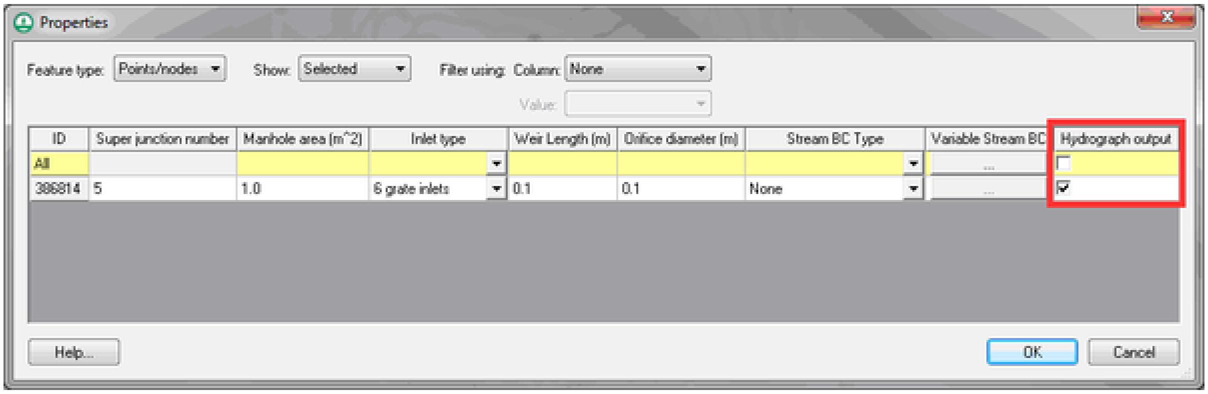
Identify Nodes with Surface Inlets

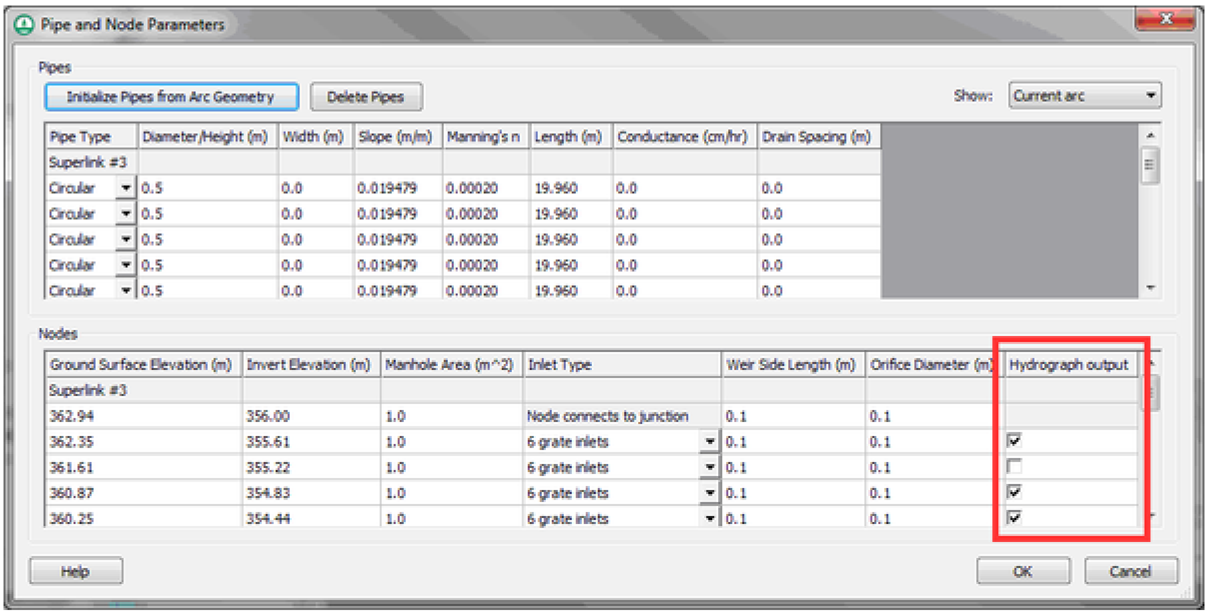
WMS 10.0 and later versions have an option in the WMS *Display Options* that allows displaying a special symbol for any nodes or superjunctions in the GSSHA Storm Drain coverage that have more than zero inlets.

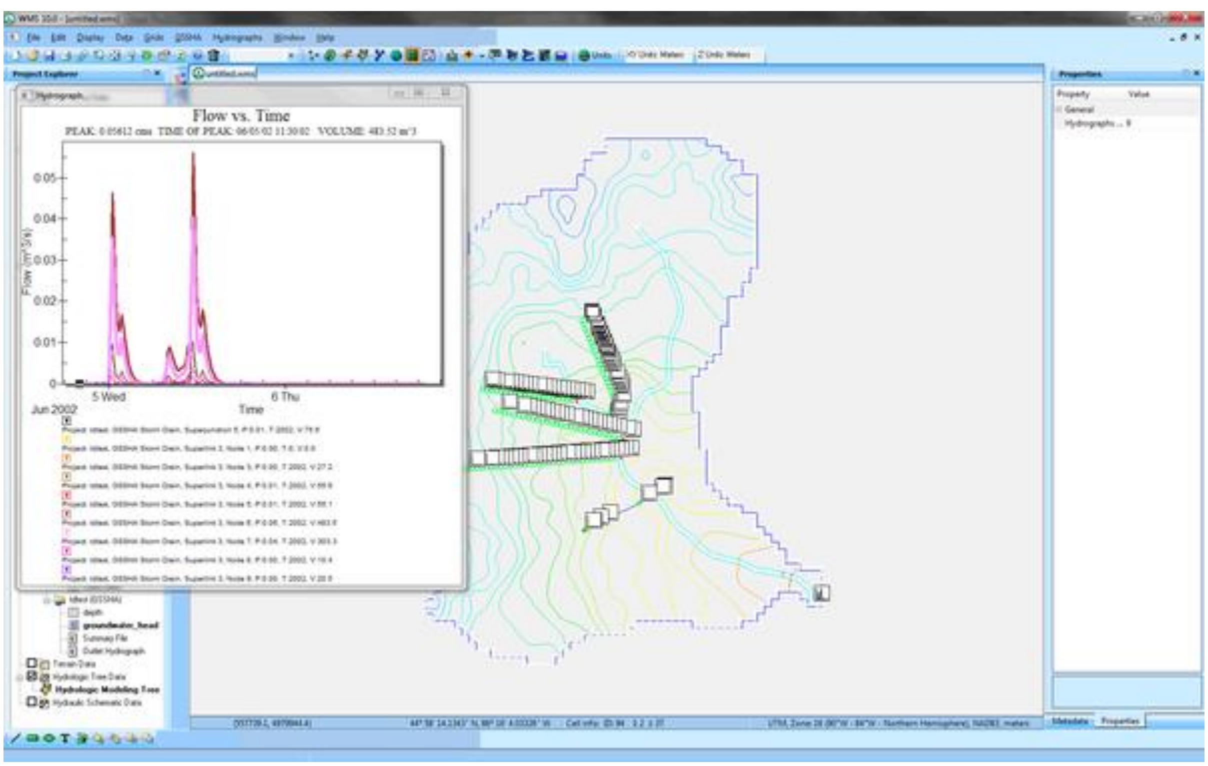


Enhanced Hydrograph Output Options

WMS 10.0 and later versions have a *Hydrograph output* option for superjunctions as well as for each of the nodes in a superlink. GSSHA hydrographs are displayed at node/vertex locations instead of at grid cell centers since WMS reads the hydrographs based on link/node, pipe/node, or superjunction numbers. See the images below:


Properties
Feature type: Points/nodes
Show: Selected
Filter using: Column: None
Value:
ID | Super junction number | Manhole area (m^2) | Inlet type | Weir Length (m) | Orifice diameter (m) | Stream BC Type | Variable Stream BC | Hydrograph output
All
386814 | 5 | 1.0 | 6 grate inlets | 0.1 | 0.1 | None
Help...
OK
Cancel


Pipe and Node Parameters
Pipes
Initialize Pipes from Arc Geometry
Delete Pipes
Show: Current arc
Pipe Type | Diameter/Height (m) | Width (m) | Slope (m/m) | Manning's n | Length (m) | Conductance (cm/hr) | Drain Spacing (m)
Superlink #3
Circular | 0.5 | 0.0 | 0.019479 | 0.00020 | 19.960 | 0.0 | 0.0
Circular | 0.5 | 0.0 | 0.019479 | 0.00020 | 19.960 | 0.0 | 0.0
Circular | 0.5 | 0.0 | 0.019479 | 0.00020 | 19.960 | 0.0 | 0.0
Circular | 0.5 | 0.0 | 0.019479 | 0.00020 | 19.960 | 0.0 | 0.0
Circular | 0.5 | 0.0 | 0.019479 | 0.00020 | 19.960 | 0.0 | 0.0
Nodes
Ground Surface Elevation (m) | Invert Elevation (m) | Manhole Area (m^2) | Inlet Type | Weir Side Length (m) | Orifice Diameter (m) | Hydrograph output
Superlink #3
362.94 | 356.00 | 1.0 | Node connects to junction | 0.1 | 0.1
362.35 | 355.61 | 1.0 | 6 grate inlets | 0.1 | 0.1
361.61 | 355.22 | 1.0 | 6 grate inlets | 0.1 | 0.1
360.87 | 354.83 | 1.0 | 6 grate inlets | 0.1 | 0.1
360.25 | 354.44 | 1.0 | 6 grate inlets | 0.1 | 0.1
Help
OK
Cancel


Flow vs. Time
0.05
0.04
0.03
0.02
0.01
0
5 Wed
6 Thu
Jun 2002
Time

Related Topics

•GSSHA

Radar Rainfall

WMS 8.1 and later allows reading and utilizing radar rainfall data from the National Weather Service. Data can be used 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 normal. If desired, use the *hydrologic modeling wizard* . After having created a model, define the precipitation in the *HEC-HMS Meteorological Parameters* dialog. Once here, do the following steps.

1. Change precipitation method to either *User Gage Weighting* or *Gridded Precipitation* . *User Gage Weighting* will only allow creating rain gages from the radar data, while *Gridded Precipitation' will allow creating gridded precipitation in a DSS file as well as creating rain gages.*
   a) To use *Gridded Precipitation', there must be a 2-D Grid defined.*
2. Click on the button, which depending on the 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 the rainfall data. Follow the instructions on the GSDA site to obtain Nexrad Arc/Info ASCII grids.
4. Select the datatype to convert the data to: *Incremental Distribution Rain Gages* , *Total Storm Rain Gages* , or *DSS* .
5. Click **Add Files..** and select the radar precipitation files.
   a) If selecting the last file first, and select the first file while holding *SHIFT* the data will come into WMS in the proper order.
   b) It is very important to arranges the data in chronological order. Once having brought the data into WMS, arrange the data's order through the use of the *Move Up* , *Move Down* , and *Reverse Order* .
   c) If 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) .
   d) Note that the dialog can be resized to view the entire filename.
6. If creating a DSS file, select the folder to export the DSS file by clicking on the **Select Folder** button. Also change the *Project name* .
7. Select the starting date and time of the precipitation grids being used.
8. If desired, change the time interval of the rainfall grids by clicking on the **HMS Job Control** and changing the project's *Time interval* . The Rainfall time step should match the project's time step.

9. If wanting to view the data in milimeters rather than inches, check the *Convert inches to millimeters* . If creating rain gages, then it is possible to also create 2D grid rainfall dataset. This will allow visualizing the 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 the 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

To use radar rainfall data with GSSHA, create an GSSHA model as normal. If desired, use the hydrologic modeling wizard . After having created a model, define the precipitation in the *GSSHA precipitation* dialog. Once here, do the following steps to define Radar precipitation:

1. Change the precipitation type to *Nexrad Radar*
2. Select **Import Radar Data...**
3. Obtain the rainfall data by following the instructions on the GSDA site
4. Select the **Add Files...** button
5. Select the datatype to convert the data to: *Incremental Distribution Rain Gages* or *Total Storm Rain Gages* .
    a) *Incremental Distribution Rain Gages* corresponds to a GSSHA *GAGE* type rain gage.
    b) *Total Storm Rain Gages* corresponds to a GSSHA *ACCUM* type rain gage.
6. Click **Add Files..** and select the radar precipitation files.
    a) If selecting the last file first, and select the first file while holding *SHIFT* the data will come into WMS in the proper order.
    b) It is very important to arrange the data in chronological order. Once having brought the data into WMS, arrange the data's order through the use of the *Move Up* , *Move Down* , and *Reverse Order* .
    c) If 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) .
    d) Note that the dialog can be resized to view the entire filename.
7. Select the starting date and time of the precipitation grids to be used.
8. Change the time interval of the rainfall grids by clicking on the *HMS Job Control* and changing the project's *Time interval* . The Rainfall time step should match the project's time step.
9. Normally leave the toggle box *Convert inches to millimeters* checked. GSSHA will be expecting the precipitation in millimeters.
10. If wanting to view the data as a 2D grid rainfall dataset, check the toggle box. This will allow visualizing the 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 the computer to finish processing the radar rainfall data.
12. Click **OK** to exit the *HMS Meteorological Model* dialog.

Frequently asked questions (FAQ's) related to using radar rainfall data

1. **Q:** xmswiki.com says you have to create a 2D grid to use "gridded precipitation", but why do you need it if you're producing rain gages and not gridded rainfall from conversion process? Do you need it to convert to/produce gridded precipitation rather than to use gridded precipitation but convert to rain gage data?

    - **A:** A grid is not needed to create rain gages. You do need a grid if you're converting to a gridded DSS file for HMS or if you're using your rainfall with a GSSHA model. There's a **Radar Data→Rain Gages** command under the *User Gage Weighting* option in the *HMS meteorological model* dialog that you can use to convert your rainfall to rain gages.

2. **Q:** Files included in DSS listed above list of files to import, but not all outputs are DSS. GSSHA doesn't seem to use DSS at all. In HMS, does it convert to DSS for rain gage format as well as gridded format?

    - **A:** If you build an HMS ModClark model, you need to convert your rainfall to a DSS grid. If you're using any other HMS model, convert your radar rainfall to regular rain gages. GSSHA does not use DSS files at all.

3. **Q:** When converting to rain gage type data, why would you turn on the *create 2D grid rainfall* option?

    - **A:** You never need to turn this option on unless you want to visualize the rainfall data on your grid.

4. **Q:** Why is "create 2D gridded rainfall" grayed out when converting to "gridded precipitation" format?

    - **A:** Create 2D rainfall is grayed out when converting to a DSS file because WMS always creates a 2D grid rainfall dataset when creating a DSS file. WMS converts this rainfall dataset to a gridded DSS file.

5. **Q:** Why is there the option to convert to gridded rainfall data with HMS when HMS doesn't deal with grids like GSSHA does?

    - **A:** The HMS ModClark model does use grids. See the HMS ModClark tutorials on the WMS Learning Center on aquaveo.com.

6. **Q:** On xmswiki.com it says " *Gridded Precipitation* will allow you to create gridded precipitation in a DSS file as well as creating rain gages." Does this mean it will automatically produce both gages and gridded outputs, or does it have the option to do gridded instead of gage, or do you choose one or both later in the process?

    - **A:** If you have the *HMS Gridded Precipitation* option selected, you must create gridded DSS files. You can also create rain gages in the *Convert Grids* dialog, but these gages won't be used with your HMS model if you have the *Gridded precipitation* option selected. If you have the *User Gage Weighting* option selected in HMS, you can use the gages. If using Gridded Precipitation, you need to define a grid in WMS and use the ModClark transform method as described in the tutorials.

7. **Q:** *HMS Gridded Precipitation* option: convert to format default is DSS, but the two gage options are also available in the drop down. Is it wrong to choose one of the gage options even though we said it was gridded? Is this an acceptable alternate way to produce gage data?

    - **A:** If you're using the Gridded Precipitation option, you need to create gridded DSS files. HMS won't run if you're using gridded precipitation and you don't create the DSS files.

8. **Q:** When 2D grid rainfall datasets (cumulative and incremental) are created, then you save, close and re-open the project in WMS, the 2D datasets both disappear. Why? Where is the data in these datasets saved (what file)? Do you have to re-convert every time you open the files? (I could not replicate the disappearing datasets with tutorial files)

- **A:** We should save the precipitation datasets with your WMS project. Make sure you are using the most recent version of WMS. If not, updating the version of WMS you're using might fix the problem with the disappearing datasets. Let me know if you continue to have problems with this. You should not need to regenerate your precipitation datasets every time you start up WMS.

9. **Q:** When calculating 2D grid data with XMDF format files (already in mm, no conversion needed), summary file shows units of inches, but numbers are actually mm values.

   - **A:** This bug has been fixed. When WMS is next updated on the website, this problem should go away. Let me know if you have any further concerns after you get the update. You should be able to get the nightly build or the latest build from our website and it should be fixed in that version (Mar 13, 2014).

10. **Q:** How does WMS define the resulting rain gage data? What file is this data saved to? How can we verify the conversion and assignment of the rainfall data (doubts due to question #8)?

    - **A:** For GSSHA, the rainfall is saved to a rain gage file (.gag). For HMS, the gridded rainfall is saved to a gridded DSS file and regular rainfall gages would be saved as time series data in your DSS file. Use HEC-DSSVue (available for free from HEC) to view DSS file information. DSSVue has limited viewing capabilities for gridded DSS data. I think the summary report will give you a correct idea of whether the conversion was correct, and in your case, since the report had bad data, you can assume the conversion was not correct. Use the nightly build to get the correct conversion from XMDF files.

11. **Q:** After conversion, in 'the 'Precipitation *dialog, why do we have to choose either Thiessen or inverse distance methods? Both seem (to Dan, I have no idea) to apply to HMS but not GSSHA.*

    - **A:** GSSHA uses rainfall gages from a rain gage coverage in the map module. When you save the GSSHA project, these gages are defined in the .gag file. GSSHA needs a way to interpolate the gage data to each of the cells on your grid since the resolution of these cells can be different than the resolution of your radar data and your rainfall gages may be scattered in a manner that is not set to a grid (they may be simple rainfall gage stations).

12. **Q:** Wiki details using NTP input data, but MPE (XMDF format) is more common and useful, but wiki doesn't talk about it at all.

    - **A:** I suspect you can use the XMDF format MPE data even though the documentation does not mention it. If you have XMDF or ASCII grid files, the precipitation just has to be in either incremental or cumulative format. If you have incremental data, the conversion goes a little bit faster. If there's a time period where there is no rainfall, sometimes the cumulative radar rainfall resets, and WMS accounts for the reset when converting to cumulative or incremental data. It takes time for WMS to do this. Get the most recent WMS build since there's been some bug fixes with the conversion (Mar 13, 2014) and the conversion has been sped up significantly.

13. **Q:** What does the multiplication factor do? Is it for adjusting radar data to align with observed data?

    - **A:** All the values in your radar data are multiplied by the multiplication factor. The default is 1.0, meaning the value are not changed. I think we added this because we were not sure about the different formats when we first implemented the tool, but you should not need it now. But if you end up with the wrong units, you can use the multiplication factor to correct the units of your converted data.

What happens when I click OK in the Convert Grids dialog

This depends if 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 data is selected 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 selected time period. 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 first time step of the storm is not selected, the first grid will contain precipitation that did not occur during the selected time period. This step creates a copy of the initial values of the selected data.

4. Remove Initial Grid: This step removes the precipitation from all the time steps that occurred prior to the first selected time step.

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 selected data 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 selected time period. 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 the first time step of the storm is not selected, the first grid will contain precipitation that did not occur during the selected time period. This step creates a copy of the initial values of the selected data.

4. Remove Initial Grid: This step removes the precipitation from all the time steps that occurred prior to the first selected time step.

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 #####.#### time step: 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.

Related Topics

•GSSHA

6.3. HY-12

HY-12

The US Federal Highway Administration's HY-12 is a DOS-based storm drain analysis program that can be used for designing inlets, pipes, and the general layout of a storm drain network. An HY-12 model can be generated by drawing the proposed pipe and inlet locations in a storm drain coverage. Then the map module locations are converted to a 1D schematic where the HY-12 model parameters are defined. Many of the HY-12 computations, such as channel calculations, curb and gutter calculations, and rational method computations, are based on computations in FHWA's Hydraulic Toolbox software. Refer to both the Hydraulic Toolbox and the HY-12 documentation to learn about the specific computation methods used in HY-12.

The HY-12 model is included will all paid editions of WMS. WMS 10.1 and later versions include the ability to run a simple HY-12 model (fewer than 50 nodes) without a license of WMS.

HY-12 Menu

The *HY-12* menu has the following commands:

•**Import HY-12 File**

•**Export HY-12 Export**

•**Run HY-12** – Opens the *Run HY-12 Simulation* dialog.

•**Edit Project Parameters** – Opens an *Edit Properties* dialog where parameters for the project can be specified.

•**Edit Parameters** – Opens an *Edit Properties* dialog where parameters for nodes and links can be specified.

•**Edit Elevations** – Opens a dialog that allows editing link and node elevations between selected links and/or nodes.

•**View Detailed Link/Node Output** – Creates a text file describing the detailed link and node computation results.

•**View Hydrographic Plots** – Shows hydrograph plots for the selected link and/or node (Unsteady flow simulations only).

•**View EGL and HGL Plots** – Brings up the *EGL and HGL Profiles* dialog.

•**Link Outlets and Inlets** – Brings up the *Link Storm Drain and Drainage Nodes* dialog.

•**Assign Elevations** – This group of commands will assign elevation data from a TIN to each node and link. Each command brings up a corresponding *Select Elevation Source* dialog where the elevation source data can be selected.

- To Ground
- To Access Hole

- To Inlets
- To Outlets
- To Channels
- To Gutters
- To Pipes
- To Pipe Storage

•**Assign Lengths and Orientations**

•**Assign Hydrology Data**

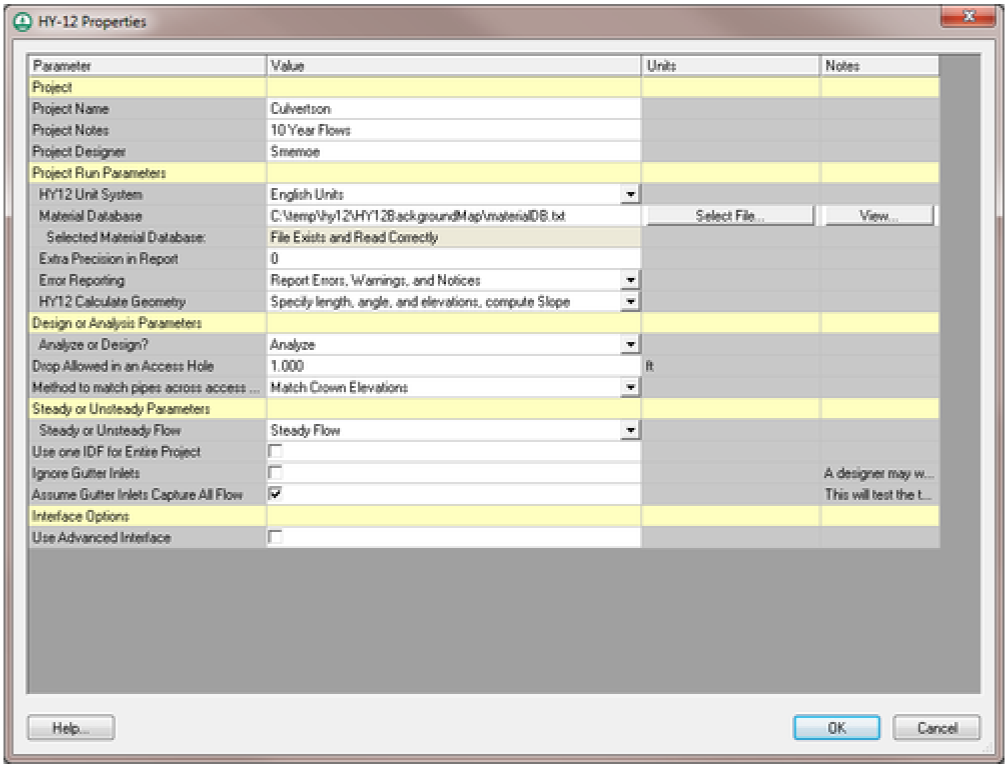
- Area – This command assigns the computed basin areas upstream from the outlet points when having linked to the rational method computations at each node.
- Runoff Coefficient – This command areaweights the runoff coefficients for the sub-basins assigned to each outlet and assigns the area-weighted runoff coefficient to the rational method computations at each node.
- Time of Concentration – This command computes the average time of concentration for the sub-basins assigned to each outlet and assigns this time of concentration to the rational method computations at each node.

HY-12 Parameters

The HY-12 Model has several *HY-12 Properties* dialog where parameters can be defined.

HY-12 Project Properties

The *HY-12 Properties* is reached by using the **Edit Project Parameters** command in the *HY-12* menu.



HY-12 Node and Link Properties

Select either a node or a link using the **Select Hydraulic Node Tool** or the **Select Hydraulic Link Tool** then right-click. In the right-click menu select the **Edit Parameters** command. This will bring up the *HY-12 Properties* dialog for Nodes and Links. This dialog can also be reached by using the **Edit Parameters** command in the *HY-12* dialog.

There are two versions of the HY-12 link/node properties dialog. The simplified interface provides an intuitive way to edit link and node information and many projects will only require using this interface. A person can add pipes or channels to links in the simplified interface. A person can also add Rational method calculations, inlets, access holes, or outfalls to nodes in the simplified interface. Using the simplified interface, it's not necessary to define advanced attributes that are not normally needed in an HY-12 storm drain simulation. A useful feature in the simplified link/node properties dialog is the ability to sort data based on any of the attributes. For example, if wanting to find any pipes with negative slopes, it's possible to sort links based on slope and any link with a negative slope would show up at the top of the list.

If needing to define structures that are not supported in the simplified interface or if needing to define advanced structure parameters, turn on the **Use Advanced Interface** option in the **HY-12 Project Parameters** and edit the project parameters using the advanced interface.

Information on items in the Advanced interface dialog

- **Attribute Type** – Chooses between viewing *nodes* or *links* .
- **Show** – Searches between *all* or *selected* nodes or links.
- **Filter using: Type** – Views nodes or links based on the structure type.
- **Group Name** – Not used
- **Value/HY-12 ID** – Shows the node or link value and the corresponding HY-12 ID.

•**Structure Type** – Assigns the structure type for the node or link. Selecting the **New** button will assign the structure type. The following structure types are available:

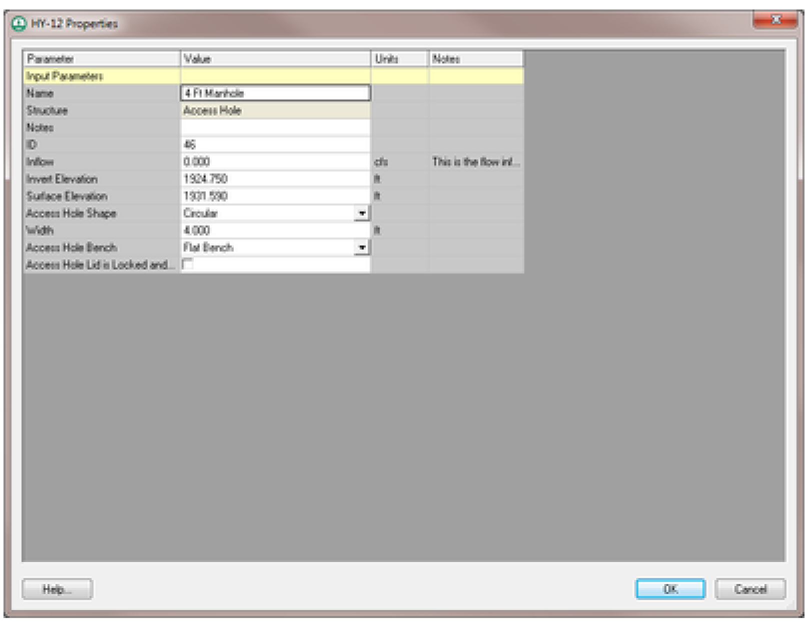
Node Structure	**Link Structure**
Access Hole	Channel
Gutter Inlet	Gutter
Junction	Pipe
Minor Loss	Pipe Storage
Outfall	
Pump Station	
Rational Method Basin	
Reservoir	
Transition	

•**Structure Name**

- New – Clicking on this button will assign the structure type to the selected nodes or links.

•**Define Structure**

- Define – Selecting this button will bring up a *HY-12 Properties* dialog where definitions for the structure type can be entered.



•**Structure Downstream** – Shows structures assigned as downstream structures.

•**Delete Structure**

- Delete – Removes the assigned structure type.

Once parameters have been assigned for nodes and links, double-clicking on a node or link will bring up a *HY-12 Properties* dialog showing the attributes for that node or link.

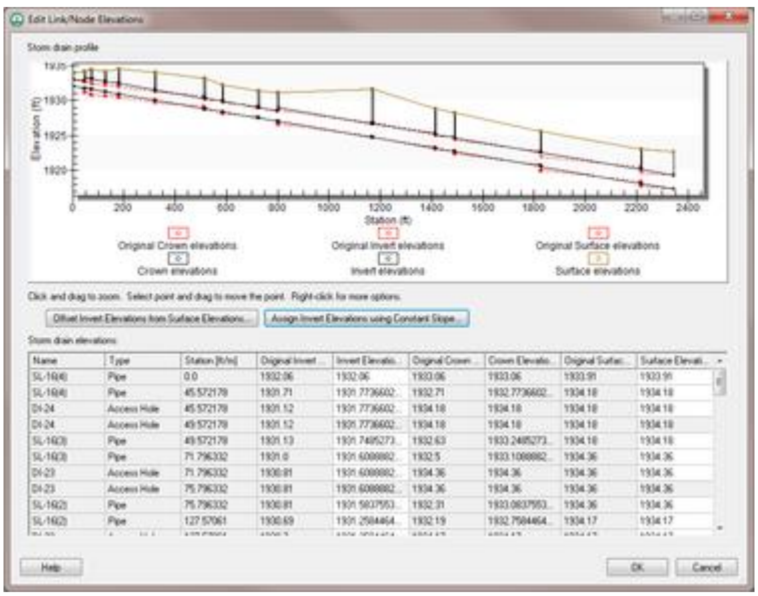
Edit Elevations

The HY-12 **Edit Elevations** menu item is a useful tool for editing elevation profiles between selected links or nodes. To edit elevations between two links or nodes, select the upstream and the downstream points for editing elevations and then select *HY-12* | **Edit Elevations** . The *Edit Link/Node Elevations* dialog will appear. In this dialog, modify elevation values graphically from the plot window or edit individual elevation values in the spreadsheet below the plot.

There are two buttons that may be useful for defining elevations in the storm drain network.

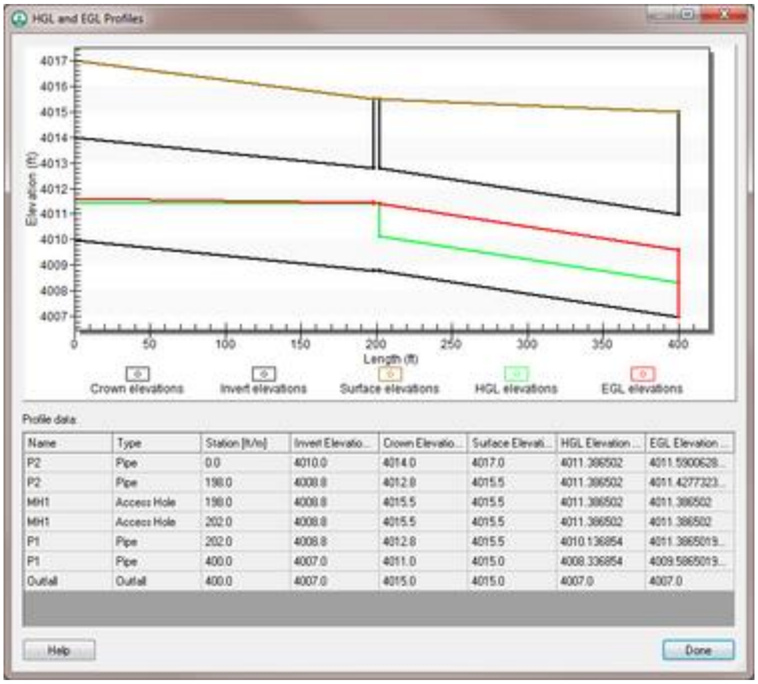
Offset Invert Elevations from Surface Elevations allows offsetting pipes from the ground surface using either a given cover depth or a given invert depth. A prompt will appear asking which of these options to use as well as the distance of the crown or invert below the ground after clicking on the button.

Assign Invert Elevations using Constant Slope assigns pipe and node elevations to the storm drain profile based on a starting elevation as well as a slope. By default, the starting elevation is the invert elevation of the most upstream selected point and the slope is the average slope of the storm drain network. A prompt will appear for these two values after pressing the button.



HGL and EGL Profiles

A HGL/EGL profile plot can be created after running the HY-12 model, selecting two links or nodes, and then by using the **View HGL and EGL Plots** command in the *HY-12* menu.



Name	Type	Station [ft/m]	Invert Elevatio...	Crown Elevatio...	Surface Elevati...	HGL Elevation ...	EGL Elevation ...
P2	Pipe	0.0	4010.0	4014.0	4017.0	4011.386502	4011.5900628...
P2	Pipe	198.0	4008.8	4012.8	4015.5	4011.386502	4011.4277323...
MH1	Access Hole	198.0	4008.8	4015.5	4015.5	4011.386502	4011.386502
MH1	Access Hole	202.0	4008.8	4015.5	4015.5	4011.386502	4011.386502
P1	Pipe	202.0	4008.8	4012.8	4015.5	4010.136854	4011.3865019...
P1	Pipe	400.0	4007.0	4011.0	4015.0	4008.336854	4009.5865019...
Outfall	Outfall	400.0	4007.0	4015.0	4015.0	4007.0	4007.0

Link Storm Drain and Drainage Nodes

The option to link storm drain and drainage coverage nodes is a useful feature if wanting to use the rational method data from the hydrologic model as an input to the HY-12 model. After defining the rational method information, use this menu item to link rational method information at the outlets in the hydrologic model to inlets in the HY-12 model.

- Storm Drain Nodes
- Drainage Nodes
- Linked Nodes
- Link Nodes
- Auto Link
- Tolerance
- Unlink Nodes

Run HY-12

The **Run HY-12** command will bring up the *Run HY-12 Simulation* dialog where specifications can be reviewed. Clicking **OK** on this dialog will run the model. HY-12 simulations with more that 50 nodes cannot be run unless there is a license to WMS.

Related Topics

- Hydraulic Modeling
- Models
- Storm Drain

6.4. Hydrologic Engineering Center-1 (HEC-1)

HEC-1

HEC-1 (Hydrologic Engineering Center 1) was developed by the US Army Corps of Engineers to estimate river flows as a result of rainfall. WMS is able to import, create, run, and visualize HEC-1 projects.

The model has the following features:

- Precipitation: storms typically defined using stations.
- Snowfall/melt can be simulated using simplified energy budget equations.
- Multi-flood analysis (calculates results for multiple fractions of design storm).
- Dam-break scenarios (overtopping and/or failure of structures).
- Flood damage assessment.
- Flood control system optimization (cost-benefit analysis).

Limitations:

- It is assumed that subbasins can be represented by areally averaged parameters.
- Limited to single storm, mostly because soil moisture changes during non-precipitation periods are not modeled.
- Stage is not accurately predicted, only discharge.
- Reservoir and stream routing methods used in HEC-1 are simplistic.

HEC-1 Menu

When the HEC-1 model is the active model, the *HEC-1 menu* becomes active. The menu has the following commands:

- **Open HEC-1 File** – Reads in existing HEC-1 files.
- **Save HEC-1 File** – Generates a HEC-1 input file.
- **Check Simulation** – Launches the HEC-1 Model Check .
- **Run Simulation** – Launches the HEC-1 model run .
- **Read Solution** – Opens the HEC-1 solution files.
- **Job Control** – Opens the *HEC-1 Job Control* dialog.
- **Edit Parameters** – Opens the *Edit HEC-1 Parameters* dialog.
- **HEC-1 Rain Gages** – Opens the *Rain Gage Properties* dialog.
- **Stochastic Modeling** – Launches the *Stochastic Run Parameters* dialog.

•**Run Stochastic Model** – Launches the stochastic model simulation .

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 running 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 it may be necessary to define it for all hydrograph stations before being 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 be aware of differences that might appear from the original file and the one created by WMS.

Model Check

The **Model Check** command should be issued once all necessary HEC-1 data has been defined. It will report any possible errors/inconsistencies in the 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. 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. It's recommended to report any additional checks that might be made while working through various problems.

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.

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, run the model checker which will help 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 which allows specifying 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. View these results by reading this file with the **Open** command from the *Hydrographs* menu.

Once these files have been defined and selecting **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 the HEC1.EXE file is included in AUTOEXEC.BAT.

If HEC-1 does not run to a successful completion, view the ASCII output file using the **View File** command in the *File* menu.

External Links

- US Army Corps of Engineers Hydrologic Engineering Center

Related Topics

- HEC-1 Job Control
- HEC-1 Parameters

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 the **Done** button is selected. Therefore, it is possible to select additional, or other hydrograph stations so that data for that object may be edited without exiting the dialog. 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

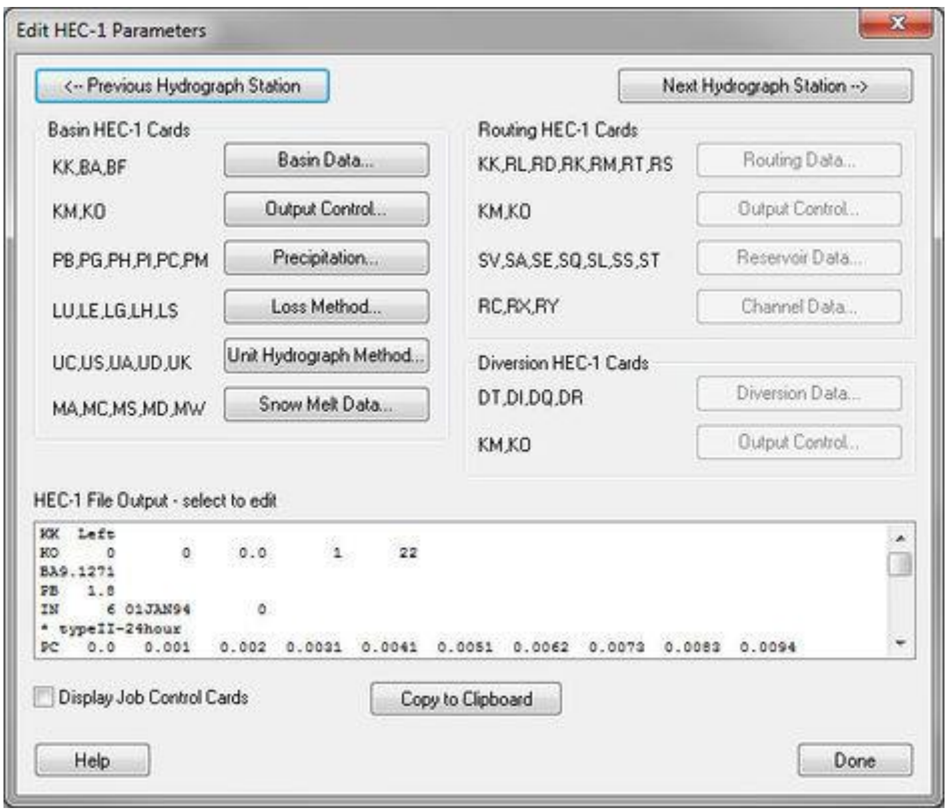
Related Topics

•Job Control Data

•Model Check

•Running an HEC-1 Analysis

Basin HEC-1 Cards



When a basin is selected in the HEC-1 model, selecting *HEC-1* | **Edit Parameters** will bring up the *Edit HEC-1 Parameters* dialog with the *Basin HEC-1 Cards* section active. The following can then be specified:

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** 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 to 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, it may be desirable 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. The project 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 * *BEXP*) 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:

$$0.2 * \frac{(1000 - 10 * CRVNBR)}{CRVNBR}.$$

•*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* .

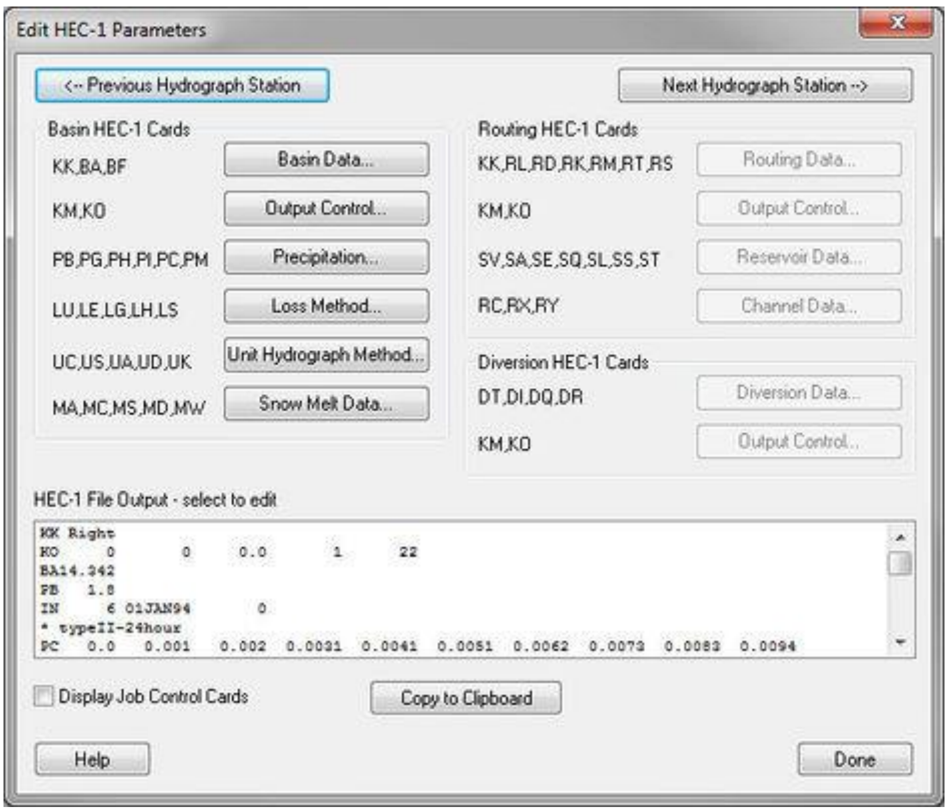
Temperature (MT)

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 L used 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 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 Δt 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 Δt 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 K coefficient 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 dividing by an assumed channel velocity (1-5 ft/s would be appropriate for most natural channels). Of course it is necessary 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 the outlet, basin, reservoir, or diversion for which to edit data and then 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 to 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, it may be desirable 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, 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) 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 choosing 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 to 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)

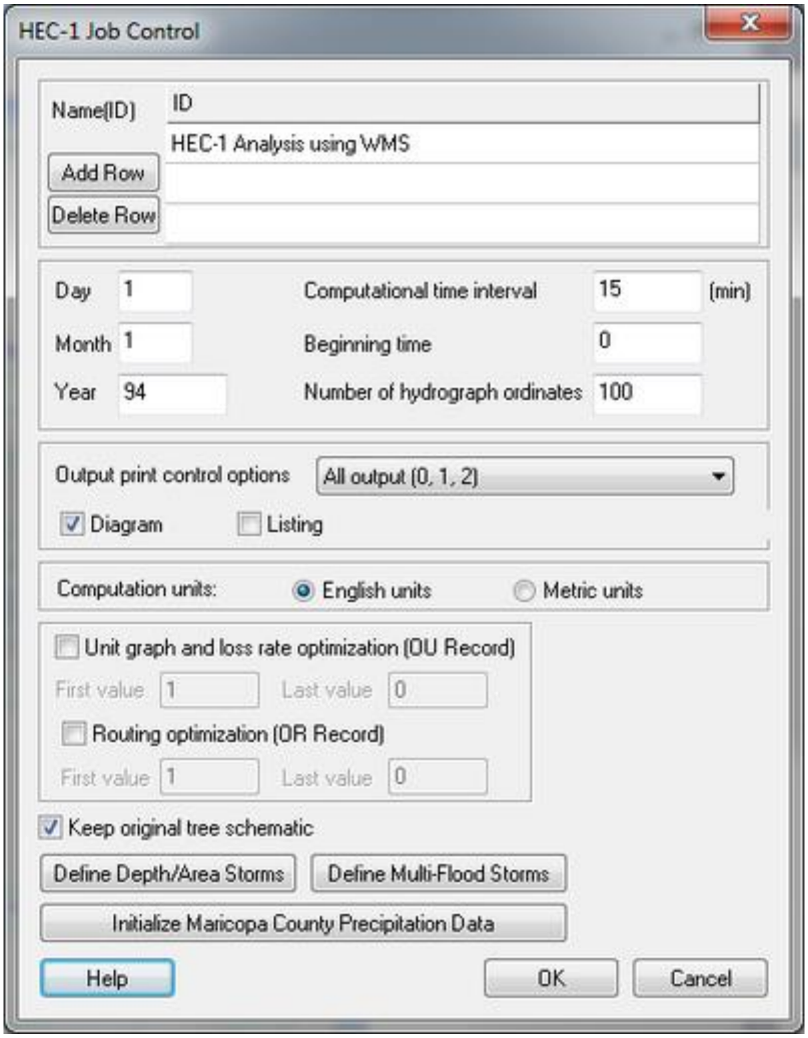
These controls determine what information about a given hydrograph station is written to the HEC-1 ASCII output file. By default the IO record information is used. However, it may be desirable to print out a more (or less) complete summary for individual hydrograph stations.

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 wanting 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 wanting to optimize routing parameters using observed inflow and outflow hydrographs and a pattern lateral inflow hydrograph for the routing reach.

- Depth Area Storms (JD)

  See below .

- MultiFlood Storms (JR)

  See below .

Depth Area Storms

Runoff simulations which use a consistent depth/area relationship as defined in the HEC-1 User's Manual [40] 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

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

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

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 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 the rain gages with the **Create Feature Point** tool. (Rain gages can also be added later on.)
8. Right-click on a rain gage with the **Select Feature Point/Node** tool and select **Attributes** (or double-click on the gage). The *Rain Gage Properties* window will appear. 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** .

Related Topics

•Rain Gage

6.5. Hydrologic Engineering Center-

Hydrologic Modeling System (HEC-HMS)

HEC-HMS

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.

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 tabs in the *HEC-HMS Job Control* dialog . These tabs 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 HEC-HMS model is included will all paid editions of WMS.

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. WMS can be used 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 there is a watershed derived in WMS from a digital terrain model then export it as part of the HMS project. In order to export the map it's necessary to convert it to feature objects. If creating the watershed from feature objects or a DEM then it will already be in this format. If creating the watershed from a TIN then it's necessary to use the **Drainage Data → Feature Objects** command found in the *Drainage* menu prior to exporting the HMS files.

Run Simulation

WMS 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. To run a simulation, first save the HMS file from WMS. Then, start HMS and run the simulation in HMS using the **Computer Current Run** macro. Finally, read the solution back into WMS by opening the HMS DSS solution file from WMS.

Saving an HMS File

When naming files, use only alphanumeric characters . Special characters such as underscore, hyphen, and characters with diacritics can not be used in the file name because HEC-HMS can not process files correctly with those in the name of the file. See the HEC-HMS website for more details.

To save an HMS file, select the *HEC-HMS* | **Save HMS File** command from the menu. Enter a file name ending in ".hms". The HMS project file and all the files (*.basin, *.control, *.map, *.met, etc.) will be saved based on the data that has been entered in WMS.

External Links

HEC-HMS Documentation

Related Topics:

- Post-Processing HMS Results
- HMS Properties Dialog
- Job Control
- Saving HEC-1 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

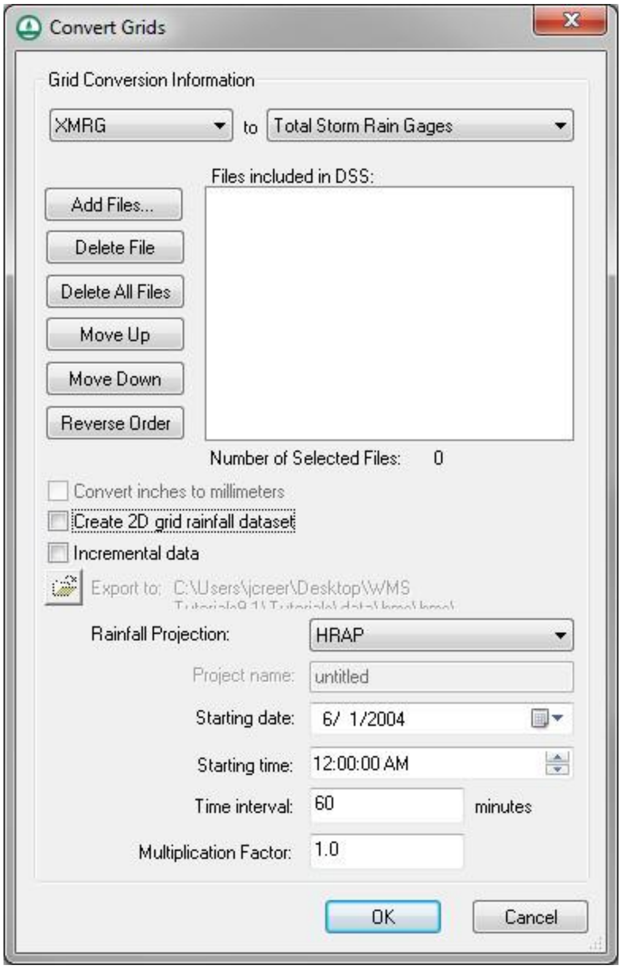
•Sub-Basins

•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 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** – 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** – First converts XMRG grids to Arc/Info ASCII grid format 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** – 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** – 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

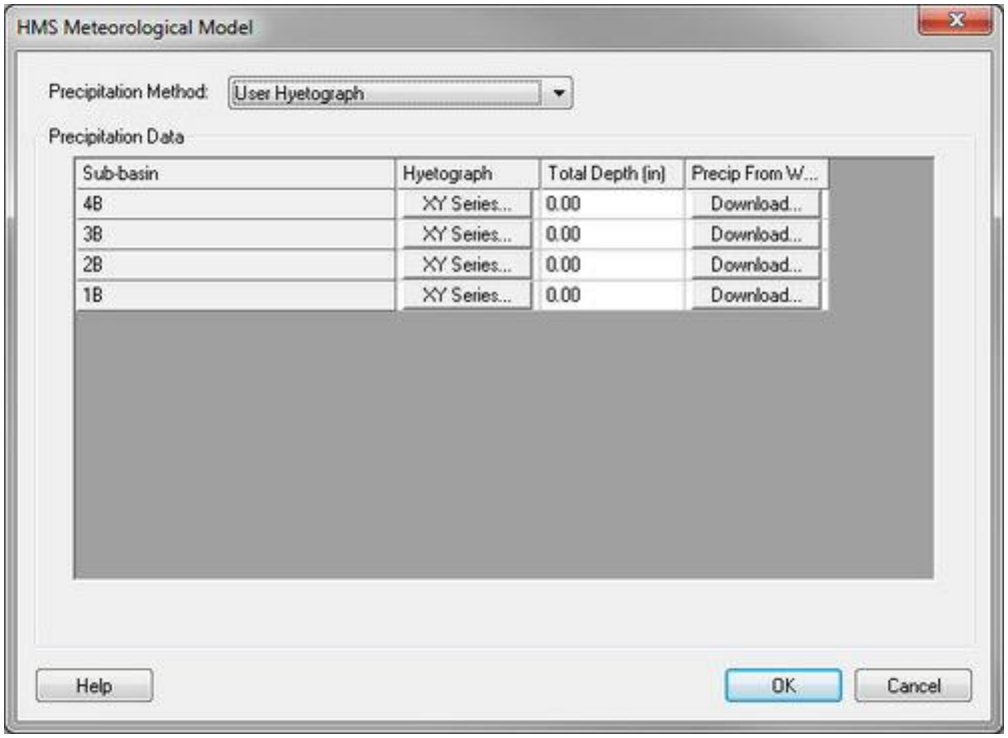
•Gages

•Rain Gage Coverage

HEC-HMS Defining the Meteorological Model

Define meteorologic data for the 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.



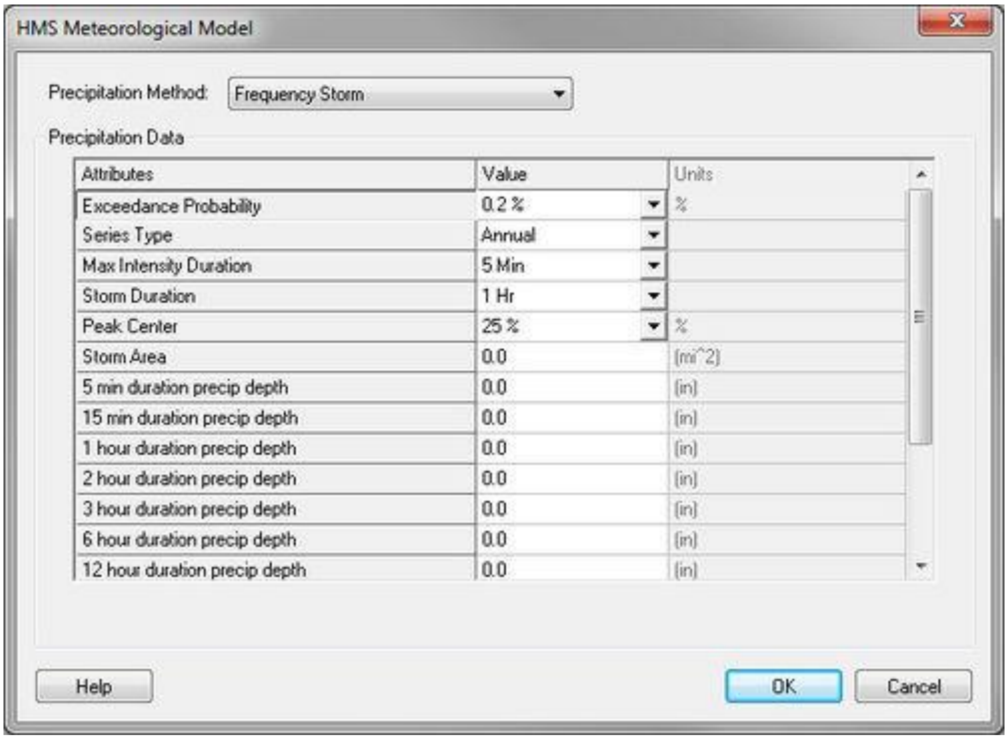
The same precipitation method must be assigned to all the basins in the HMS model. The precipitation method choosen should be based on the type of analysis to be run. If having rainfall gage data for the watershed, use the user hyetograph or the user gage weighting method. If wanting to create a synthetic storm with a known exceedance probability, select the frequency storm method. If wanting to model a standard project storm, select the standard project storm or the SCS hypothetical storm method. Only use the *No Precipitation* option if the 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 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.

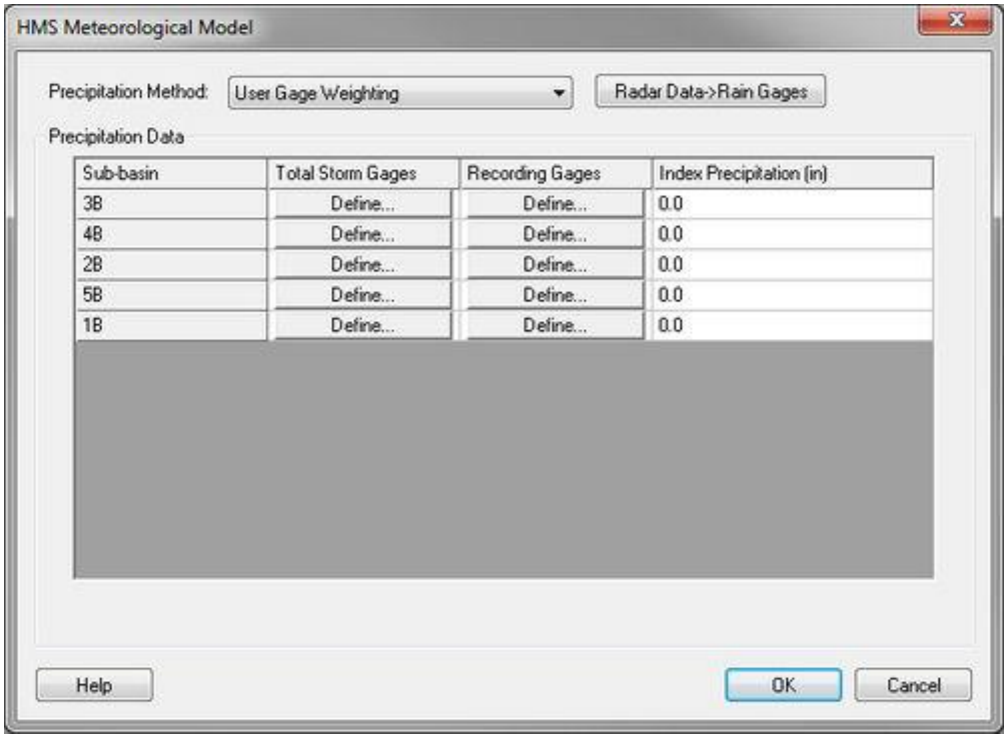
Related Topics:

•Defining the Meteorological Model

HEC-HMS Gages

The User Gage Weighting method allows defining one or more gages for each sub-basin in the watershed model.

The method of defining gages in HMS is similar to the method of defining gages in HEC-1. For each basin, 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:

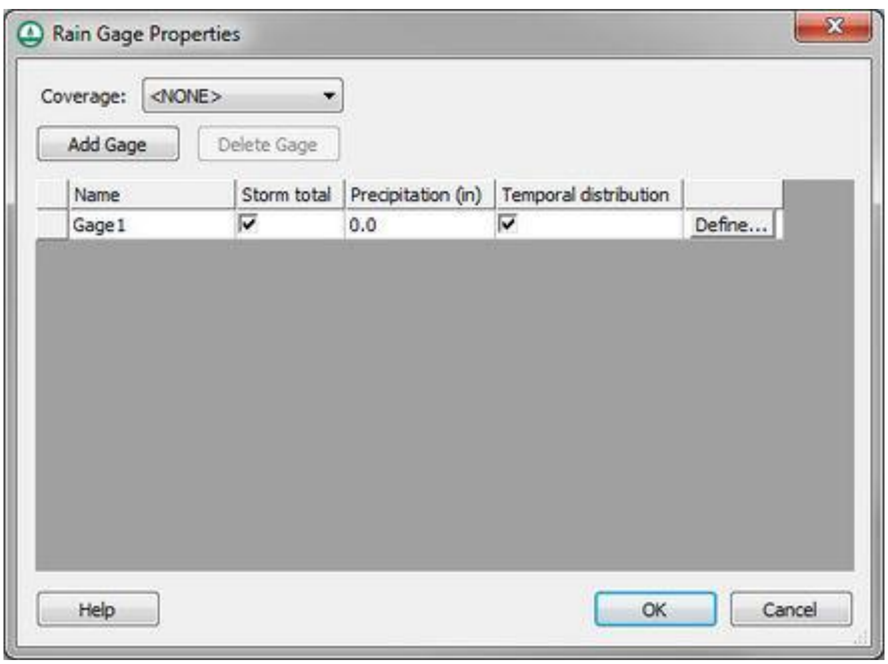


If there is a Map or DEM-based watershed associated with a coordinate system, import NEXRAD RADAR grids in XMRG format and WMS will convert the radar data to gages in a rain gage coverage in the current coordinate system . The RADAR grids are read by selecting the **Radar Data→Rain Gages** button and defining the grids and the time step associated with the grids.

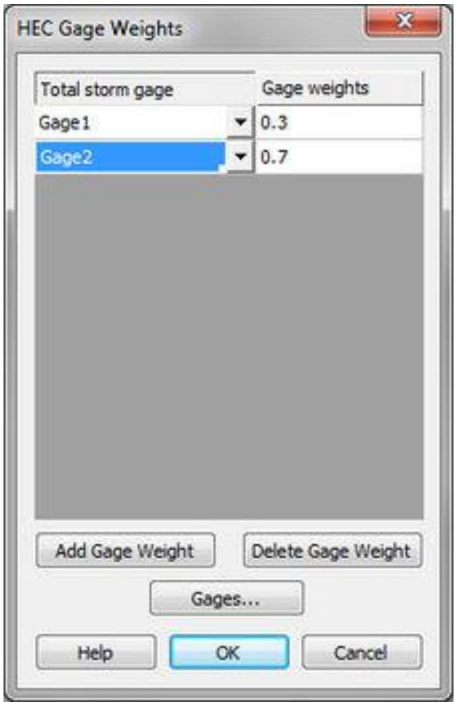
Here is how to define meteorological data for each sub-basin using the user gage weighting method:

1. Select the **Define** button under the total storm gages column. The following *HMS Gage Weights* dialog will appear:

2. Define any rain gage data that exists by selecting the **Gages** button. The *Rain Gage Properties* 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 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, define only a single recording gage, though it is possible to 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

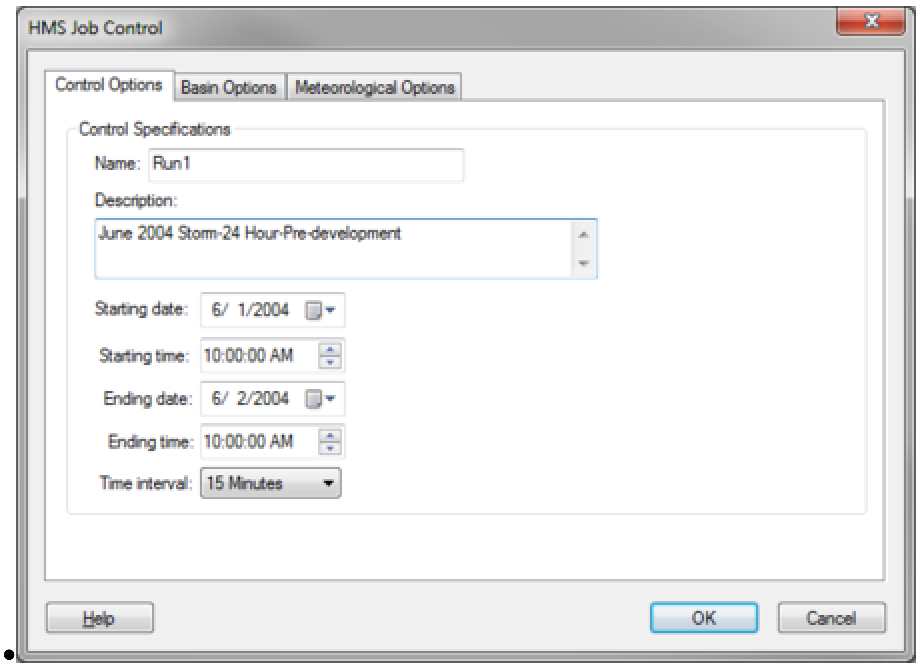
HEC-HMS Job Control

The *HMS Job Control* dialog is where parameters for the HMS model run can be set. All the HMS data is entered using one of three tabs in the *HMS Job Control* dialog . These tabs 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 dialog is reached through the *HEC-HMS* | **Job Control** menu command.

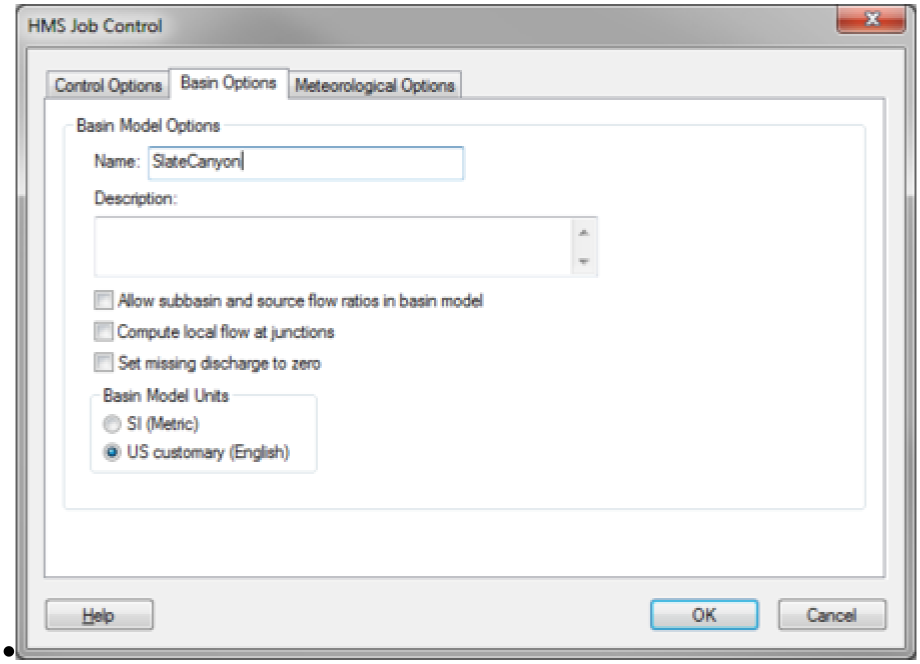
Control Options

The control option data entered in WMS is exported to the HMS project and used to set the appropriate HMS parameters when reading 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:

• 

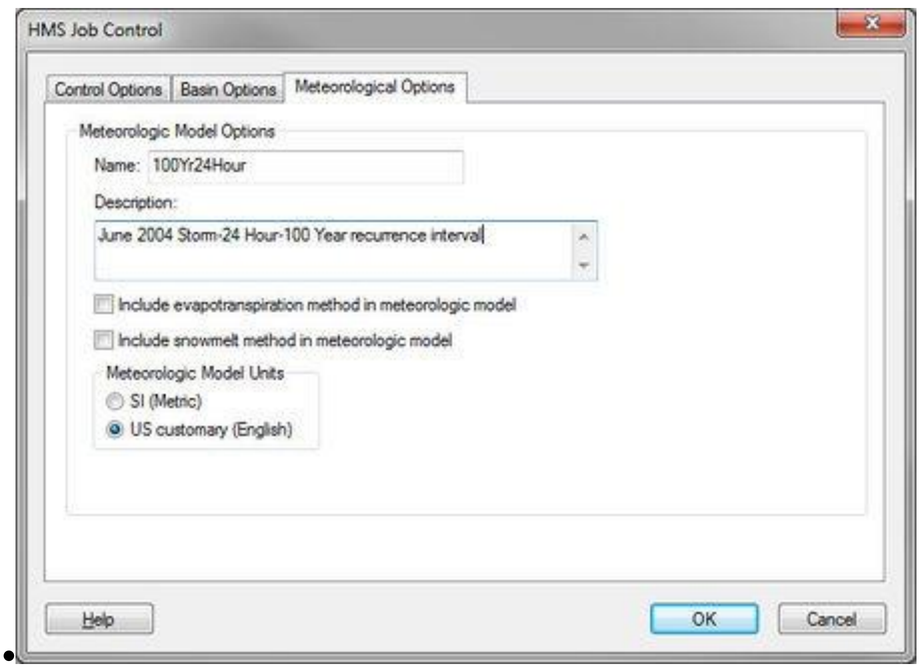
Basin Options

The basin option data entered in WMS is exported to the HMS project and used to set the appropriate HMS parameters when reading 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 reading 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:

- 

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 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, 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

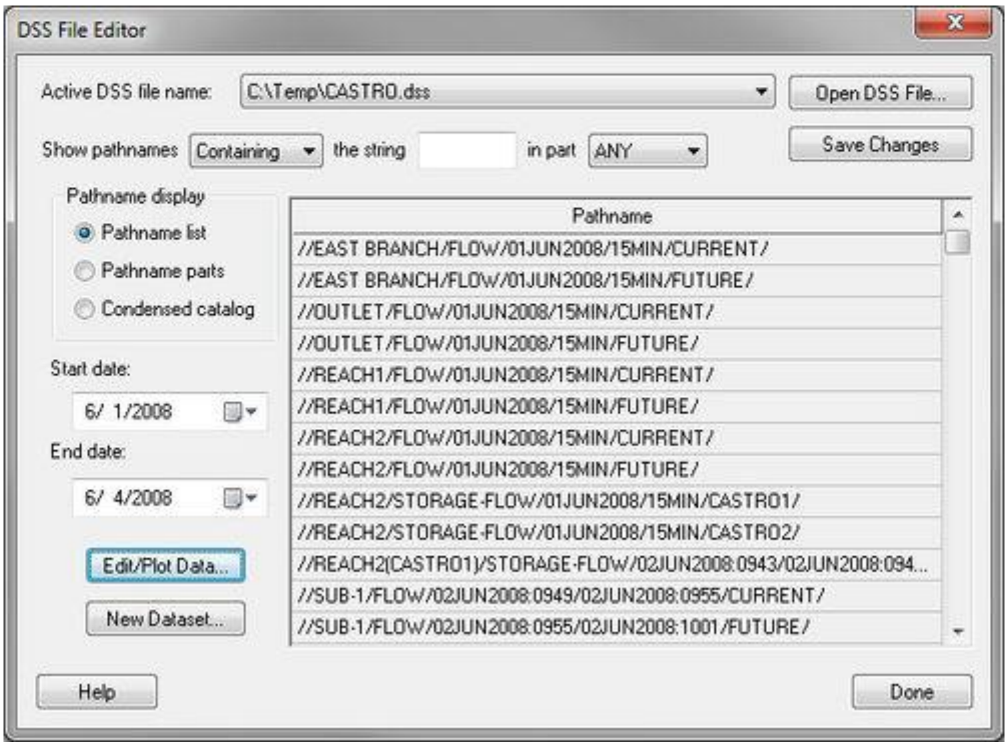
- Editing HMS Properties
- Sub-Basins
- Base Flow

•Transform

HEC-HMS Post-Processing

To run an HMS simulation, a use first needs to save the HMS file from WMS. Then start HMS, read the HMS file, and run the simulation in HMS. Finally, 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", 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 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 of interest and select the **Edit/Plot Data...** button. This will bring the data into the XY series editor where a plot of the output results can be seen as shown below.

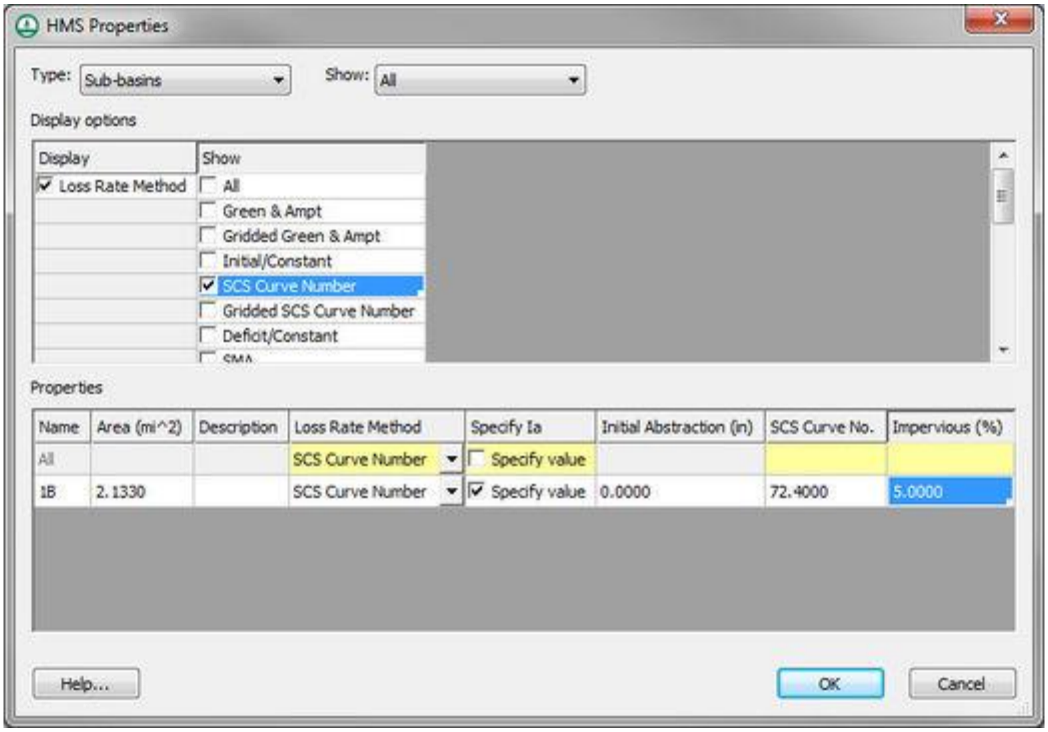
Related Topics

•[HMS Overview]

•[Running HMS]

HEC-HMS Properties

For HMS, all of the parameters (properties) are edited from the same *HMS Properties* dialog shown below.



Displaying and showing options allow seeing only those variables for which to enter data. Toggling on *Loss Rate Method* in the *Display* column allows picking which method they want to use. Then toggle the display for the different parameters associated with a given methodology from the *Show* column.

The *HMS Properties* window is versatile in that it shows properties for all or selected basins, junctions, reaches, reservoirs, etc. 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 to view or edit.

•*Show* – Choose to show or 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.

HMS includes the following hydrologic units which can be used to define a model:

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 to be defined. 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.

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 creating an outlet, WMS is actually creating a junction. WMS is also creating a reach when creating an outlet. The only data associated with junctions is a description. This description is exported to the HMS file when saving the file. A reach is also associated with each outlet.

Reaches

Reach routing will lag and attenuate the hydrograph computed (combined) at an outlet according to the reach properties defined. To define reach parameters, open the *HMS Properties* dialog, shown below, and then select the *HMS Property type* to be *Reaches* . Show or edit properties for only the selected reach(es), or for all reaches in the model. Select a reach in WMS by selecting the upstream outlet of the reach since it is from this point hydrographs are routed.

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 (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). 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. 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 a project 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.

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 Diversions** and **Retrieve Diversion** commands found in the *Tree* menu. Once created, data for the diversion can be defined and/or edited by using the *HEC-HMS Properties* 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).

•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 .

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

Also establish the initial conditions (whether there is storage), and the number of sub-reaches.

Sources

If wanting to run a model that is interior to a larger watershed then define a source at the headwaters of a stream within 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 defining a source in the *HMS properties* dialog, first add the source to the outlet. This is done using the *Add* | **Source** command from the *Tree* menu.

Sinks

If wanting to run a model that is interior to a larger watershed then define a sink at the outlet of the 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 defining a sink the *HMS properties* dialog, first add the sink to the outlet. This is done using the *Add* | **Sink** command from the *Tree* menu.

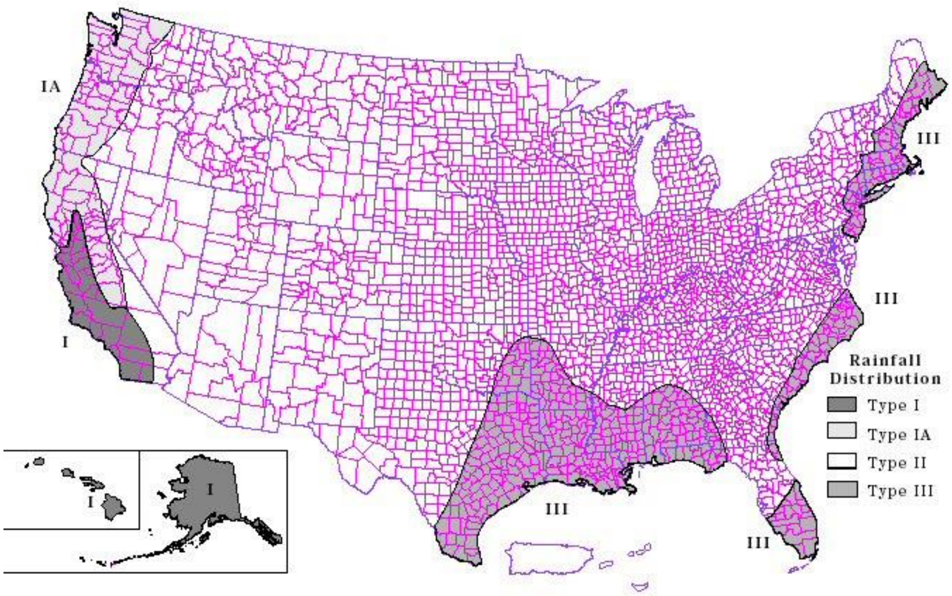
Related Topics

•HEC-HMS

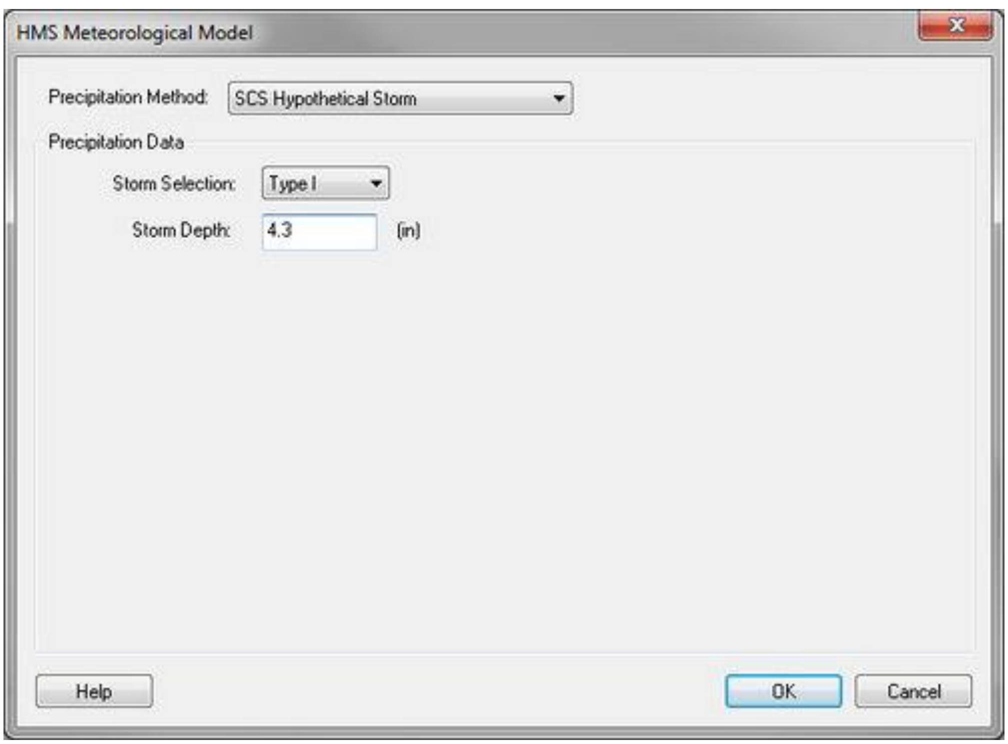
HEC-HMS SCS Hypothetical Storm

"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:



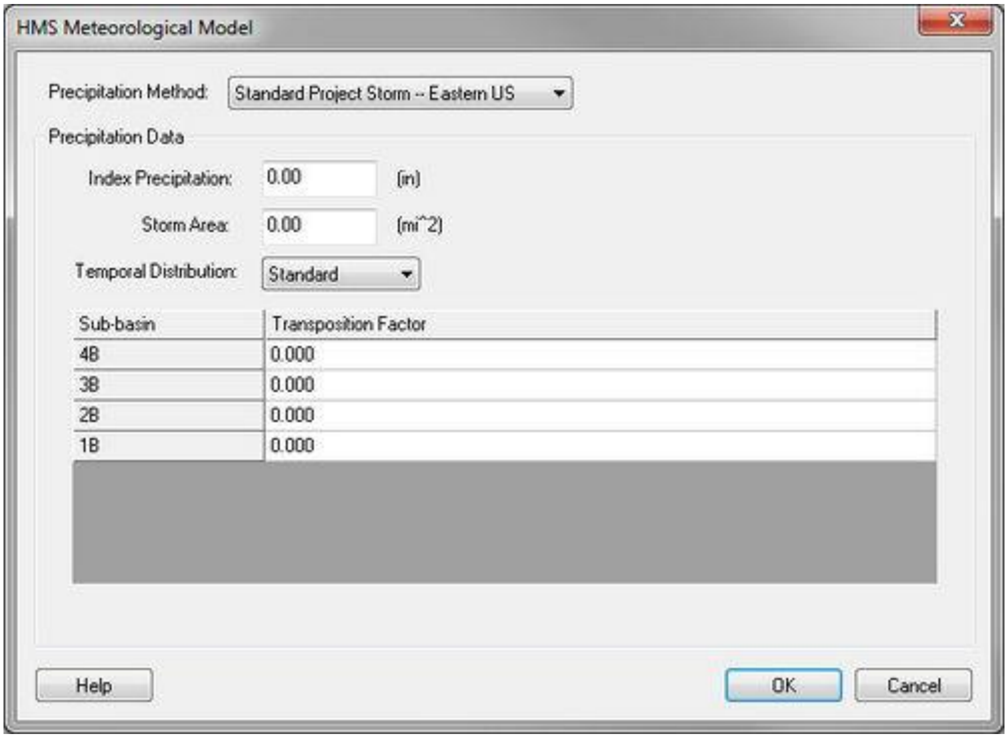
Related Topics:

- [Defining the Meteorological Model]

HEC-HMS 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)

The *HMS Meteorological Model* dialog for the Standard Project Storm method is shown below:



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

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 it is possible to use the HEC-1 model if wanting to derive a time area curve that is more specific to the 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 it is possible to use the HEC-1 model if wanting to derive a time area curve that is more specific to the 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 entering 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

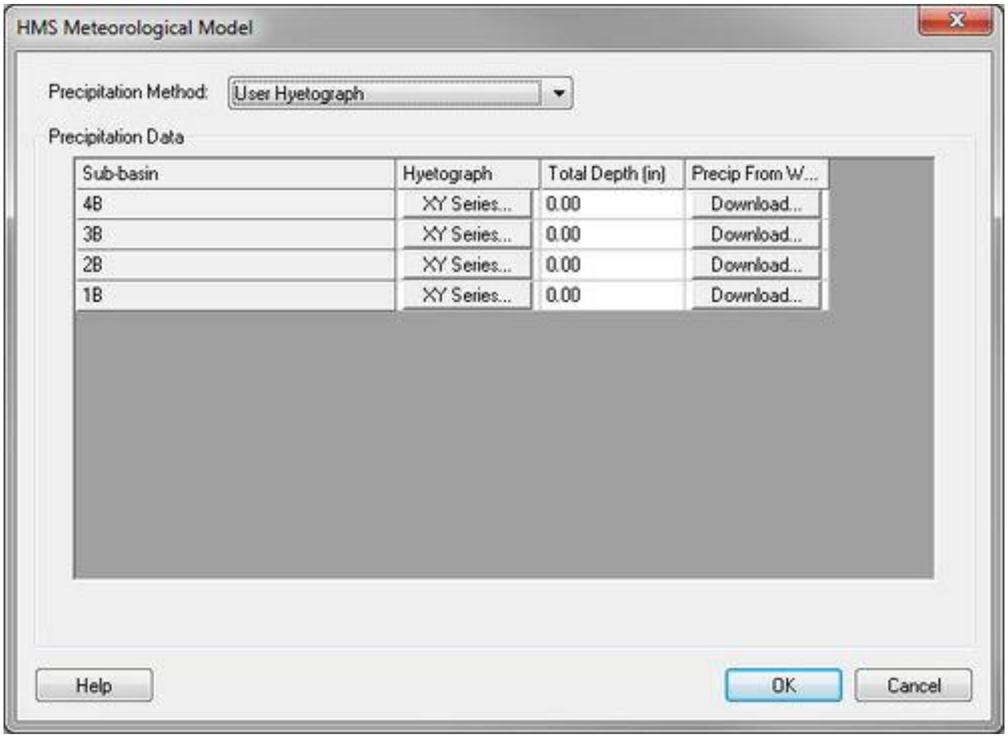
The user specified unit hydrograph option allows defining the exact unit hydrograph for the basin in the time series editor.

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 being modeled. Get this hyetograph from recorded data or from any other method. 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

6.6. 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 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 the *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. Read more about the stochastic simulation in the Stochastic Modeling article.

HEC-RAS Load Project

While the WMS interface does not allow developing a complete HEC-RAS model, it is possible to launch the HEC-RAS compute engine, or the GUI, by having a project handle in WMS. The **Load Project** command allows defining the current working HEC-RAS project and will prompt for the filename of an existing HEC-RAS project.

HEC-RAS Run Simulation

The **Run Simulation** command allows starting 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 sending 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

•Import and Export GIS Files

•Managing Cross Sections

•Bridges/Culverts

•Area Property Coverage

•Extracting Cross Sections

•Recompute All Stations

HEC-RAS Bridges/Culverts

Either add bridges directly to a HEC-RAS model or setup a bridge as a cross section in WMS. In either case, when defining a bridge or a culvert, it is recommended to add twp cross sections upstream of the bridge or culvert and two cross sections downstream in WMS. Also define a fifth cross section in WMS that represents a bridge itself or culvert itself, and it's a good idea to do this. 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, select the option in the cross section attributes dialog that makes it a bridge/culvert.

The floodplain delineation will work whether entering the culverts or bridges in HEC-RAS or as cross sections in WMS, but it is necessarry to define two upstream and two downstream cross sections in WMS for WMS to read the water surface elevations at these locations from HEC-RAS.

Trouble reading the water surface elevations back into WMS will occur if having added 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 making this specification. If no definition has been provided when exporting the GIS file, a prompt will ask 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. 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 having exported the 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 the steady state simulation is set up, 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. 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 the 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, it's necessary 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. Use this command if any changes have been made to centerline arcs in the 1D hydraulic centerline coverage that change the cross section stations in the 1D hydraulic cross section coverage. For example, if there are 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 the 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

•Import and Export GIS Files

•Managing Cross Sections

•Area Property Coverage

•Extracting Cross Sections

6.7. Hydrological Simulation Program—FORTRAN (HSPF)

HSPF

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 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.

The HSPF model is included will all paid editions of WMS.

Using HSPF With WMS

WMS does not currently have any post-processing capabilities for the HSPF model. When WMS is installed the 4 HSPF-related programs (GenScn, WDMUtil, WinHSPF, and WinHSPFLt) are not installed by default. To install them, choose to install 'HSPF' in the 'select features to install' step of the installation process. Once WMS is installed, the HSPF programs will be placed in the C:\basins folder and can be found in the BASINS folder in the Windows Start menu.

GenScn is used for generating various types of scenarios and does some post-processing. There is 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 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 the model, but it is also possible to 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, it's necessary to close WMS after saving the 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.

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, it's possible to 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 to consult the HSPF documentation to receive a complete description of how HSPF uses the parameters.

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 specifying 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, select **OK** , and 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, 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 having viewed or edited the WDM file data, it's necessary to close WMS after saving the 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.

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.

HSPF Saving and Reading HSPF UCI Files

While working on the definition of HSPF parameter, or when finished, 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, 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

•Automated Land Segmentation

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
- 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 selecting the **Done** button. This enables switching 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 specifying whether the segment is pervious or impervious. The **Define Activities** button brings up dialog which allows activating modules (i.e. PWAT, PQUAL, IWAT, etc.) for the selected segment and define the required parameters. The **External Targets** button allows defining 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 defining 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 [41]). The *Assign External Sources* dialog in WMS has been developed to allow setting 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, 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

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, 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 setting 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. [42]

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.

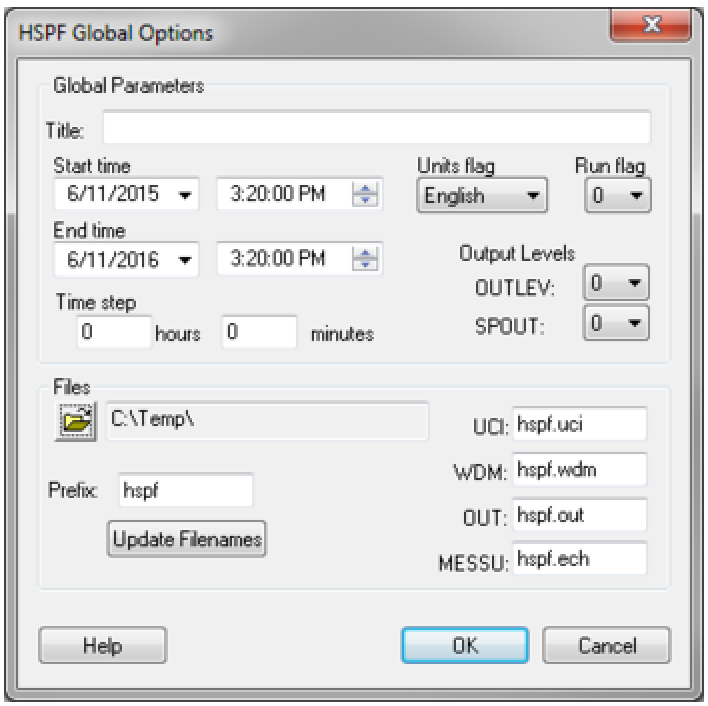
This dialog also contains an **Apply targets to segments** button. This button works that same as the **Apply parameters to segments** button explained above; the targets defined will be copied to the selected segments.

Related Topics

•WDM File Interaction

•External Sources

HSPF 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 it can be accessed. Many features within the HSPF interface will be dimmed and remain dimmed until the HSPF data is initialized. By clicking on the **Initialize HSPF Data** button within the *Global Options* dialog box, all HSPF data within the watershed will be initialized. The **Delete HSPF Data** button will remove all HSPF data from memory. This may be useful if wanting to start over with HSPF input or wanting 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 [43] 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 Filenames** button.

Related Topics

- Overview of HSPF
- 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 [44] . 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 adding or deleting 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 specifying a file name and then the mass links are saved as an UCI-formatted ASCII text file. The **Import** button allows reading 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 there is 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, there is 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 [45]). 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 selecting 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 using 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 allows entering 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. [46]

SNOW

To access the *SNOW Parameters* dialog box, press the **SNOW** button in the *Pervious Land Activity* dialog box. The *SNOW Parameters* dialog box allows entering 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. [47]

IWATER

To access the *IWATER Parameters* dialog box, press the **IWATER** button in the *Impervious Land Activity* dialog box. The *IWATER Parameters* dialog box allows entering 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. [48]

SOLIDS

To access the *SOLIDS Parameters* dialog box, press the **SOLIDS** button in the *Impervious Land Activity* dialog box. The *SOLIDS Parameters* dialog box allows entering 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. [49]

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 specifying up to 10 water quality constituents or pollutants in the outflows from an impervious land segment. The *IQUAL Parameters* dialog box allows entering 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. [50]

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 allows entering 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. [51]

SNOW

To access the *SNOW Parameters* dialog box, press the **SNOW** button in the *Pervious Land Activity* dialog box. The *SNOW Parameters* dialog box allows entering 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. [52]

PWATER

To access the *PWATER Parameters* dialog box, press the **PWATER** button in the *Pervious Land Activity* dialog box. The *PWATER Parameters* dialog box allows entering 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. [53]

SEDMNT

To access the *SEDMNT Parameters* dialog box, press the **SED** button in the *Pervious Land Activity* dialog box. The *SEDMNT Parameters* dialog box allows entering 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. [54]

PQUAL

To access the PQUAL Parameters dialog box, press the **PQUAL** button in the *Pervious Land Activity* dialog box. This dialog box allows specifying up to ten water quality constituents or pollutants in the outflows from a pervious land segment. The *PQUAL Parameters* dialog box allows entering 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. [55]

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 allows entering 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. [56]

PEST

To access the *PEST Parameters* dialog box, press the **PEST** button in the *Pervious Land Activity* dialog box. This dialog box allows specifying up to three pesticides on a pervious land segment. The PEST Parameters dialog box allows entering 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. [57]

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 allows entering 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. [58]

PHOS

To access the *PHOS Parameters* dialog box, press the **PHOS** button in the *Pervious Land Activity* dialog box. The *PHOS Parameters* dialog box allows entering 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. [59]

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 allows entering 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. [60]

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 allows entering 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. [61]

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 allows entering 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. [62]

CONS

To access the *CONS Parameters* dialog box, press the **CONS** button in the *Reach/Reservoir Activity* dialog box. This dialog box allows specifying up to 10 water quality constituents or pollutants in the flows from a reach/reservoir segment. The CONS Parameters dialog box lets entering 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. [63]

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 entering 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. [64]

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 entering 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. [65]

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 entering 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. [66]

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 entering 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. [67]

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 entering 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. [68]

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 entering 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. [69]

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 entering 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. [70]

Related Topics

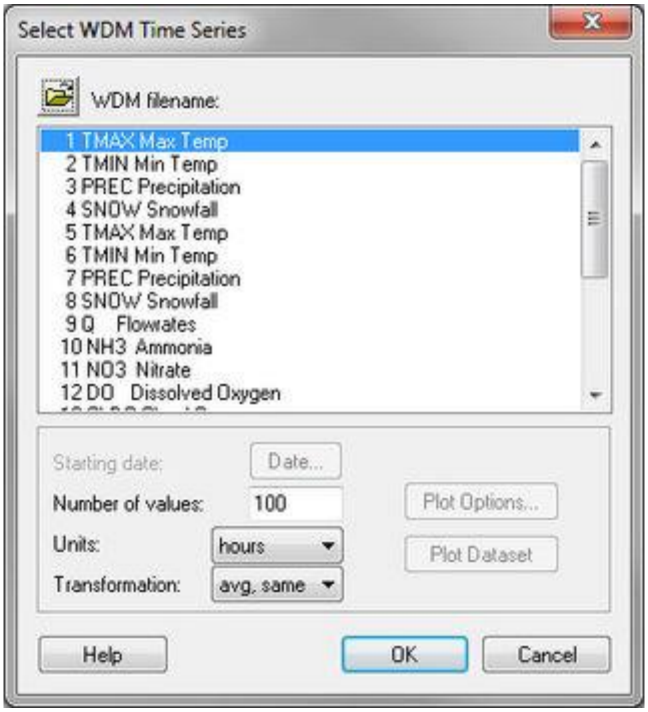
•HSPF Edit Parameters

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.

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. 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

•Overview of HSPF

•External Sources

•External Targets

6.8. MODified RATional Method (MODRAT)

MODRAT

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.

The MODRAT model is included will all paid editions of WMS.

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.

Run MODRAT

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 saving the MODRAT input and output files and enter a new file name 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 changing 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 to be inputted 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 having selected **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, view the ASCII output file using the **View File** command in the *File* menu.

Saving and Reading MODRAT Files

While working on the definition of MODRAT parameters, or when finished, save the data. There are two 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 when 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, 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

- MODRAT WMS Interface
- Editing MODRAT Parameters

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.

MODRAT Menu

When the MODRAT model is active in the hydrologic module, the *MODRAT* menu can be used. The menu has the following commands:

- **Open MODRAT File** – Launches a file browser to open MODRAT files (*.lac).
- **Save MODRAT File** – Saves all MODRAT input data to standard MODRAT input files.
- **Check Simulation** – Brings up the *MODRAT Model Check* dialog which will display any errors found in the simulation.
- **Run Simulation** – Launches the *MODRAT Run Options* dialog for starting the model run.
- **Read Solution** – Launches a file browser to load in MODRAT solution files.
- **Job Control** – Opens the *MODRAT Job Control* dialog where model run parameters are set.
- **Numbering Tree** – Brings up the *MODRAT Renumber* dialog to initiate the numbering process.
- **Edit Parameters** – Brings up the *MODRAT Parameters* dialog where attributes or parameters for all MODRAT hydrograph stations are defined or edited.
- **Compute Tc** – Brings up the *Compute MODRAT Time of Concentration Wizard* . The first step of the wizard will check the input requirements. The second step will show the time of concentration results.
- **Map Attributes** – Brings up the *Map MODRAT Attributes* dialog where a digital map parameters can be assigned to a watershed model.
- **Insert Tree Spacing** – Updates all location numbers by multiplying them by the scaling factor in order to create node spacing.
- **Debris Production/Bulking** – Brings up the *Debris Production* dialog.
- **Create Burned Simulation** – Brings up the *Generate Burned Simulation* dialog to create burned simulations.

Related Topics

- Overview of MODRAT
- Editing MODRAT Parameters

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 the simulation. Be sure to save the *.lac file with a different name before creating a burned simulation if wanting to keep the normal soil simulation.

Related Topics

- Editing MODRAT Parameters

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 is a dynamic dialog that will change depending on what is selected when the dialog is brought up. This dialog is accessed by selecting the **Edit 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 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 *MODRAT Parameters* dialog appears, it will remain open until selecting the **Done** button. This enables switching between basins, reaches, or relief drains quickly and define the necessary parameters without closing and opening the dialog for each hydrograph unit.

General Features

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 a 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 a watershed model, see the MODRAT Numbering the Tree section.

Basin Parameters

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, 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 launch the *Time Computation* dialog where Tc can be computed 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 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 launches the *XY Series Editor* in WMS. This editor allows importing or manually entering 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 the WMS installation.

Outlet and Diversion Parameters

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, calculate and enter the length of each reach manually.

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.

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.

Routing/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

•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.

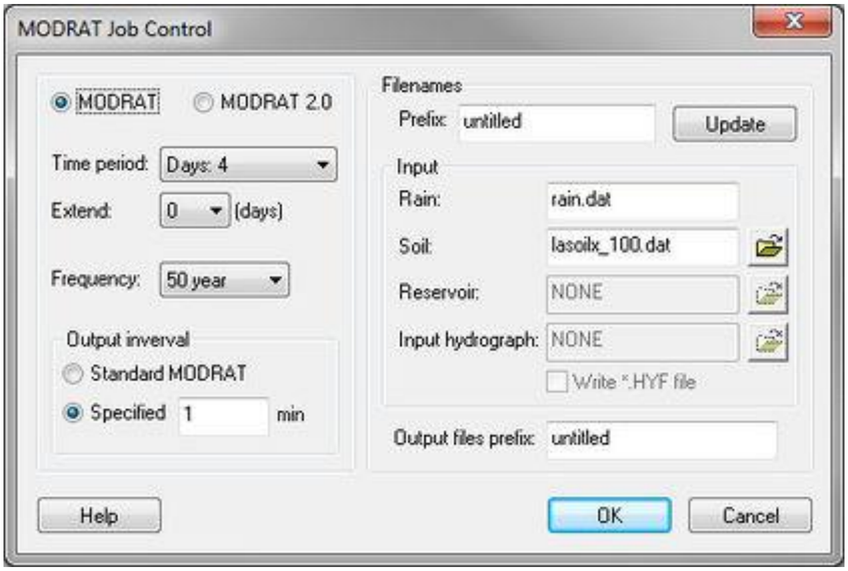
Related Topics

•Editing MODRAT Parameters

•Job Control Parameters

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 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—name it anything 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 having defined the reservoir with the *Detention Basin Calculator* in WMS, this file will be created by WMS – name it anything appropriate. If having manually created this file in the format required by MODRAT, enter the name of the file here.

Input Hydrograph File

If an input hydrograph has been specified at any location in the 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 was 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 for all output files that MODRAT will create when it runs (*.out, *.sum, *.out, etc.). This may be any name deemed 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

•Editing MODRAT Parameters

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, a prompt will ask 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. Select the basins corresponding to each lateral as 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, it is possible to 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

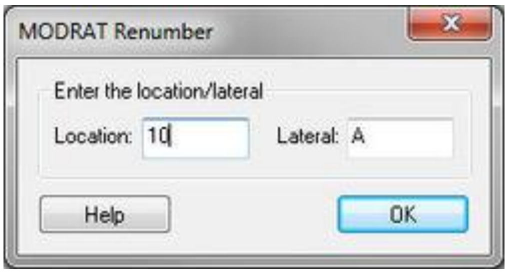
•It is not necessary to select a sub-basin for all laterals, just select **Done** without selecting any basins

If the renumbering results are not what was wanted, choose the **Number Tree** command again and try something different. If it is not possible to achieve the numbering/lettering with the automatic process, 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 a new basin, reach, or diversion is added. 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 adding a basin at a confluence point, the default location number will be updated based on the lateral entered.

Deleting Tree Items

The tree is renumbered when a basin or diversion is deleted. If a reach is deleted, it will be necessary to number the entire tree again if wanting to update the location numbers and laterals for each tree node.

Renumbering automatically occurs when editing 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 in the *MODRAT Parameters* dialog. 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 inputing 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 user entered peak flow, with residual flow transferred to the relief drain.

•Specified percentage – This option requires inputing 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 specifying 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 specifying 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

•Reservoirs

MODRAT Reservoirs

Checking the *Reservoir* routing box in the *MODRAT Parameters* dialog 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

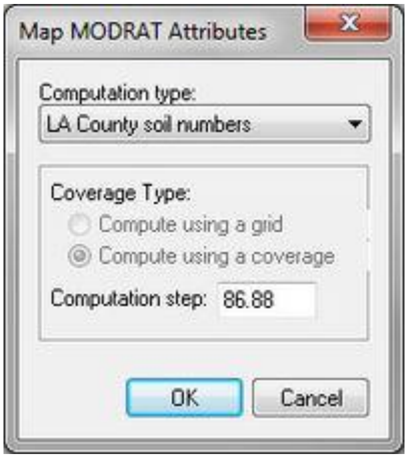
•Editing MODRAT Parameters

•Relief Drains

•Detention Basin Calculator

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 the 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 – Determines the predominate soil type in each sub-basin of the watershed.

To compute the soil number for each sub-basin of the MODRAT model, 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, compute and assign soil numbers using the **Map Attributes** command from the *MODRAT* menu.

LA County land use – Determines 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 the MODRAT model, 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, 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 – Mapped using the GIS calculator.

To compute the rainfall depth on the sub-basins of the MODRAT model, 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 there is other rainfall data in vector format (ArcView® Shapefiles, DXF, etc.), import these into WMS. Make sure to 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, 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 dataset and cannot be changed.

Related Topics

- Editing MODRAT Parameters

6.9. 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.

It may be necessary 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 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 wanting to run the NSS outside of WMS it is still possible to 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.

The NSS interface is included will all paid editions of WMS.

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 to continue with a particular model at a later time.

Model Troubleshooting

If unable to get NSS to run from within WMS, try the following:

There may not be write access to the C:\Program Files directory on the 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 where there is 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 where there is write privileges and click **Open**

NSS can now be run.

Related Topics

- NSS Region Coverage
- Defining an NSS Simulation

NSS Computing Peak Discharges

Setting up the analysis consists of the following steps, all done within the *National Streamline Statistics Method* dialog. The dialog can be used to do the follow:

1. Delineate the watershed and compute the basin area and other geometric parameters. Enter the value manually if not having delineated from a DEM, drainage coverage, or TIN.
2. Select the *State* the 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.

National Streamline Statistics Method Dialog Overview

The dialog is divided into four sections:

Basin Information

•*Basin Name*

Displays the basin ID by default. A different name can be manually entered if desired.

•*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.

•*State*

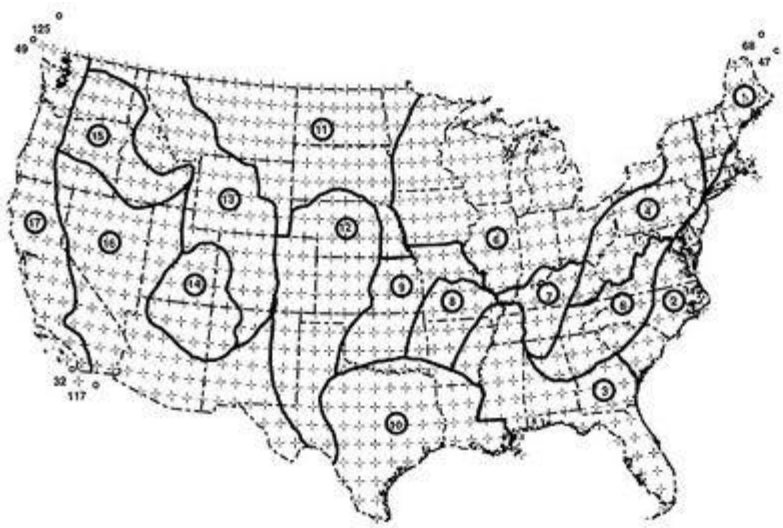
The NSS regression equations are separated by state. The *State* drop down menu is used to specify the state that the watershed is located in. Once the state is specified, the available regions will appear in the regional equations text window. Besides the 50 US states, an equation for Puerto Rico and a custom defined equation can be selected from the available choices.

•*Max 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 the watershed. A default value has been determined for each region, but this value may be overwritten.

•*Units*

Units can be displayed in either "English" or "Metric".



Regional Regression Equations

Each state is divided up into hydrologically similar regions with separate regression equations. Once the state is selected, the available regions appear in this section of the dialog.

- *Available Equations*

  Lists all available regional regression equations for the selected *State* .

- *Selected Equations*

  Lists all regional regression equations that have been selected.

- **Select→**

  Moves selected equations from the *Available Equations* list to the *Selected Equations* list.

- **←Remove**

  Deletes a selected equation from the *Selected Equations* list.

- **Compute Overlapping Areas**

  Opens a dialog that shows overlapping regions in the *Selected Equations* section. See NSS Overlapping Regions for more information.

Variable Values

As regions are selected in the *Regional regression equations* section, the relevant parameters for the regional equations are added to the *Variable values* spreadsheet. Parameters that can be computed by WMS are defaulted to their computed values at the time the regression equation is selected for use. If desired, override defaulted values 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 the 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.

The following values are typically displayed in this field.

- *Variable Name*
- *Abbreviation*
- *Value*
- *Units*
- *Minimum*
- *Maximum*

Results

- **Weighting Options**
- **Compute Results**

  Estimate the peak flows for the 2, 5, 10, 25, 50, 100 and 500-year recurrence intervals.

- *Max Flood Envelope*
- **Compute Hydrograph**

  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.

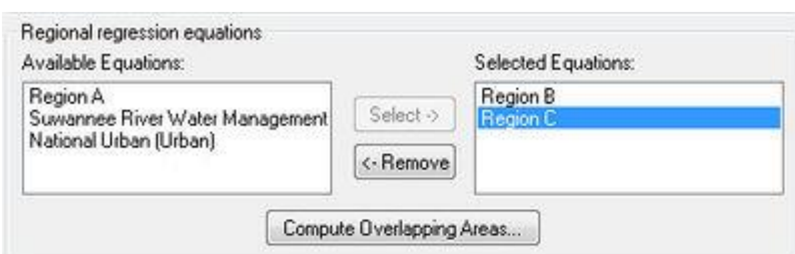
- **Export**

  Allows saving the results as a Comma-Separated Values File (*.csv).

Related Topics

- Overview of NSS
- GIS Overlay

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

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 the study watershed. It is also possible to 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 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

6.10. Orange County Hydrology

Orange 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.

Both the OC Rational and OC Hydrograph models are included will all paid editions of WMS.

Related Topics

•Orange County Rational Method

•Orange County Unit Hydrograph Method

•Creating Watershed Models

•GIS Module

6.10.a 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. 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

a) Stream/channel routing – A convex routing method is included in HEC-1

b) Flow-through detention – Defines using reservoirs or storage routing in HEC-1

c) Flow-by detention – Defines using diversions in HEC-1

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 saving a WMS project file using the **Save** command in the *File* menu.

Related Topics

•Orange County Hydrology

•Editing HEC-1 Parameters

•Job Control

- Design Storms
- Small Area Hydrographs
- Using GIS Data
- Running a HEC-1 Analysis
- 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. See the article HEC-1 Job Control for more information about 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 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 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.

1. Click the **Update Frequency** button to select the return period and IDF curves
2. Computational time interval must be equal to or greater than the time of concentration
3. Use the **Computes Losses** button to compute losses using GIS data
4. Select **Next** to view a tabular output of the calculations
5. 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 entering a value for T_c, but it is always possible to 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. Turn on the check box on next to each S-graph that is to be used. 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

6.10.b 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.

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.

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 to continue with a particular model.

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.

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

- Orange County Hydrology
- Orange County Rational Job Control
- Reach Routing
- Sub-Area Data
- Orange County Rational Equation
- 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 $(ft^3/s \text{ or } m^3/s)$
- 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:

$$Tc = K(L^3/H)^E$$

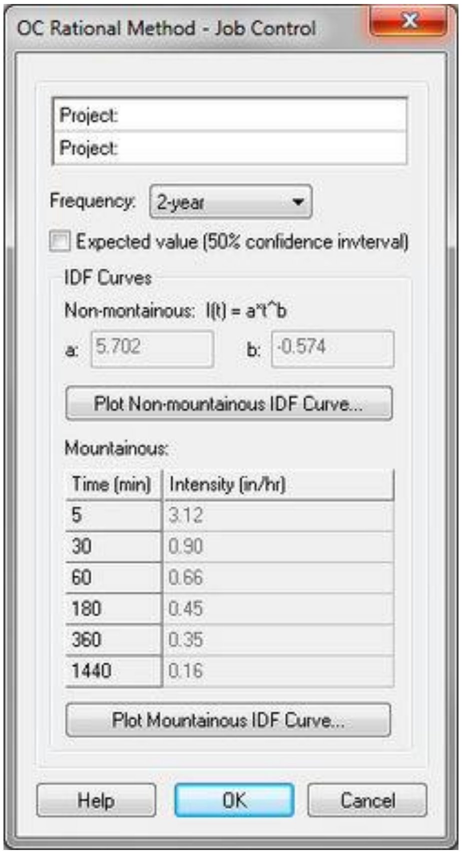
where:

- 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. 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.

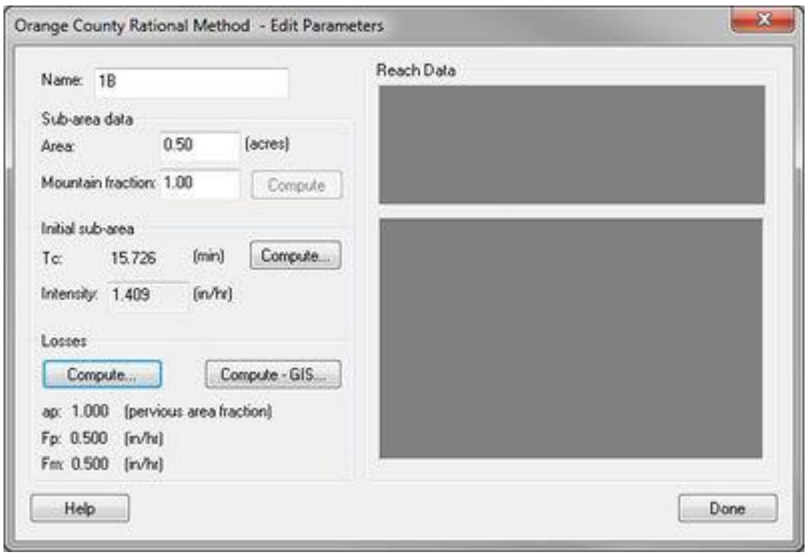
Related Topics

•Sub-area Data

•Reach Routing

•Losses

OC Rational 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. 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 if 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.

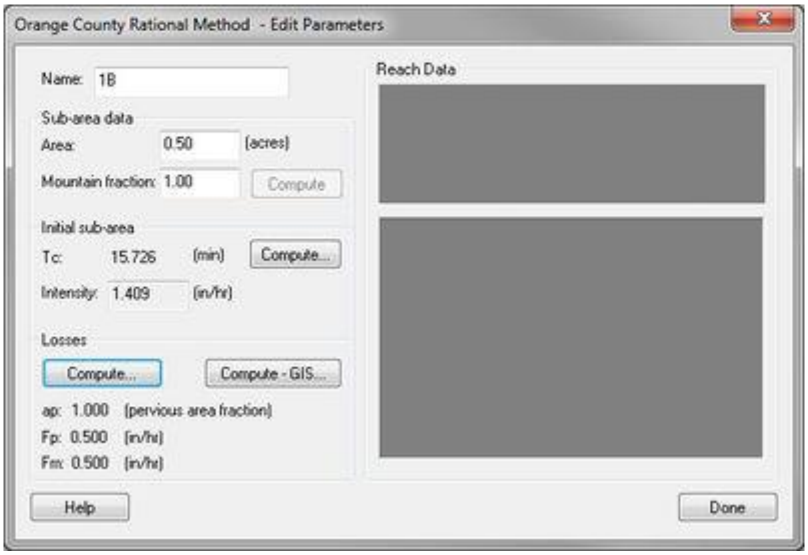
Design/Analysis Mode

There is a design and analysis mode if 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]
- [Creating Watershed Models]

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** 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

- Orange County Rational Method

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.

The Rational Method interface is included will all paid editions of 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 to continue with a particular model.

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 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 wanting to set the value of i (rainfall intensity) for all basins then 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 the edit fields are filled with the values of that basin each time a basin is selected. 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 update the value for a given parameter (alternatively, 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 computing 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 combining 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. Options include:

•*Traditional method* – Computes a peak flow at an outlet by determining composite C , t_c , and areas from upstream basins and tributaries.

•*Route by summing* – 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.

•*Hydrographaph computation method*

- Rational Hydrograph
- Modified Rational Hydrograph
- Dekalb Rational Hydrograph
- Universal Rational Hydrograph
- User Defined Rational Hydrograph

Related Topics

•Rational Method Edit Parameters

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 entering all of the necessary values for computing a peak flow for a selected catchment area, or confluence point. The values for C , i, and A represent the values of the currently selected basin or outlet. Options include:

- *Display*
  - *Type* – Options include "Basins" or "Outlets" which will determine the options in the parameters section below.
  - *Show* – Options include "Selected" or "All". The "Selected" option will on show options for the currently selected outlet or basin. The "All" option will show values for all basins and outlets in the project.
- *Units*
- *Parameters* – Section for all values that can be set. Along with the following options, there can be a column for Basin, Outlet, and Unit options depending on how the dialog was reached.
  - Name
  - Runoff Coefficient
  - Rainfall Intensity
  - Compute I – IDF Curves
  - Area
  - Time of Concentration
  - Compute Tc – Basin Data
  - Compute Tc – Map Data
  - Routing Lag Time
  - Compute Lag Time
  - Define Reservoir
  - Flowrate
  - Compute Hydrographs

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:

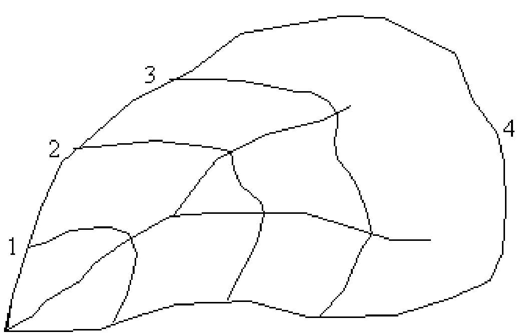
$$Q = kCiA$$

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 values 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.

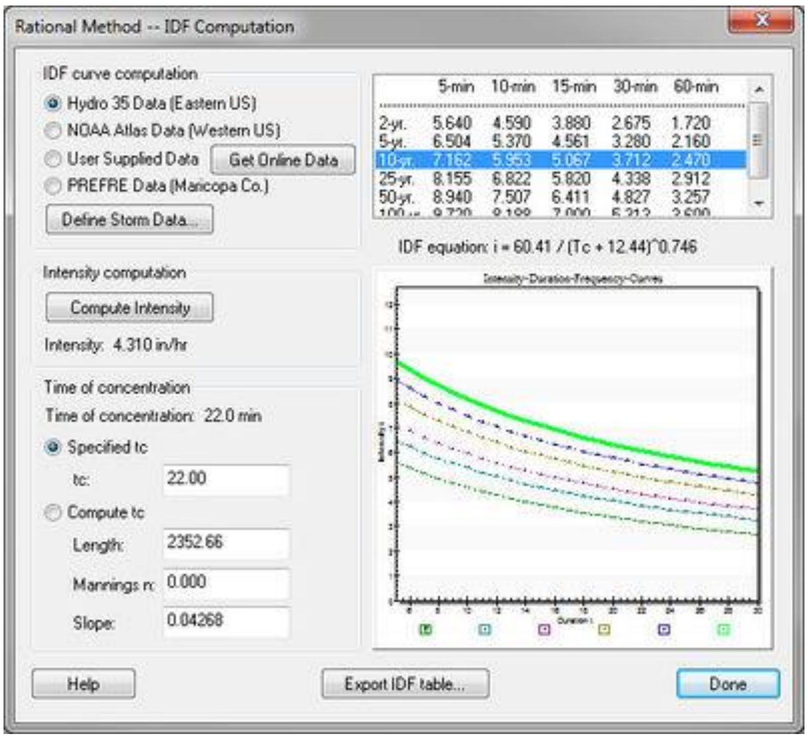
Related Topics

- 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 directly entering an i value if typically computed in another way or the design value to use is already known.

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 entering 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 and in FHWA HDS-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. Then specify the recurrence interval 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:

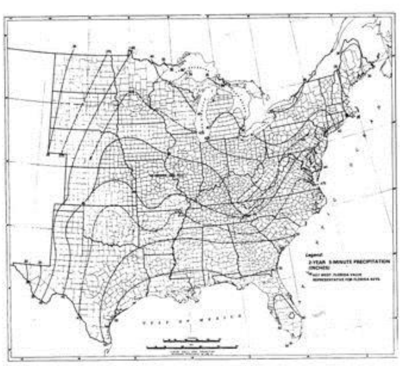
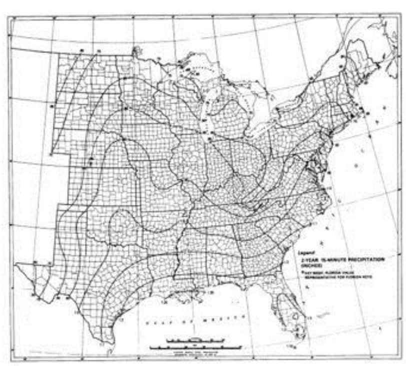
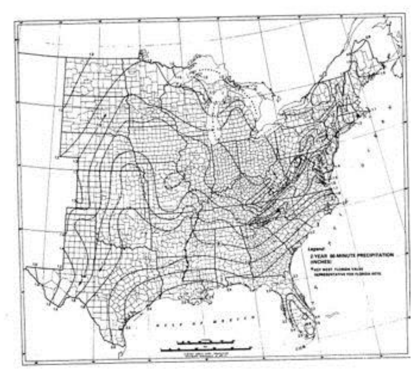
$$t_c = \frac{KL^{0.6}n^{0.6}}{i^{0.4}S^{0.3}}$$

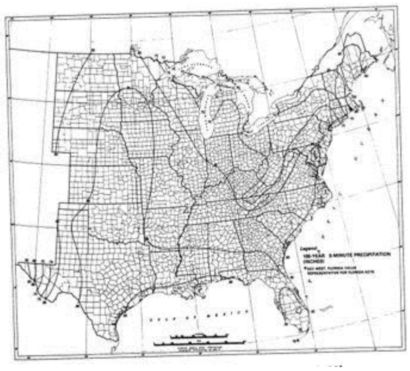
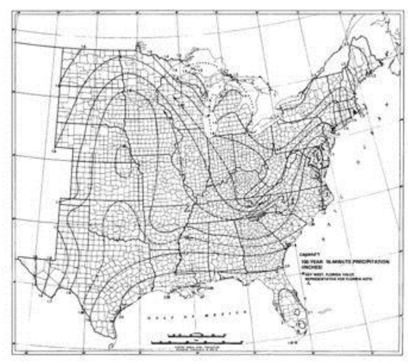
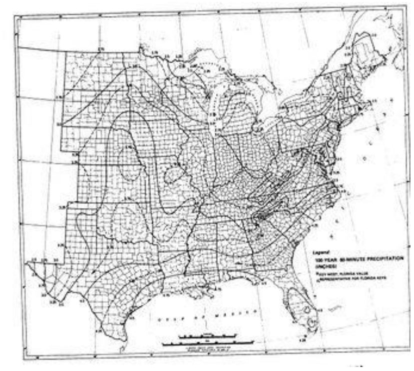
where:

- t_c = 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

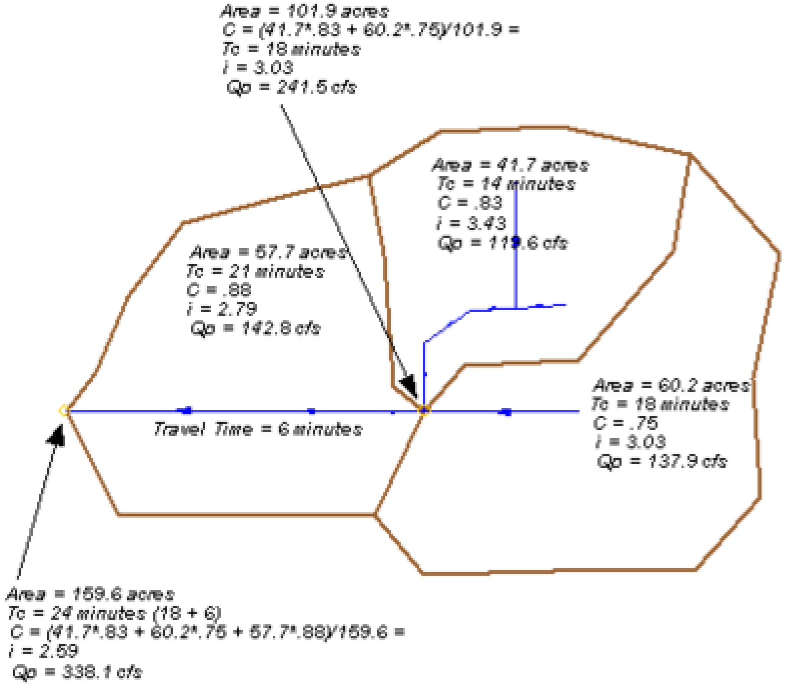
Related Topics

- Entering Rational Method Parameters
- Basin Data
- Outlet Data

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_c a 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, there is also 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.

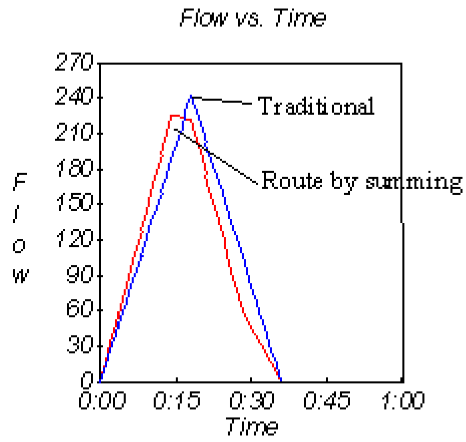


Rainfall intensities are determined using the times of concentration and an IDF curve. As would be expected the higher the time of concentration, the smaller the intensity.

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.

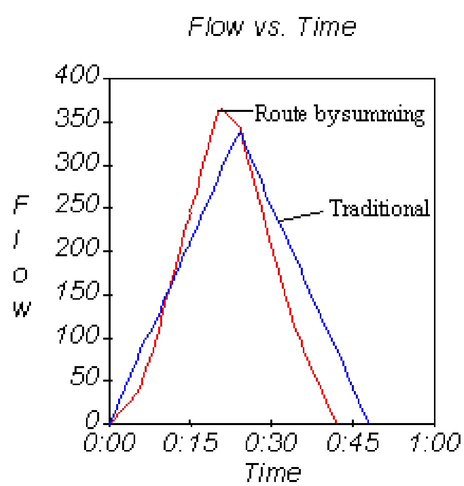
In the case of the traditional method the time of concentration for the "upstream" outlet (the location where the two upper basins join) is determined to be 18 minutes (the largest t_c of the two basins). From this t_c a rainfall intensity of 3.03 is determined from the IDF curve. The contributing area 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

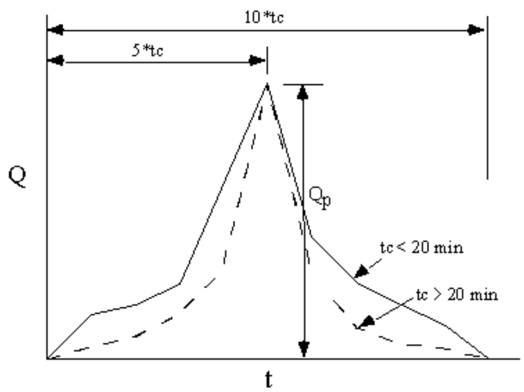
Dekalb Rational Hydrograph

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.



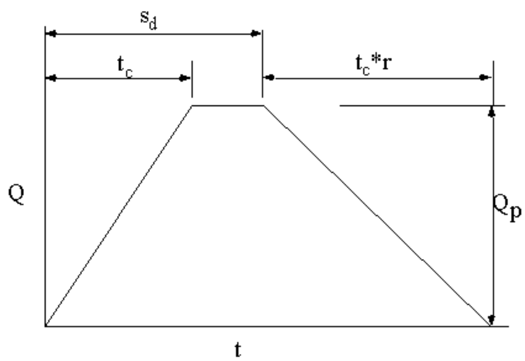
Related Topics

- Computing Hydrographs with Rational Method

•Rational Method Equation

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:

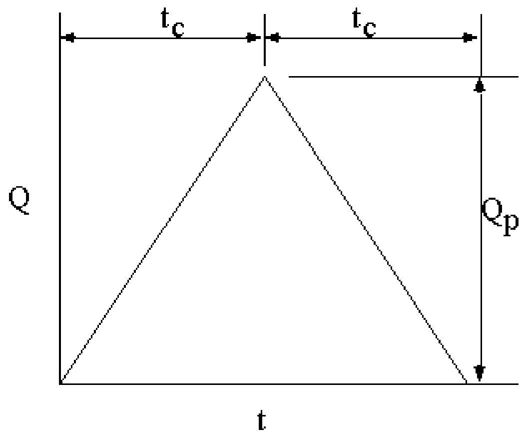
- t_c = Time of concentration.
- Q = Flow at time t , in cfs.
- Q_p = Peak flow.
- r = Falling limb coefficient
- s_d = Storm duration

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

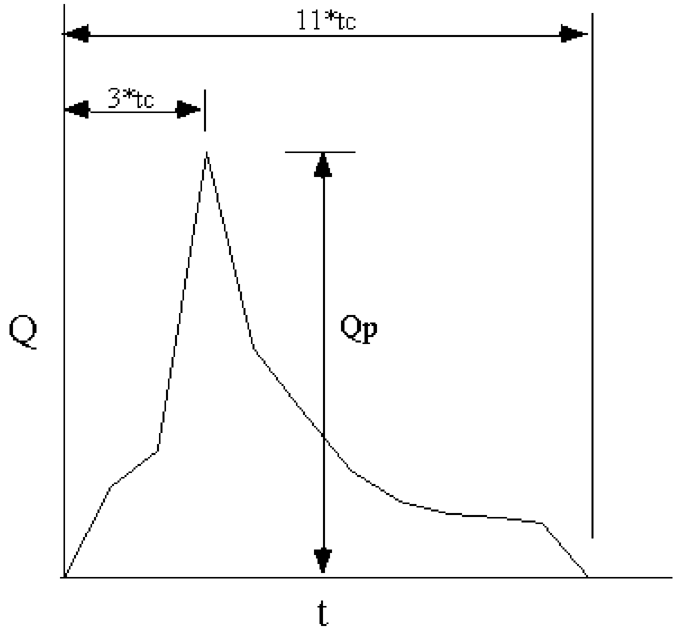
Universal Rational Hydrograph

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.
- Q_p = Peak flow.

Related Topics

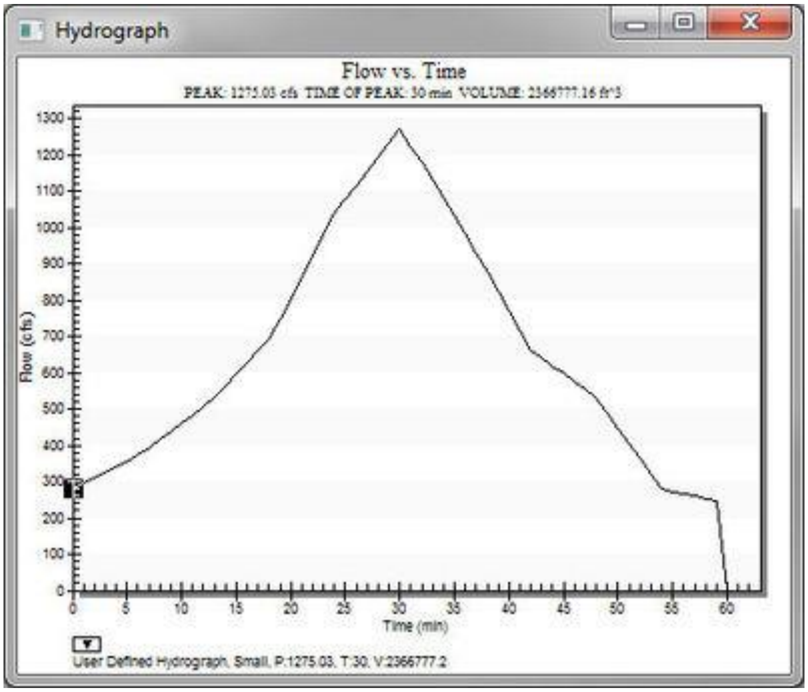
- Computing Hydrographs with Rational Method
- Rational Method Equation

User Defined Rational Hydrograph

With the User Defined method of hydrograph generation, 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 having created a 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 having done this, define model specific data on the hydraulic schematic in the river module .

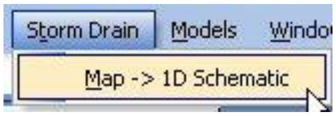
In other words, here are the steps to take to define a simple urban hydraulic model:

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 model-specific data in the river module (currently, only SWMM model data is supported).

The Storm Drain model can be added to a paid edition of WMS.

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.

Reading Storm Drain Shapefiles and LandXML Files

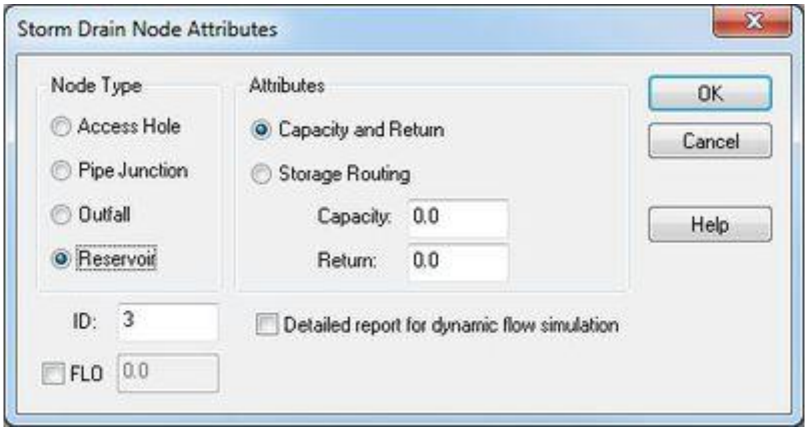
WMS has options to import storm drain attributes from either a set of point and node shapefiles or from a LandXML files. Shapefiles are commonly used GIS vector data files. Government agencies often store storm drain network information in a GIS database and exporting these databases to point and arc shapefiles allows reading the data in these databases into WMS. A LandXML file is a commonly used file format that can be exported from all the major CAD software applications. One feature of LandXML files is the ability to store storm drain, sanitary sewer, or culinary water distribution network information. WMS reads this information directly from LandXML files.

When reading a LandXML file, WMS automatically generates a hydraulic schematic and transfers the information in the LandXML file to the SWMM and HY-12 model inputs. When reading point and arc shapefile data, convert the shapefiles to feature objects in a storm drain coverage, map any attributes that need to be mapped, and make sure the storm drain network is topologically correct. When using a shapefile, select the **Map to 1D Schematic** command after making sure the storm drain network is properly defined in a storm drain coverage in the map module.

FHWA Storm Drain Simulations

NOTE: The FHWA Storm Drain coverage and model has been deprecated in versions of WMS after 9.1. This model can still be used, but it will be taken out and replaced by the HY-12 storm drain model in future versions.

FHWA Storm Drain Nodal Attributes



Run Simulation

Build an FHWA Storm Drain simulation by creating *Storm Drain-FHWA* type coverage. The *Storm Drain* coverage type (NOT *Storm Drain-FHWA*) is used for building HY-12 and SWMM models.

When running 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

After running the FHWA storm drain model and creating a solution file, plots of results can be created by reading the output file using the **Read Solution** command and then setting up an *FHWA Storm Drain* plot using the *Plot Wizard* .

Save Simulation

If there are defined arcs representing pipes and nodes in a Storm Drain-FHWA coverage, WMS can save the simulation using the **Save Simulation** command. WMS will save a Storm Drain model in the correct format to run the FHWA storm drain model.

Storm Drain Feature Objects

The Storm Drain Coverage used to generate HY-12 and SWMM models does not have any attributes for feature objects. This coverage is for storing the storm drain geometry. The FHWA Storm Drain Coverage allows attributes for Storm Drain nodes.

Related Topics

- Storm Drain Job Control
- Plot Wizard
- HY-12

•SWMM

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

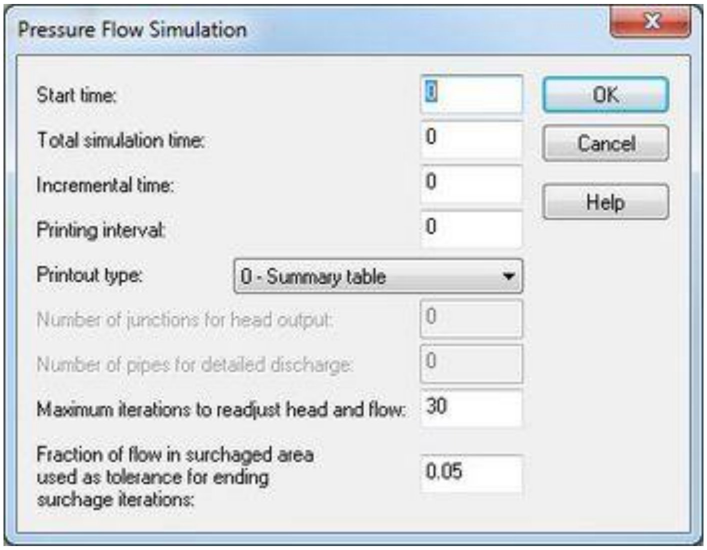
Storm drain can be run using either Metric or English units. Parameters entered should be consistent with the units specified.

Print Hydraulic Grade Line

Turn this option on to see a printout of the hydraulic grade line in the output file.

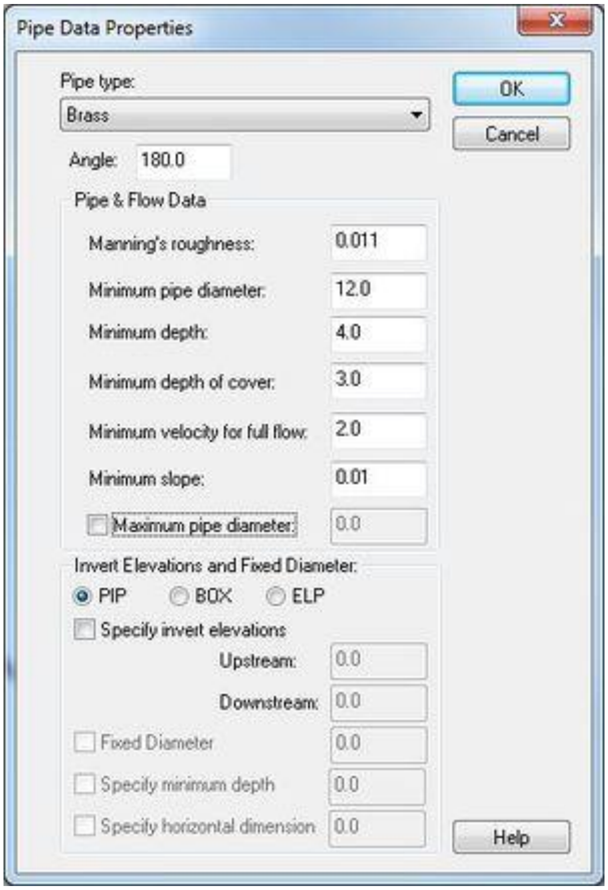
Run Pressure Flow Simulation

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).

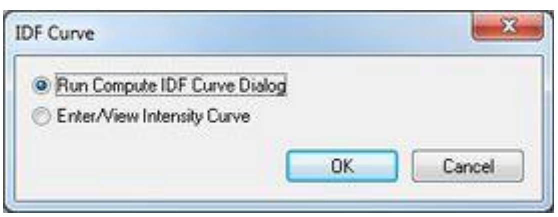


Pipe Data Properties

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 entering 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, 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

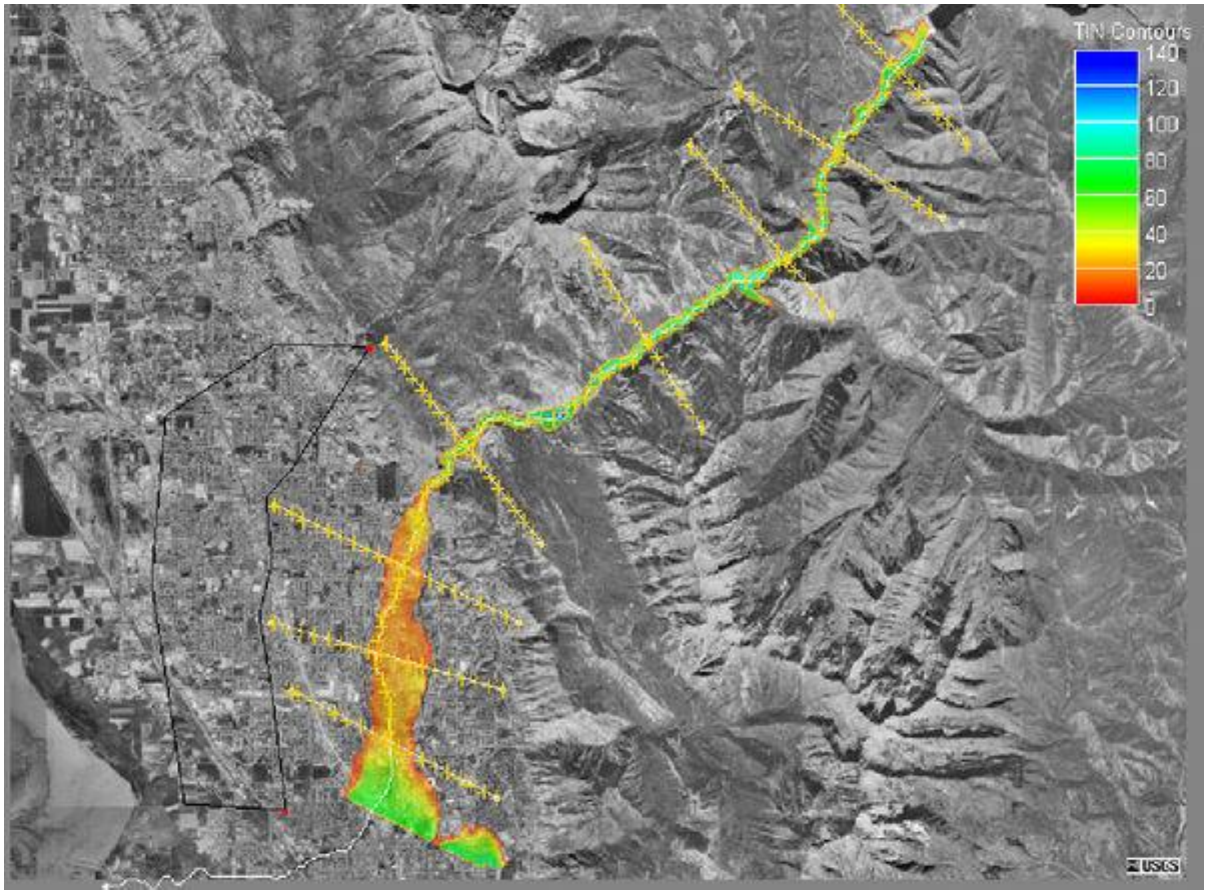
6.13. Simplified Dam-Break (SMPDBK)

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.

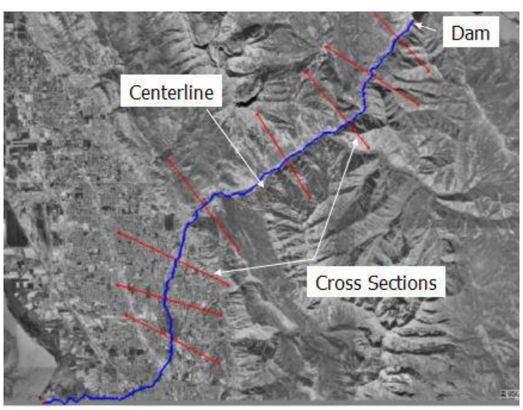
The SMPDBK model can be added to a paid edition of WMS.

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

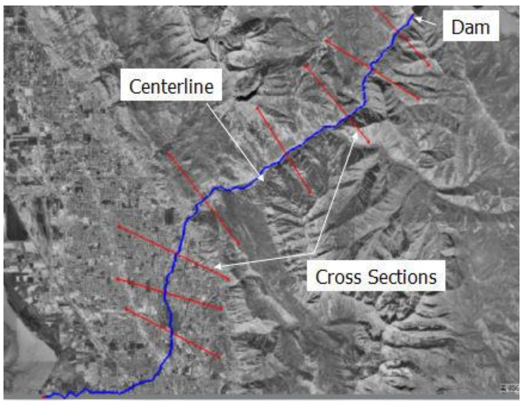
The centerline must be defined in a 1D-Hydraulic Centerline coverage and consists of a single feature arc that should start at the dam being modeled and end at or just after the last cross section. The centerline determines the extents of the simulation and helps to establish the stationing of cross sections that are defined along its length. Create centerlines from the upstream (dam side) end to the downstream end in order to provide the proper direction to the model.



Cross Sections

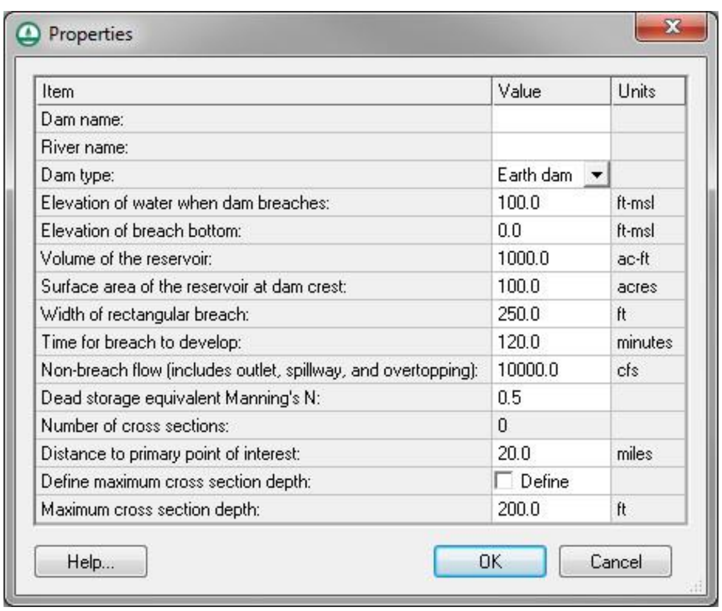
Cross sections for a simplified dam-break analysis must be defined in a 1D-Hydraulic Cross Section coverage. A minimum of two cross sections are required (upstream and downstream ends of the river to be modeled), but sufficient cross sections to define the floodplain should be included. If using a digital 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 (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, a prompt will ask 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.

Important note: If running on a 64-bit Windows operating system, it will not be possible to run SMPDBK from WMS. Run SMPDBK from a DOS command prompt by installing a DOS emulation program (see the SMPDBK Tutorial PDF on the WMS Tutorial page).

Export Simulation

If wanting to save a completed SMPDBK input file that is properly formatted, choose the **Export SMPDBK File** command.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.

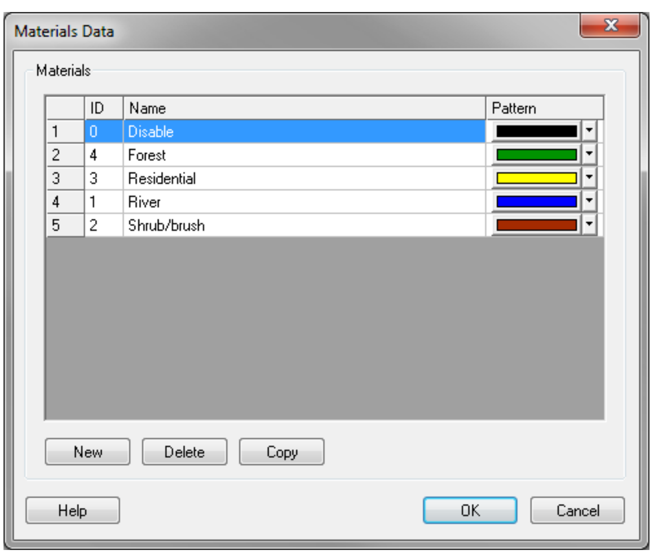
Related Topics

- Defining Roughness
- Post Processing
- Automated Floodplain Delineation

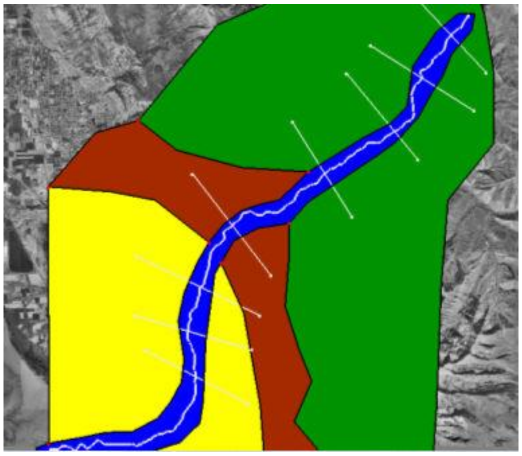
Defining Roughness

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.

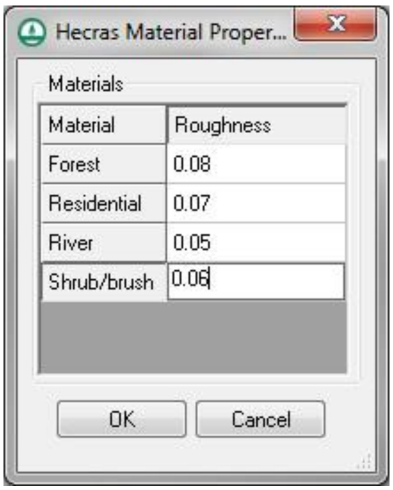
1. Create a master set of materials (land use) types that define the possible choices to be found in the floodplain being modeled. Typically, define as many materials as there will be defined manning's roughness values. This is done using the **Materials** command in the *SMPDBK* menu.



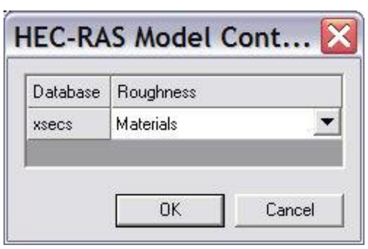
2. Create an Area Property Coverage that consists of polygons corresponding to the different land use (roughness) types found in the floodplain. After building the coverage, define each polygon attribute by double-clicking on the polygon and assigning the appropriate roughness value. Generally these 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, 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 use *Flood* | **Delineate** to 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 better visualize the flood extents.
5. If there are areas known to be interpolated incorrectly and are not flooded (see Figure 1), draw a polygon around these regions so the floodplain will not be delineated in these areas. 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).

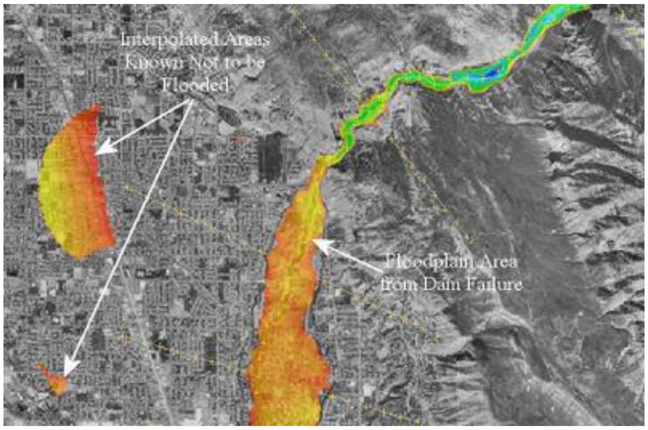


Figure 1 – Areas known to be interpolated incorrectly and are not flooded.

Figure 2 – A flood barrier coverage.

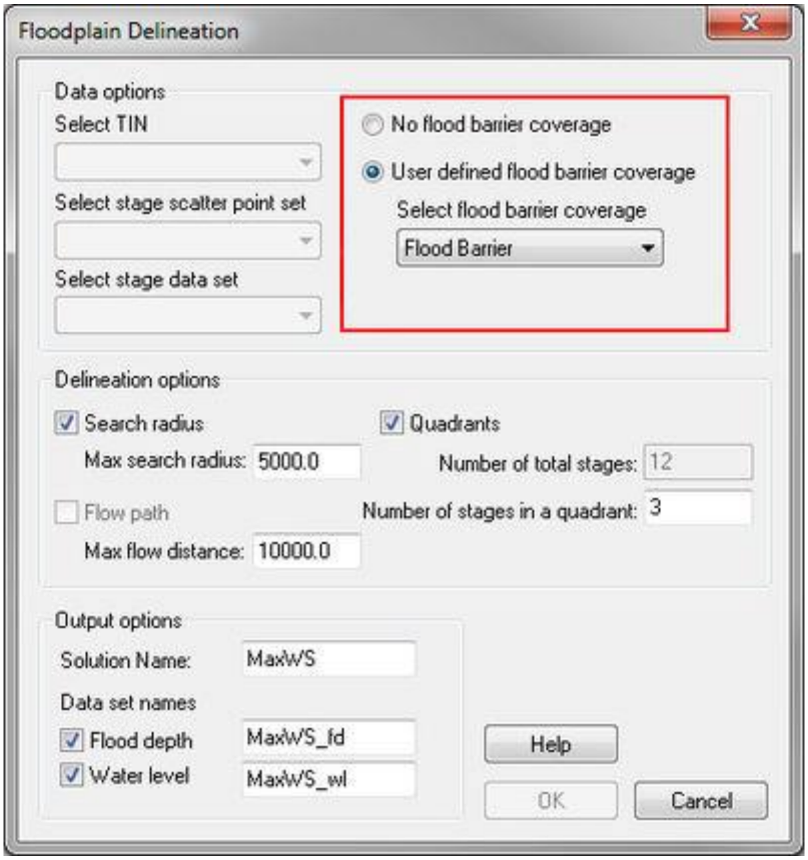


Figure 3 – Floodplan Delineation dialog.

Related Topics

- Simplified Dam-Break Analysis
- Interpolate Results
- Flood Plain Delineation

6.14. SWMM

SWMM

WMS 8.1 and later versions contain an interface to the Storm Water Management Model (SWMM). The SWMM model can be added to a paid edition of WMS. 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.

6.15. TR-20

TR-20

TR20 models can be defined in WMS using the commands in the TR20 menu and resulting dialogs. The TR-20 model is included will all paid editions of WMS. 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 allows editing values for the currently selected hydrograph station.

Model Check

This step allows check ther 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 all necessary TR20 data has been defined. It will report any possible errors/inconsistencies in the 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. It's encouraged to report any additional checks that might be made while working 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 [71] .

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, run the model checker. The model checker will help 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 specifying three 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. View these results by reading this file with the **Open** command from the *Hydrographs* menu.

Once these files have been defined and select **OK** , TR20 will be executed. A separate window will appear and information about the TR20 simulation will be reported. If running with Microsoft Windows, 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 the TR20.EXE file (tr20 for UNIX) is located in the same directory as the WMS executable file.

If TR20 does not run to a successful completion, view the ASCII output file using the **View File** command in the *File* menu.

View Hydrographs

Once a TR20 simulation has been run, view resulting hydrographs using the commands in the *Hydrographs* menu. After viewing the hydrographs, if desired, 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 it may be necessary to define it for all hydrograph stations before being 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 needing 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 running into any other problems reading TR20 files, please contact the distributor WMS was purchased from.

Saving Files

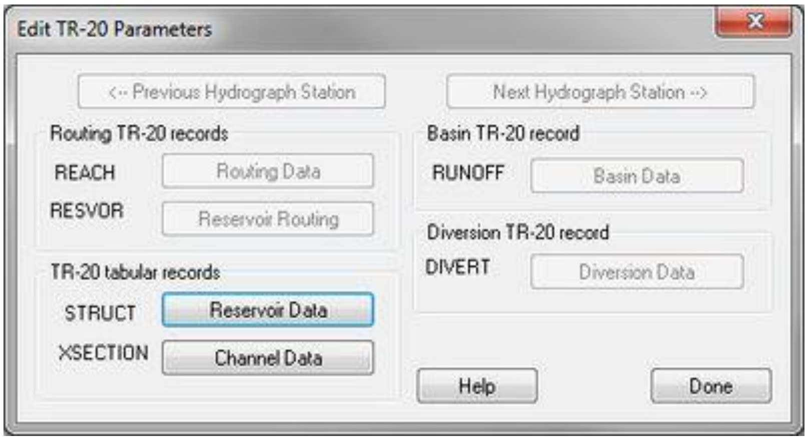
Once a tree has been created and all of the necessary data entered, a TR20 input file can be generated by selecting the **Save TR20 File** command the *TR20* menu. When writing the file, the proper order for computing, adding, and routing hydrographs is automatically determined. TR20 can be run without 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 for which one 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

TR-20 Edit Parameters



Attributes or parameters for all TR-20 hydrograph stations are defined and/or later edited using the *Edit TR-20 Parameters* dialog. This dialog is accessed by selecting the **Edit TR-20 Parameters** command from the *TR-20* menu or when TR-20 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 TR-20 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 selecting the **Done** button. Therefore, continue to select additional, or other, hydrograph stations so that data for that object may be edited without exiting the dialog. Use the **Previous** and **Next Hydrograph Station** buttons to cycle through hydrograph stations in the order they are computed by TR-20. While the dialog is up, all menu commands are active.

Related Topics

•TR-20 Overview

•Entering Job Control Parameters

•Entering Basin Data

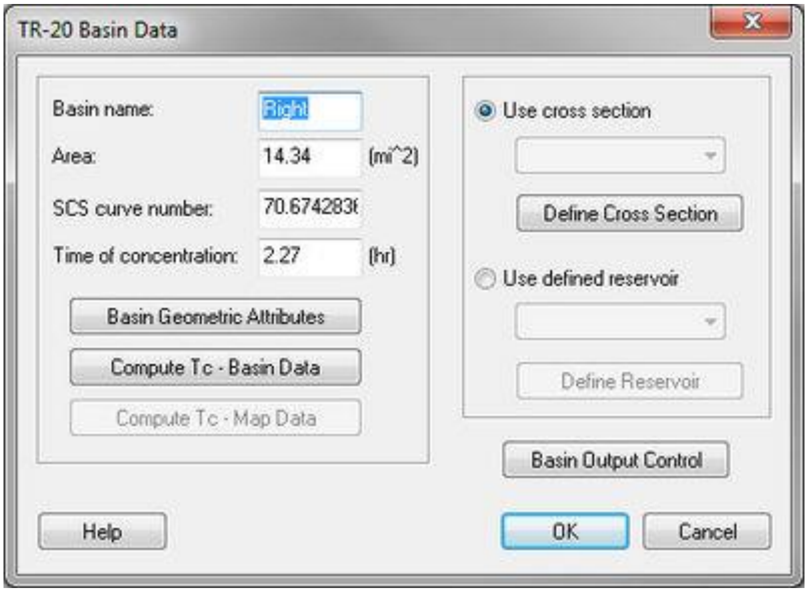
•Entering Routing Data

•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 selecting and assigning 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 *TR-20 Basin Data* dialog by choosing the **Basin Geometrical Attributes** button.

Basin Geometric Attributes

When drainage data are computed, 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, define the type of computation method to use in computing the time of concentration for TR-20. 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 TR-20 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.

See Travel Times from Basin Data for more information.

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 the basin runoff computations. By selecting the **Define Cross Section** button, 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 the basin runoff computations. By selecting the **Define Reservoir** button, 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 TR-20 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, define what this constant outflow is. 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, define the cross section for the diversion. TR20 uses a rating curve to divide flow between the two cross sections that comprise the diversion. 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

TR-20 allows routing a hydrograph through a reservoir using the RESVOR TR20 file card. Define these reservoir routing parameters in the *TR-20 Reservoir Routing* dialog. Reservoirs and other types of structures (such as detention basins) can be defined from the *TR-20 Reservoir Data* dialog.

Define reservoir data by pressing the **Define reservoir data** button. This will bring up a dialog where the elevation-discharge-storage relationship for the reservoir can be defined. Enter up to twenty elevation-discharge-storage relationships for each reservoir. To assign a particular set of data to a reservoir, select the reservoir and select the **OK** button on the *TR-20 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 TR-20 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

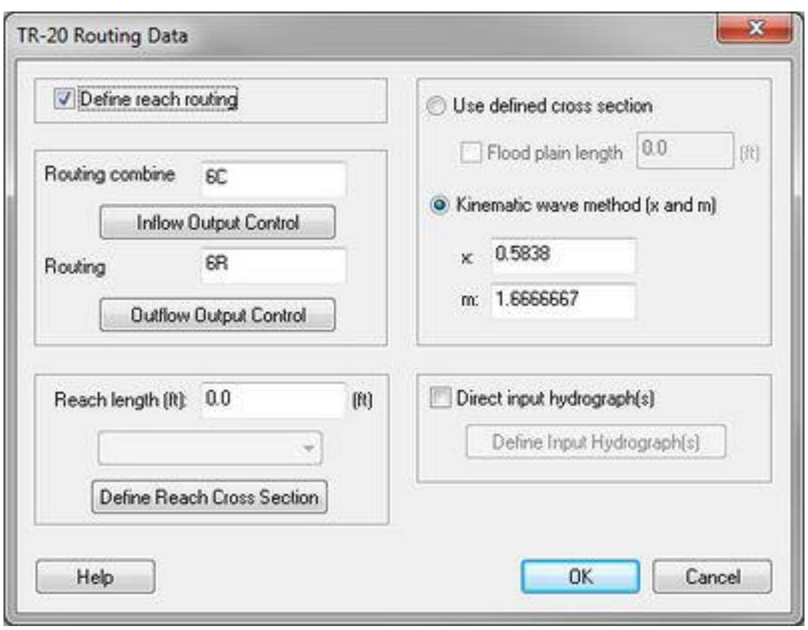
One or more input hydrographs can be defined for a reservoir. WMS will combine any input hydrographs with other input hydrographs to the reservoir. The resulting hydrograph will then be routed through the reservoir. This option is useful for defining measured stream flows into a reservoir.

Related Topics

•TR-20 Overview

•Editing TR-20 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 TR-20 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 TR-20 Parameters* dialog.

Routing Method

TR-20 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 *TR-20 Cross Section Data* dialog, and is accessed by selecting the **Define Cross Section Data** button.

Bankfull Elevation

When entering the bankfull elevation here, it will trigger a warning message in the TR-20 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 TR-20 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 by selecting the appropriate radio button in the *Define Cross Section* dialog. If flow is defined in csm, define the drainage area of the basin in square miles.

Defining Cross Sections

Once a cross section has been defined in TR-20, this cross section can be used in other locations in the TR-20 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 *TR-20 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 TR-20 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

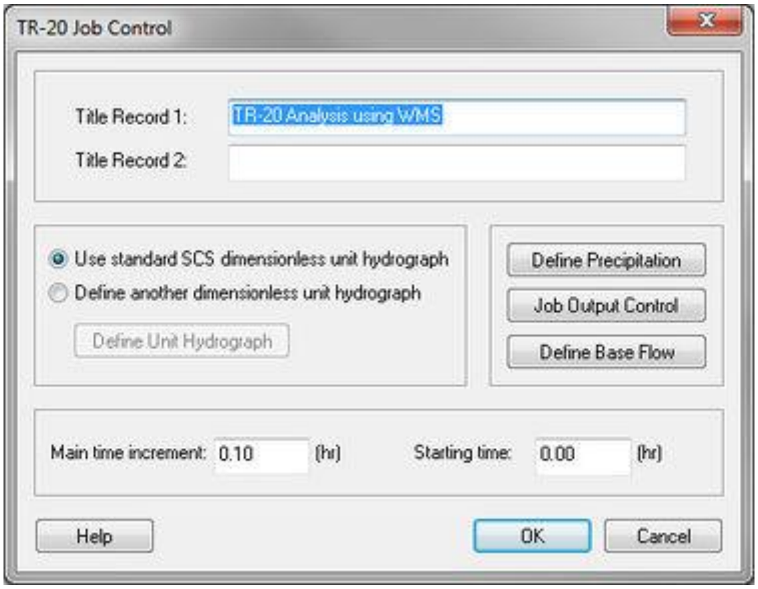
Hydrographs can be input directly, and then routed down stream, using the different routing options. To do this, select the *Direct Input Hydrograph* option. Hit the **New** button in the *Define input hydrographs* dialog to define a new hydrograph. The base flow and contributing drainage area for the 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 TR-20, the discharge in cfs at each time interval must be entered.

Related Topics

- TR-20 Overview
- Editing TR-20 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:

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

TR-20 offers two ways of defining the unit hydrograph used to compute runoff from the watershed. The first way is to use the standard SCS hydrograph provided with TR-20. Another way is to define an 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.

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.

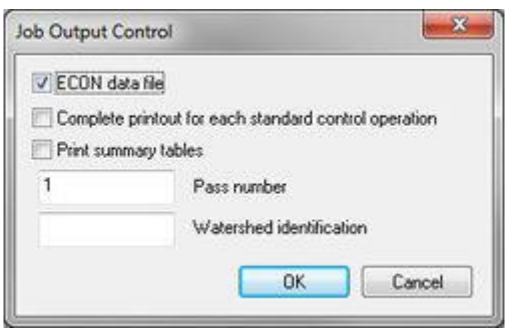
Precipitation

In TR-20, there are two different ways to define precipitation. Precipitation is defined by selecting the **Job Control** command from the *TR-20* menu. Then, select the **Define Precipitation** button to define precipitation over the entire watershed. Define precipitation by using either a standard SCS rainfall distribution or by defining a custom rainfall distribution. If a custom rainfall distribution is used, define a custom rainfall distribution in the *XY Series Editor* . If one of the standard SCS rainfall distributions are used, 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

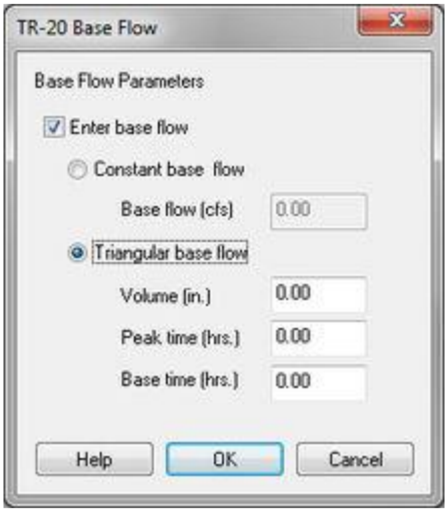
For all the standard distributions, define the rainfall depth. For the dimensionless distribution, also define the rainfall duration. It's only necessary to define the depth and/or duration of the rainfall for a custom distribution if one of the units entered in the *XY Series Editor* is dimensionless.

Output Control



Base Flow

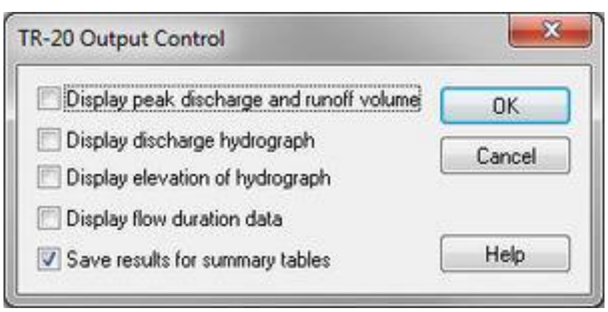
Base flow can be entered in TR-20 from the *Job Control* dialog by selecting the **Define Base Flow** button. There are two ways for defining base flow in TR-20. One way is to define a constant base flow. The second way is to define a triangular base flow in the watershed by specifying a volume in inches, a peak time in hours, and a base time in hours. By selecting the appropriate option, define either a triangular or a constant base flow for the 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** 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

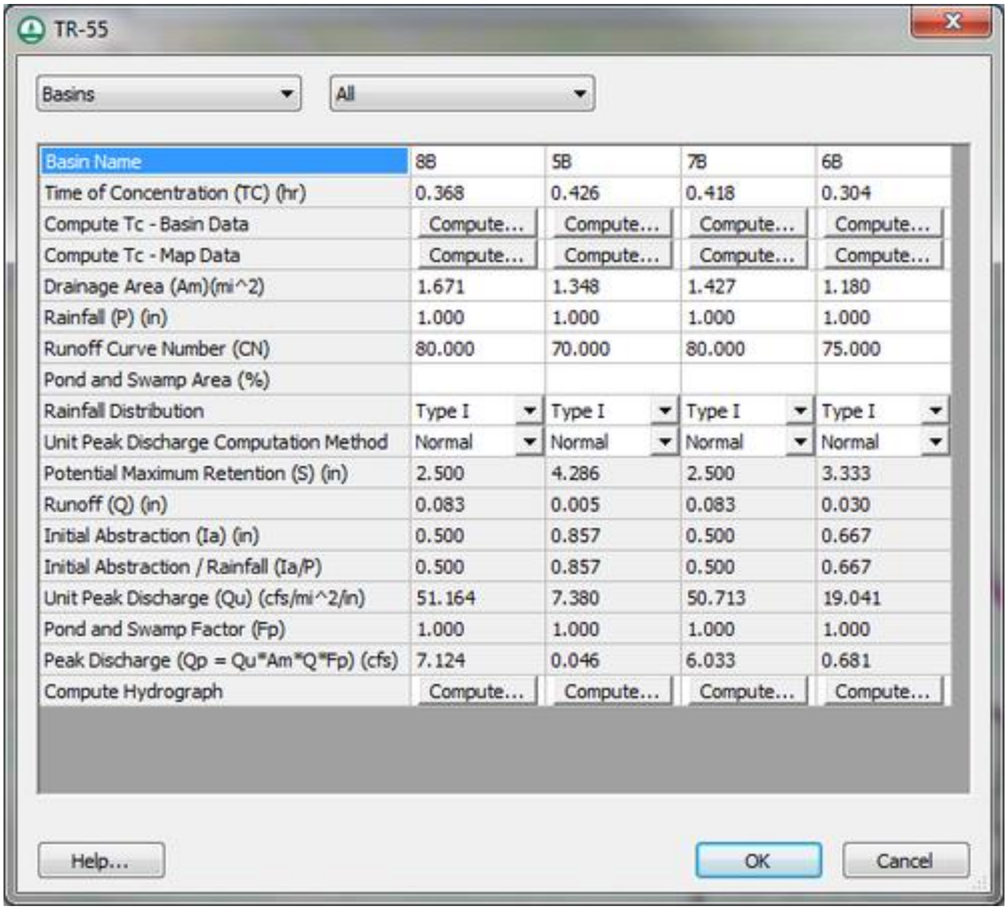
Control of the output files can be specified individually for each of the reaches or outlets. The output control parameters for the inflow hydrograph corresponds to the ADDHYD record whereas the parameters for the outflow hydrograph correspond to the REACH record. Output control parameters for all hydrograph stations are identical and are discussed in an earlier section.

Related Topics

- TR-20 Overview
- Editing TR-20 Parameters

6.16. 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, read "Technical Release 55: Urban Hydrology for Small Watersheds, 2nd Edition."

WMS uses its own format for saving or reading a TR-55 simulation. When in the Hydrologic Modeling Module, select *TR-55* from the drop-down list of models to make TR-55 the active model. To edit the TR-55 parameters and run a TR-55 simulation, select the outlets or basins to edit and/or run and select the **Run Simulation** menu item from the *TR-55* menu.

The TR-55 model is included will all paid editions of WMS.

Related Topics

- Basin Data
- Outlet Data
- Hydrographs

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 having 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 having 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 combining 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 [72]) 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 defining the watershed as a single basin or 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 creating a watershed with multiple sub-basins, 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. If desired, 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 there is a basin selected when computing hydrographs, then a hydrograph for only the selected basin will be computed. However, if there is an outlet selected when computing hydrographs, then a new hydrograph for each upstream basin will be computed along with the routed hydrographs at outlet points.

There may be 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/P value 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/P value.

Discrepancies Between the TR-55 Hydrograph and the Unit Peak Discharge Method

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 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. Using the manual method gets the same basic 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 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 are already lagged when running TR-55 in WMS. 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 selecting the *TR-55 data* window, it 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 controlling which information is 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

6.16.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, it is also possible to enter a user defined 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:

1. Sheet Flow Equation
2. Shallow Concentrated Flow
3. Channel Flow

TR55:

1. Sheet Flow Equation
2. Shallow Concentrated Flow Equation
3. 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 having developed a watershed model from a TIN or DEM, it is possible to 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, the *Modify Equation* dialog can be used 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, one of the pre-defined equations can be selected from the *Sample equations* drop-down list and add it to the equation line as a starting point to create a new equation. Any of the variables that WMS can manage can be used as part of the equation (e.g. length, slope, rainfall intensity, etc.). However, it is not possible to 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 the equation, 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 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, 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, 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).

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 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, 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:

t_c = 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 $K_b = m\log(A) + b$ in the Maricopa County t_c Equation

Type	Description	Typical Applications	Parameters	
			m	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, it is necessary 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. Modeling

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)

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/ and then go to the GeoSpatial Data Acquisition Page http://www.xmswiki.com/xms/GSDA:GSDA Some of the primary locations there are:

DEMS: http://nationalmap.gov/viewer.html

LandUse: http://www.webgis.com/

Soils: http://websoilsurvey.nrcs.usda.gov/

Aquaveo: http://www.aquaveo.com/

The LSU site can also be helpful, especially for DRG Images of USGS Maps. To download and use these maps:

1. Go the site: http://atlas.lsu.edu/
2. Choose **Download Data**
3. Choose the map scale such as **http://atlas.lsu.edu/q24k/ Scanned Topographic Maps**
4. Use the map to navigate to the area to be utilized.
5. Select the map
6. Choose the **Download** button
7. Unzip the map on the 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. The image is now in WMS. Repeat for other Quads as necessary.

Louisiana DOTD Models Using the NRCS (SCS) Method

Creating an NRCS (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 the study area – make sure it is “bigger” than the study area
3. With *Web Services* download the DEM (10 meter cell size recommended) and Topo map (aerial photograph too if desired)
    a) 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.
    b) Download the land use file(s) from http://www.webgis.com/
    c) Ask for the Ssurgo data and get an email with an ftp from: http://websoilsurvey.nrcs.usda.gov/
    d) Read these shapefiles into WMS and be sure they are converted to the UTM, NAD83, Zone 15 coordinate system

e) If there are problems with the topo map, go to http://atlas.lsu.edu/ and navigate to download the DRG Files

f) For the DEM, go to http://nationalmap.gov/viewer.html/

4. Delineate the watershed.

a) Turning on the *Create TC Coverage* is optional

5. Select the HEC-HMS model (this won't be used but it will line up best with what to do for the NRCS method).

6. In the *Define Land Use* option

a) Join the Ssurgo attributes tables by choosing the *Join NRCS Data* option

b) Back in the wizard Create the Land Use and Soils coverage

7. Under *Hydrologic Computations* in the Wizard.

a) Compute GIS Attributes to determine the composite CN

b) In this load the standard LADOTD NRCS table (the current definition for Residential is for ¼ acre lots, some may want to adjust this value or any other as appropriate). This table can be downloaded here.

c) Compute Basin Data and select the **Lag Time – SCS** equation.

8. Set up the Job Control and Precipitation data for an HMS model if desired, but it is not necessary.

9. Clean up the model.

a) Turn off the *Model Checker*

10. Exit the wizard and identify the data necessary to run the model.

11. Go into the *Display Options* under *Drainage* and turn on the *Maximum Flow Distance and Maximum Flow Slope* .

12. Save the WMS Project

13. Use NOAA Atlas 14 or the LADOTD Hydraulics manual Figure 3.4-2 and Table 3.4-2 to get the desired rainfall depth.

14. Run the model with the standard LADOTD program.

a) http://wwwsp.dotd.la.gov/Inside_LaDOTD/Divisions/Engineering/Public_Works/Hydraulics/Pages/Hydraulics_Software.aspx

LADOTD NRCS 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 the 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 desired)
    a) If there are problems with the topo map, go to http://atlas.lsu.edu/ and navigate to download the DRG Files
    b) For the DEM, go to http://nationalmap.gov/viewer.html
    c) The Land Use and Soils files shouldn't be necessary for these projects because they do not require to compute CN, but if wanting to run HMS or something else then refer to the SCS document to process the land use and soils files
4. Delineate the watershed
    a) Before delineating turn on the *Create TC Coverage* option
    b) Set the *Accumulation Threshold* to 1-3 sq. miles
5. Clean up the model
    a) Turn off the *Model Checker* if desired
6. Exit the wizard and identify the data necessary to run the model.
7. Use the Maximum Flow Slope *5280 to get feet/mile
8. Save the WMS Project.
9. Open the Louisiana Mean Annual Rainfall .wms file file and determine the Mean Annual Rainfall.
10. Run the model with the standard LADOTD program.
    a) http://wwwsp.dotd.la.gov/Inside_LaDOTD/Divisions/Engineering/Public_Works/Hydraulics/Pages/Hydraulics_Software.aspx

Maricopa County Modeling

Tips and Locations for Downloading Hydrologic Data

GSDA Site

HEC-1 Modeling with Green and Ampt

1. Read the DEM into WMS (download DEMs from the USGS Seamless site).
2. Convert coordinates of the DEM when reading:
   a) **Convert from:** Horizontal system of Geographic NAD 83 and vertical system of NAVD 88, Meters.
   b) **Convert to:** State Plane NAD 83, Arizona Central Zone, horizontal and vertical units of US Survey feet, NAVD 88.
3. Read into WMS any image data to be used.
4. Run Topaz (Drainage module)
5. Set the flow accumulation threshold (*Display Options* | select **DEM Data** option)
6. Create the outlet(s) (Drainage module tools)
7. Use the delineate basins wizard
8. Read the Land Use shapefile. Download land use shapefiles from here . The file ending in "giras.exe" will be the file needed. The soils shapefile will be in the file ending in "core31.exe". A HUC number is needed for the watershed, whichcan be gotten by searching for the watershed on this site .
9. If needed, convert the coordinates of the 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 the watershed, and select the *Mapping* | **Shapes→Feature Objects** command.
12. Make sure the land use ID is mapped to the correct field in the land use coverage. Also map the land use name if needed.
13. Read the soils shapefile. The soil shapefile for Maricopa County can be downloaded with a bunch of other data in the FCDMC.zip file here . 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 . "Ssurgo" data can be downloaded from the soil data mart here .
14. If needed, convert the coordinates of the 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 the watershed, and select the *Mapping* | **Shapes→Feature Objects** command.
17. Make sure the soil type ID is mapped to the correct field in the soil type coverage. 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. 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 downloaded here .

19. Define the necessary *Green and Ampt* parameters for each soil type in the soil type coverage. 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 downloaded here .

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 the 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 that was 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 the basin and view the 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.

Tutorials

Download Maricopa County Tutorials at http://wmsdocs.aquaveo.com/

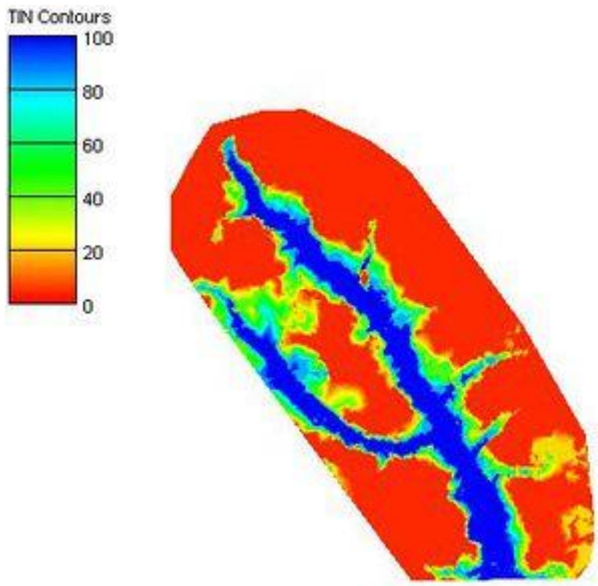
Additional Information

More information about the Maricopa County interface development is located at http://emrl.byu.edu/FCDMC/

Stochastic Modeling

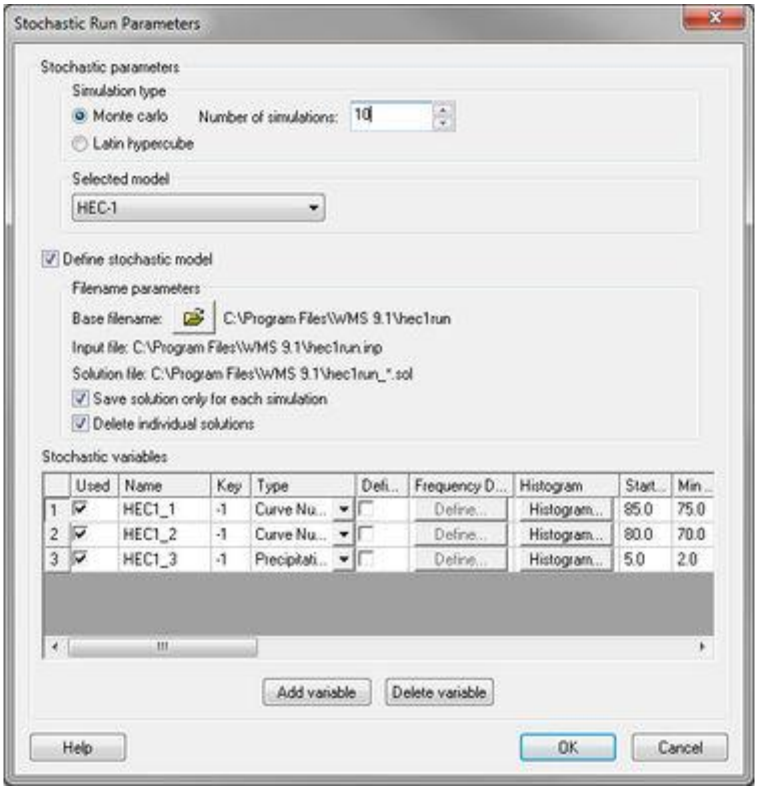
There is always a great deal of uncertainty in hydrologic and hydraulic modeling and the parameters that are used to develop solutions. Despite this, a typical flood plain boundary is black and white in that the project is either in or out of the flood plain. A good engineer might be able to dispute 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.

It is possible to 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:



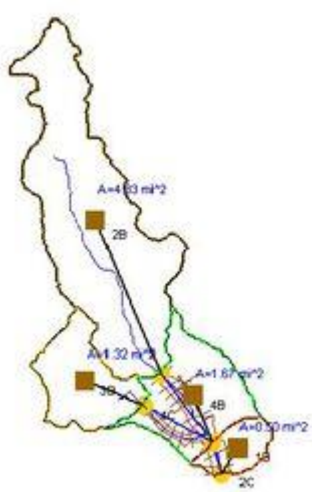
The following steps outline the process for developing a stochastic model of floodplain boundaries using WMS, HEC-1, and HEC-RAS.

Step 1. Develop a hydrologic model with HEC-1

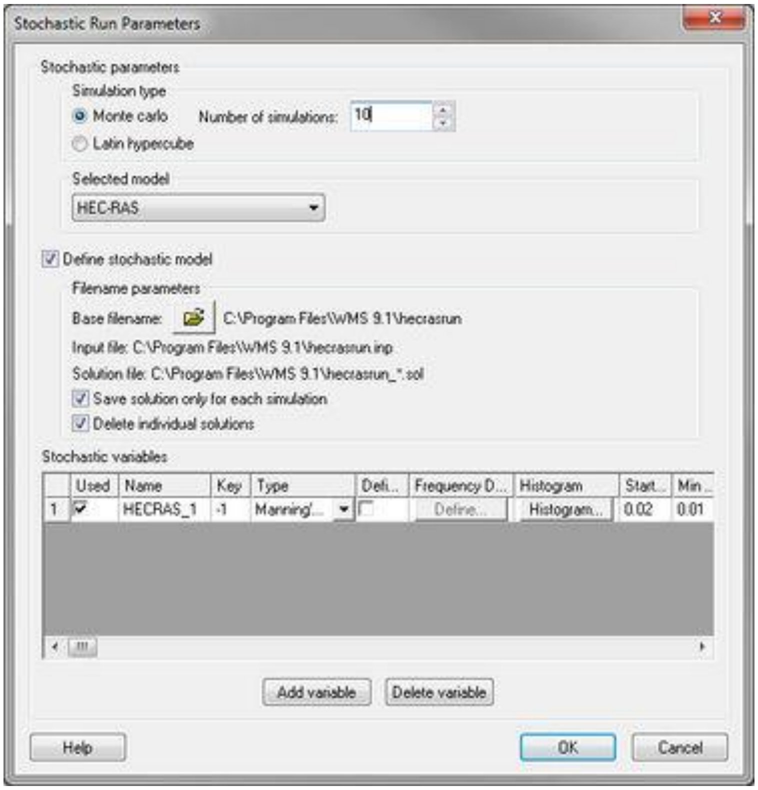


The HEC-1 Interface tools in WMS can be used to develop a working HEC-1 model. It is important that this model be running effectively (i.e. calibrated and/or adjusted to give credible results) prior to using it for the Stochastic Modeling simulation.

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 created. When running the stochastic simulation WMS will substitute the simulation specific parameter for the defined key. Then setup a stochastic variable for HEC-1 in the *Stochastic Run Parameters* dialog. A key value (matching the key 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 created. When running the stochastic simulation WMS will substitute the simulation specific parameter for the defined key. Then setup a stochastic variable for HEC-RAS in the *Stochastic Run Parameters* dialog. A key value (matching the key defined in the materials property) starting value, minimum value, maximum value, standard deviation and distribution type.

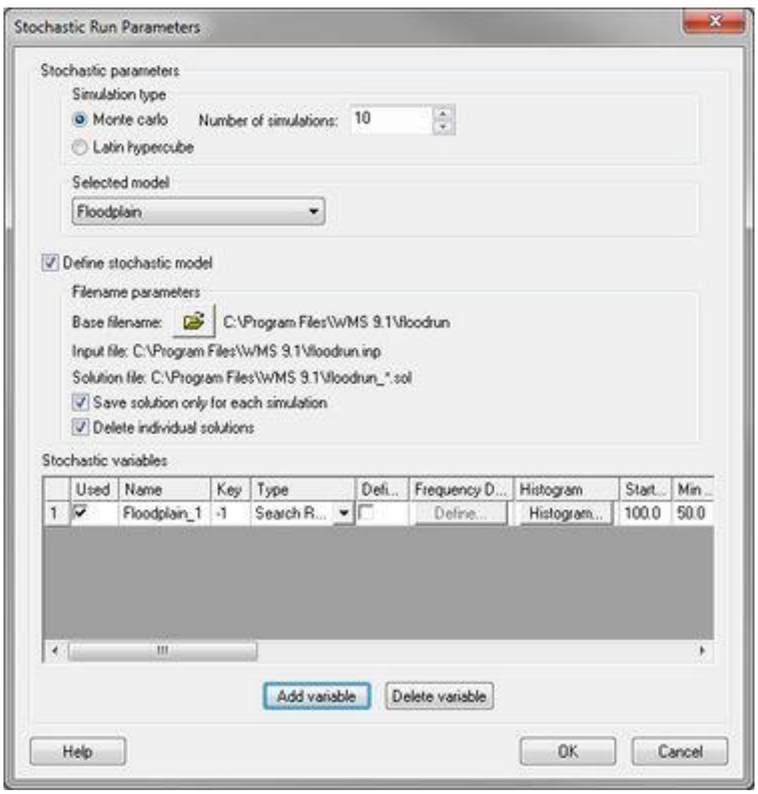
Step 3. Establish appropriate flood plain delineation parameters for the area

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 created. When running the stochastic simulation WMS will substitute the simulation specific parameter for the defined key. Then setup a stochastic variable for Floodplain in the *Stochastic Run Parameters* dialog. A key value (matching the key 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, it is easier 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 including a basin input file, a solution files directory needs to be defined. For HEC-1, TR-20, or HEC-RAS, 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.

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.

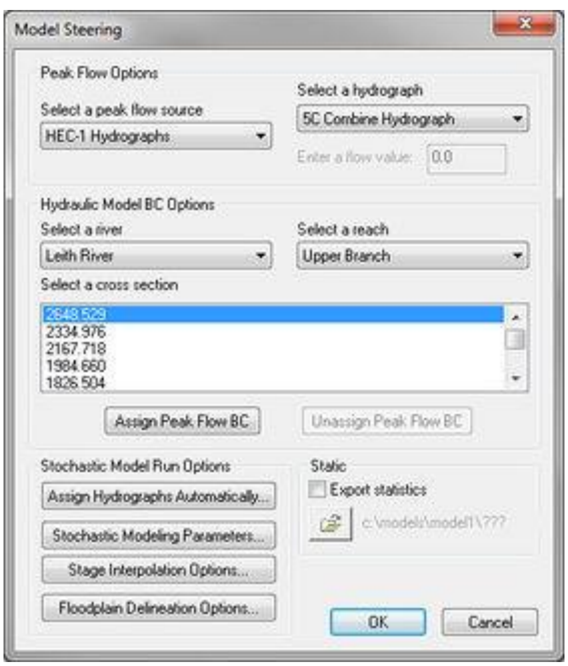
Step 5. Assign boundary conditions between models and run the stochastic simulation

The final step before running a stochastic simulation is to link any required models. In particular, it's necessary to link how the hydrologic modeling results are used as boundary conditions for the HEC-RAS hydraulic model. When choosing the **Run Stochastic Model** command, the *Model Steering Dialog* appears. For the hydrologic model (HEC-1), assign the appropriate hydrograph (basin or outlet) to the river reach and section in the hydraulic model.

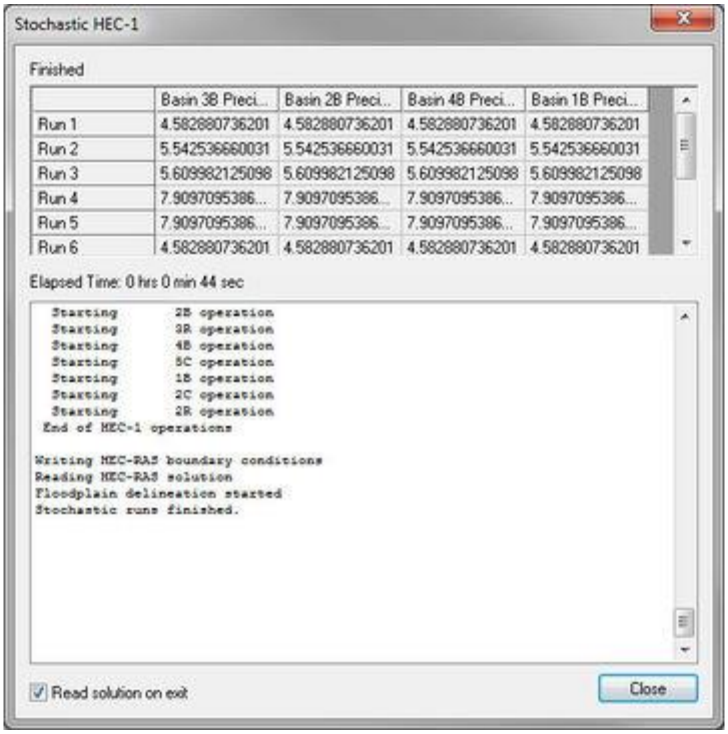
The option to *Assign Hydrographs Automatically* can be used providing a drainage coverage and centerline coverage are being used to set up the hydrologic and hydraulic models.

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** the model simulation will run.



Step 6. Run the simulation



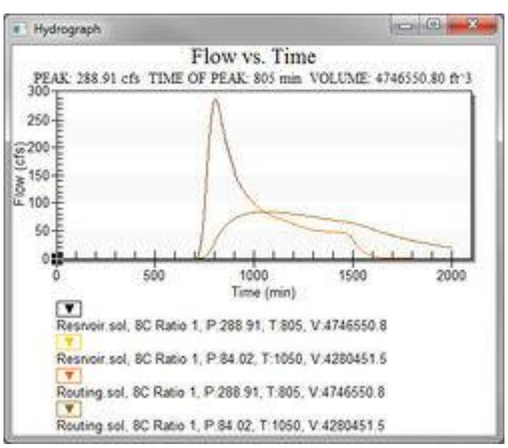
Once the simulation begins running, notice the parameters selected for each model run, as well as the status of each run. The *Read solution on exit* toggle is on by default and will cause that all model solutions (hydrographs, water surface elevations, and floodplain delineations) are read when the 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.

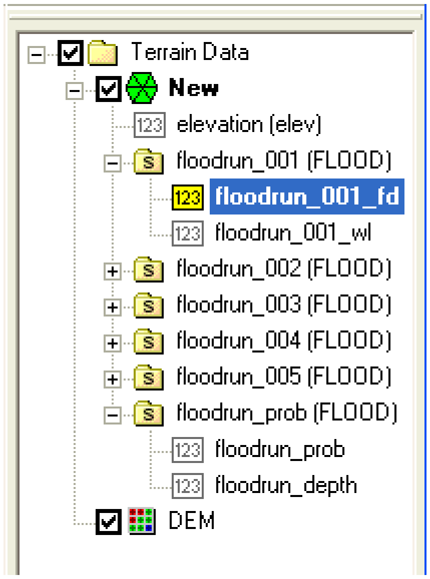
Hydrographs

A series of hydrographs are loaded for each hydrograph station and can be viewed in the normal way hydrographs are viewed.



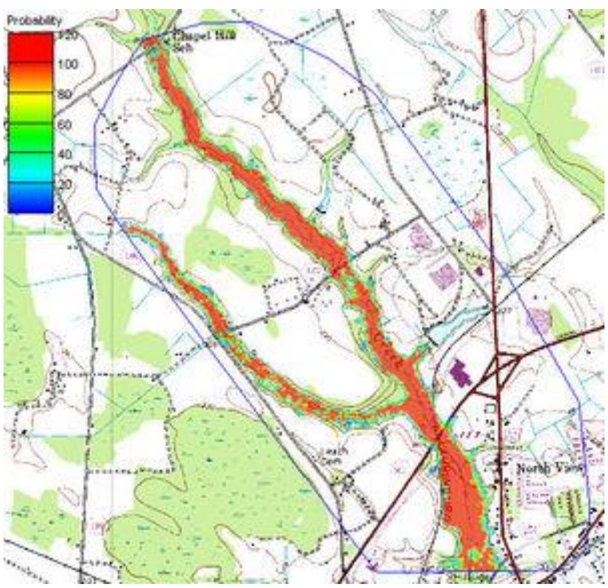
Floodplains

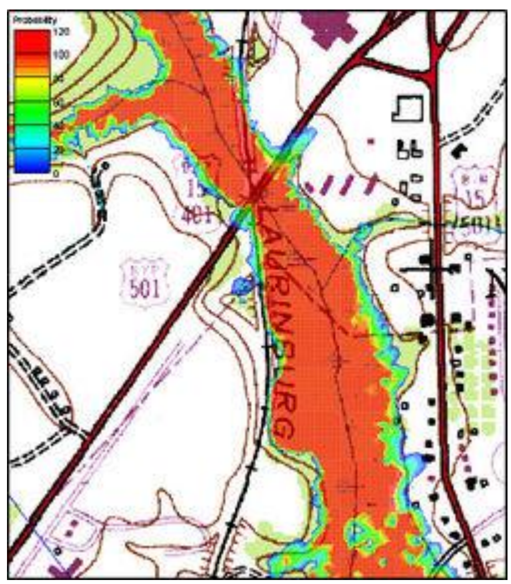
Each floodplain delineation results in a water surface elevation and a flood depth dataset. Each pair of datasets are organized in a folder underneath the TIN in the *Project Explorer* . Set the contour options for a TIN and select the dataset to be active and displayed from the *Project Explorer* .



Probabilistic Floodplain Boundary

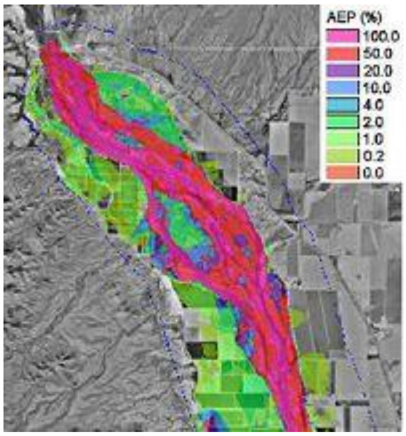
A final dataset showing the probability of flooding is also created from all of the individual floodplain datasets. The probabilistic flood plain indicates for each vertex on the TIN the percentage of model runs that resulting in inundation at the point.

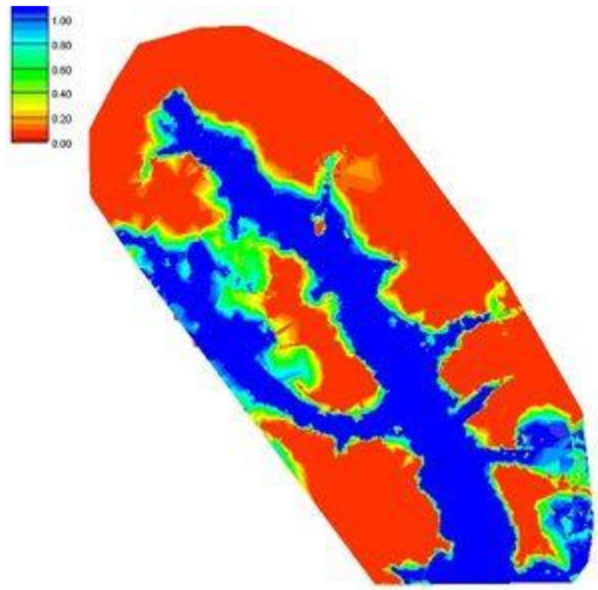




Annual Exceedance Probability (AEP) Map

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 generating contours from the AEP map for specified return periods. 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

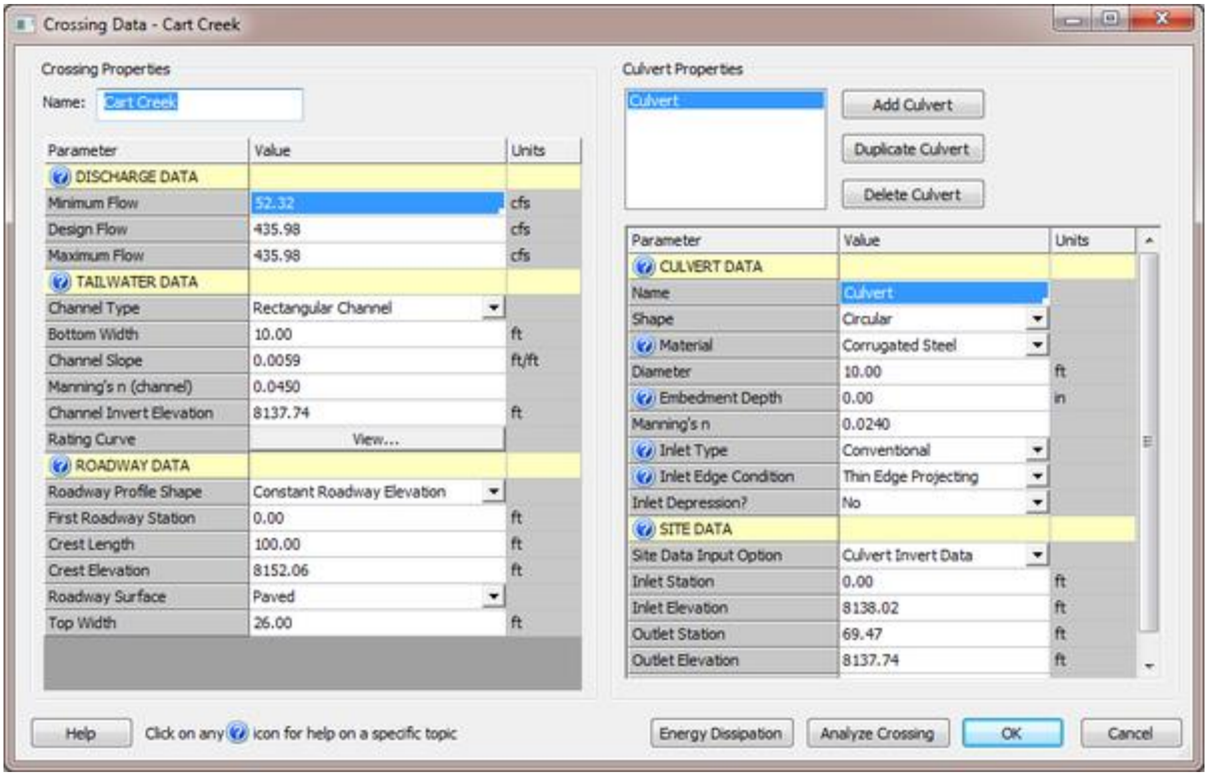
•Flood Plain Delineation

7.1. WMS to HY-8

Path of Data from WMS To HY8

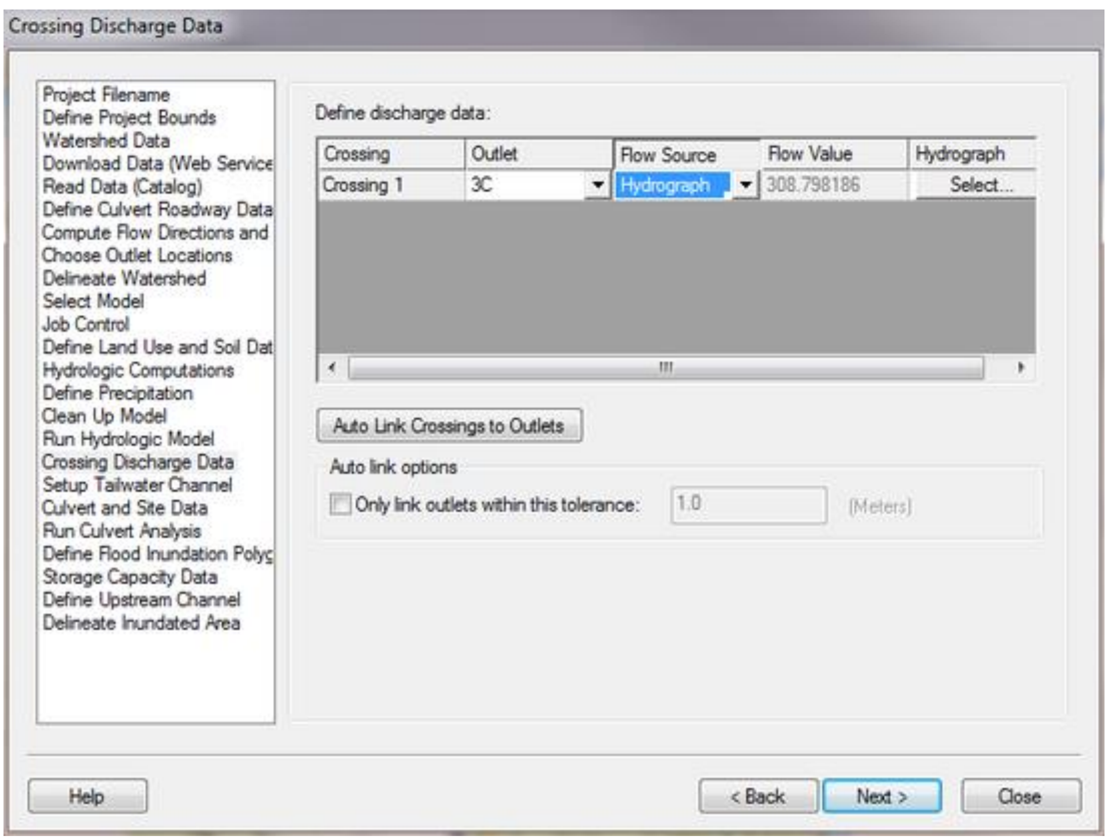
The Path of Data from WMS to HY8 in the HY8 Modeling Wizard

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 .



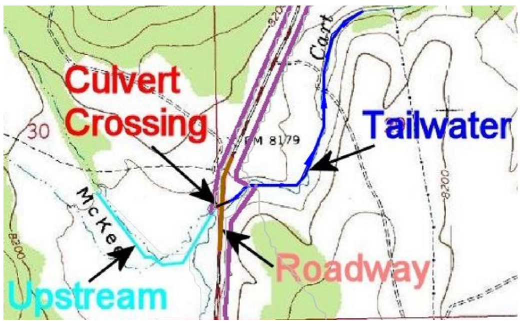
If specifying the flow source as "Design Flow", the minimum flow value will be zero. Enter the design and maximum flow value.

If specifying 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.

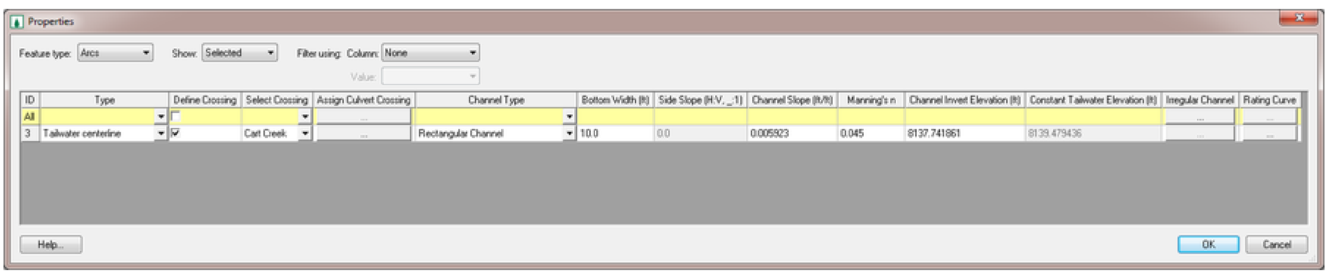
Finally, if specifying "Design Flow & Hydrograph" as the flow source, WMS will use the minimum and maximum flow values from the hydrograph and allow entering the design value.

Tailwater Data

The information for the tailwater is saved in the tailwater arc.

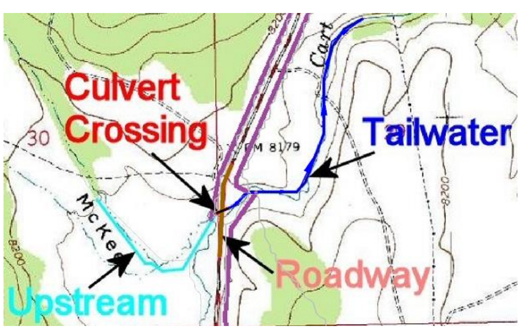


The "Channel Slope" is determined from the slope of the tailwater arc. The "Channel Invert Elevation" is initialized with the value of the elevation at upstream end of the tailwater arc.

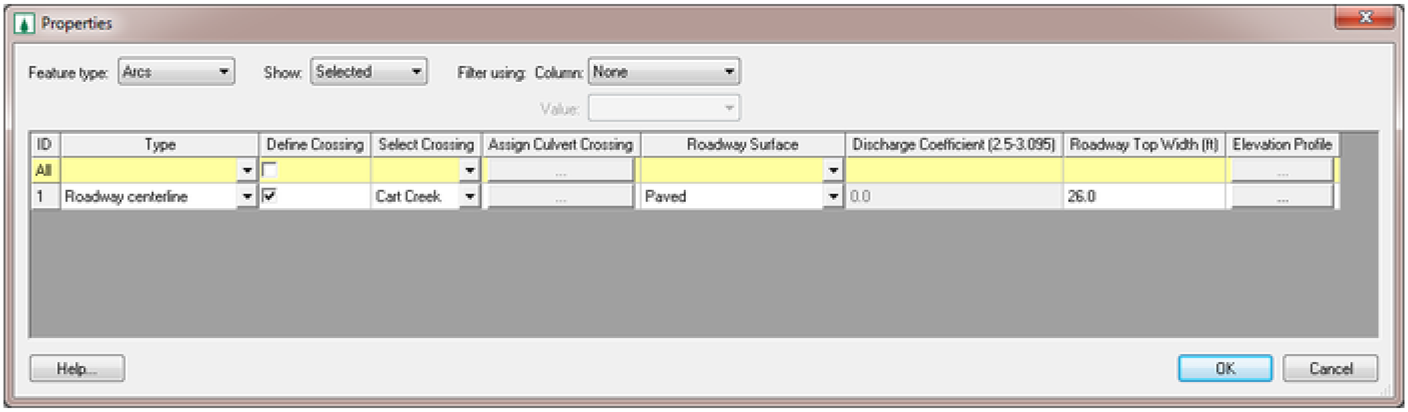


The "Channel Type", "Bottom Width", "Side Slope", "Manning's n", "Constant Tailwater Elevations", "Irregular Channel Cross-Sections", and "Rating Curves" are set in the attribute table for the arc.

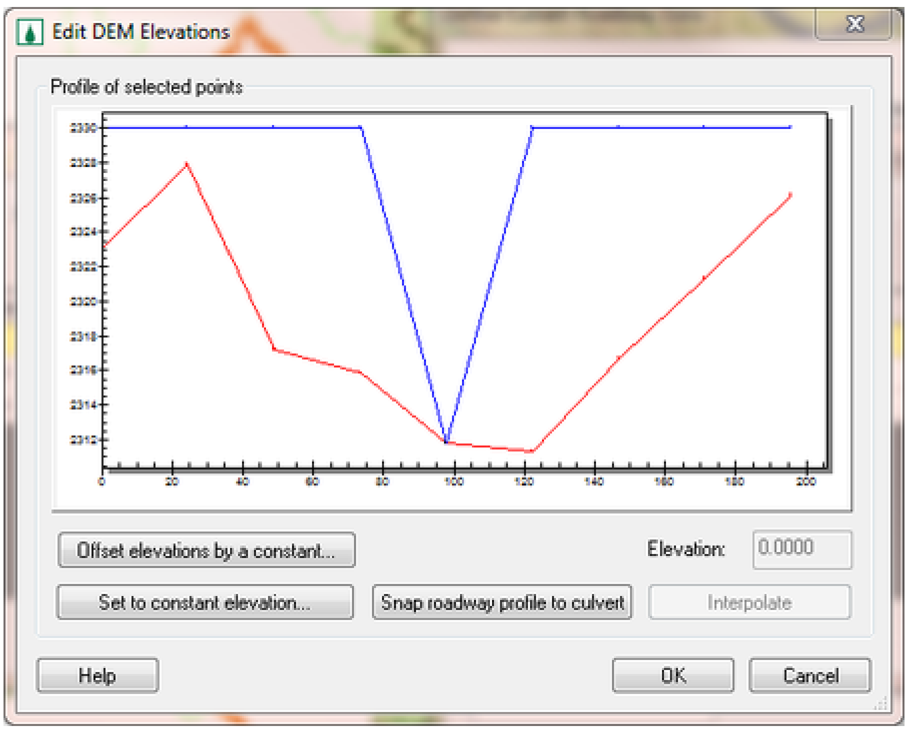
Roadway Data



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" are set in the *Attributes* dialog for the arc. The elevation profile will set whether the roadway has a constant roadway elevation or if it is an irregular shape.



If editing the elevation profile, a plot will appear 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 clicking **OK** on this dialog, WMS will change the DEM under the roadway arc to match the finished roadway. For this reason, 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).

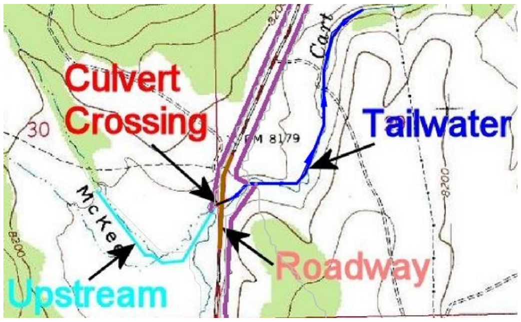
HY8 requires that the irregular roadway has between 3 and 15 points. If the profile has two points after removing the lowest and unnecessary points, WMS will assign it to be a constant roadway elevation. Otherwise it will assign it to be an irregular roadway profile. If the profile has more than fifteen points, WMS will remove the least significant points until it has only fifteen points remaining.

Culvert Data

There is no culvert data that is set in WMS. 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

The WMS HY-8 Modeling Wizard () is a simple tool that guides 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

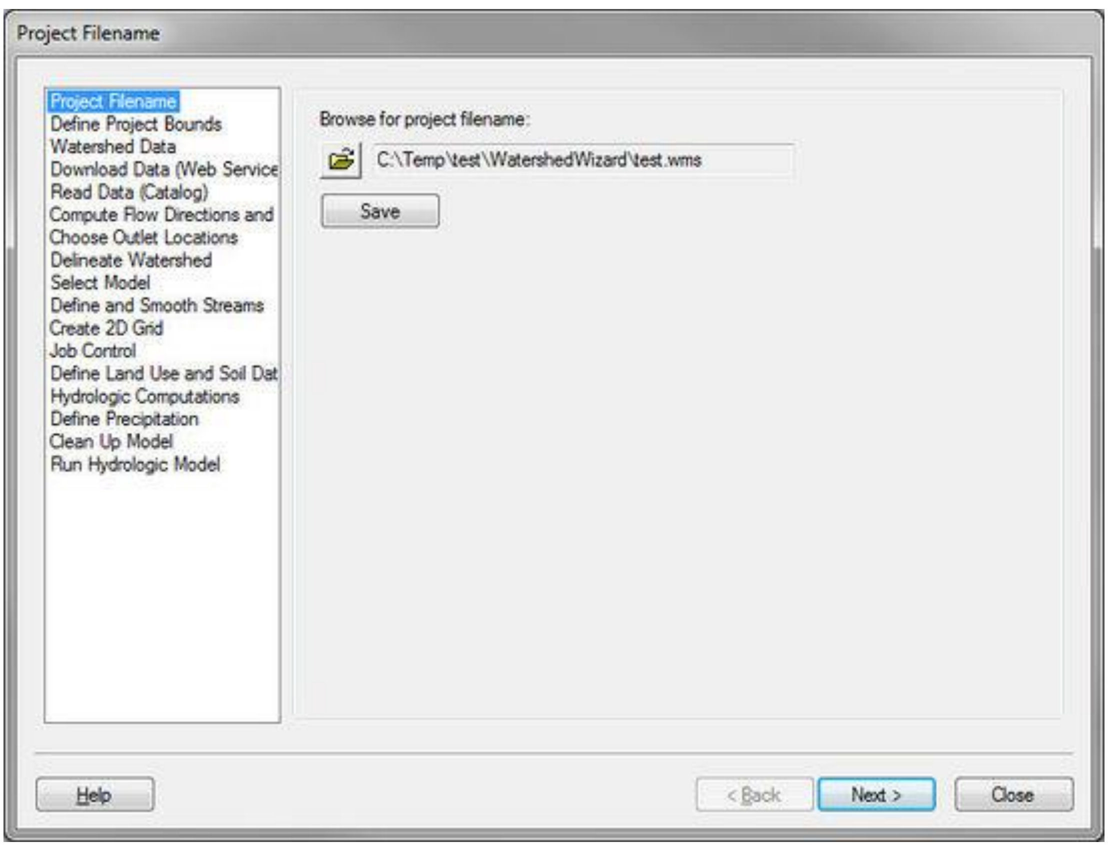
The following steps are included in the HY-8 modeling wizard:

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

General Information

If wanting 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

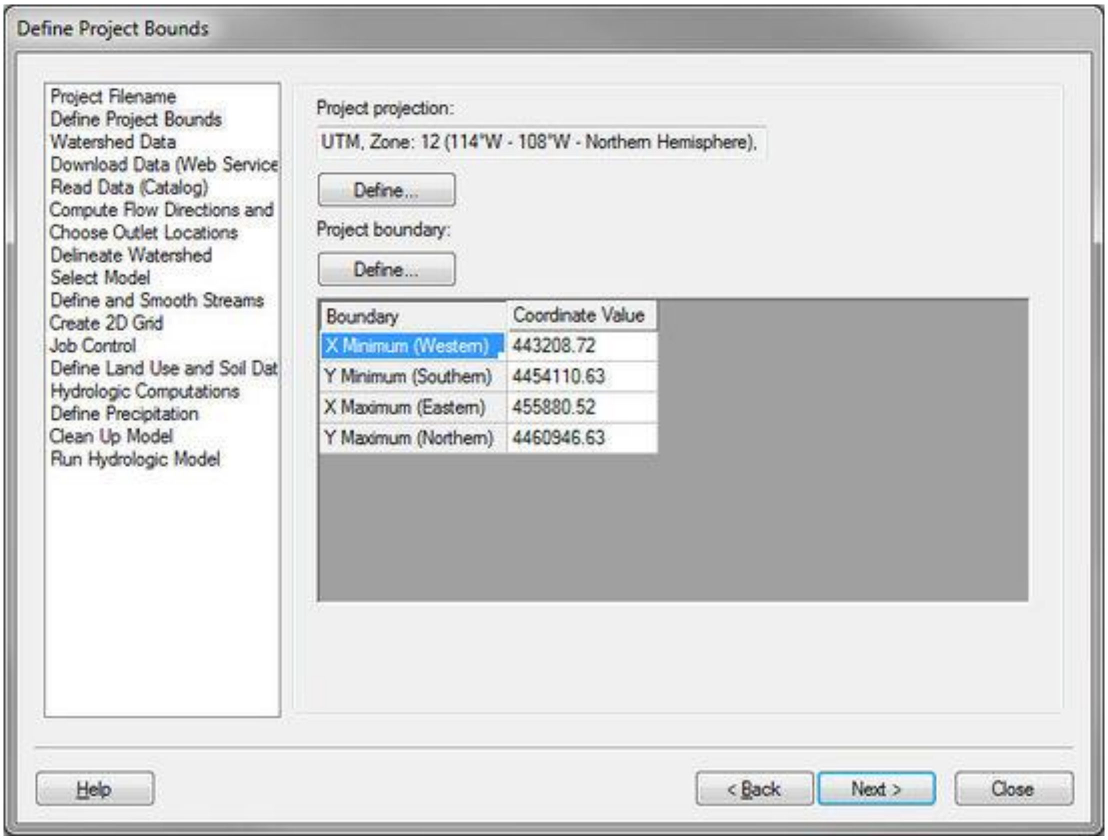


The Project Filename step is used for defining a project filename. This filename is used when saving files from the hydrologic modeling wizard.

Help

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

Define Project Bounds



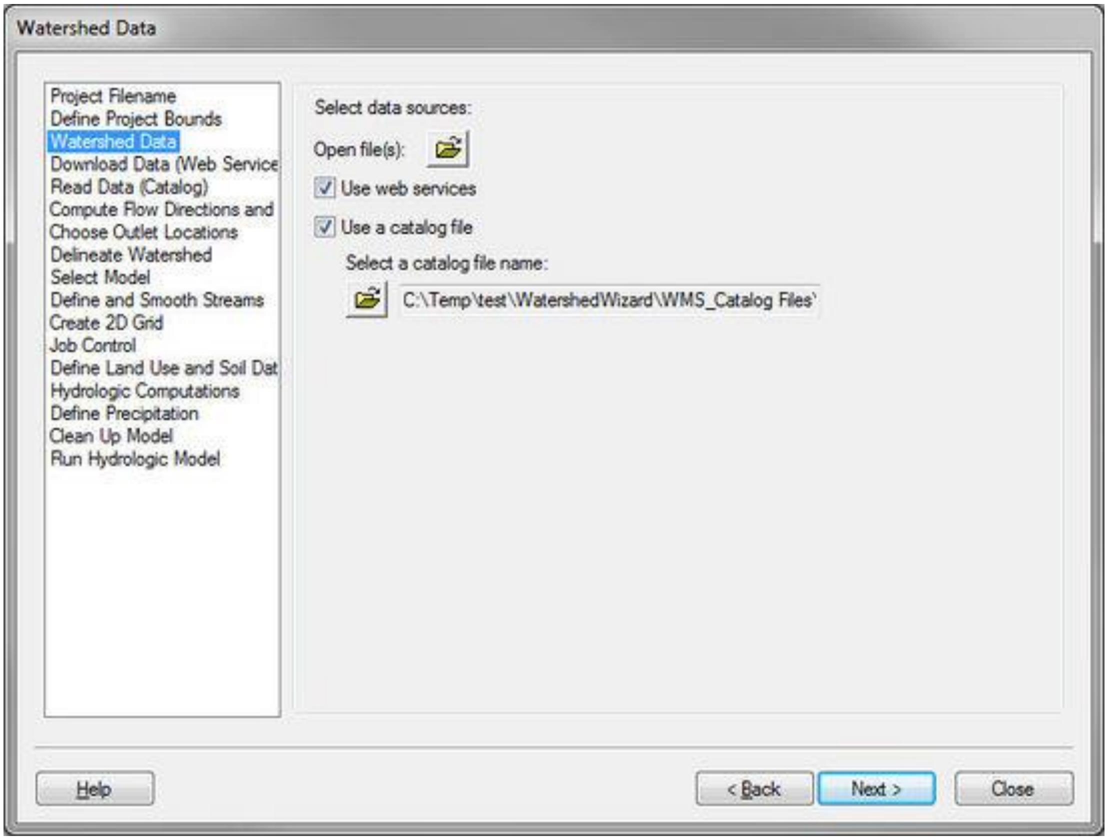
The define project bounds step is used for defining the project boundary.

Help

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

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

Watershed Data



The *Watershed Data* step is used for defining how to read data into WMS. Obtain data using the WMS web service client, a catalog file, or by simply opening files.

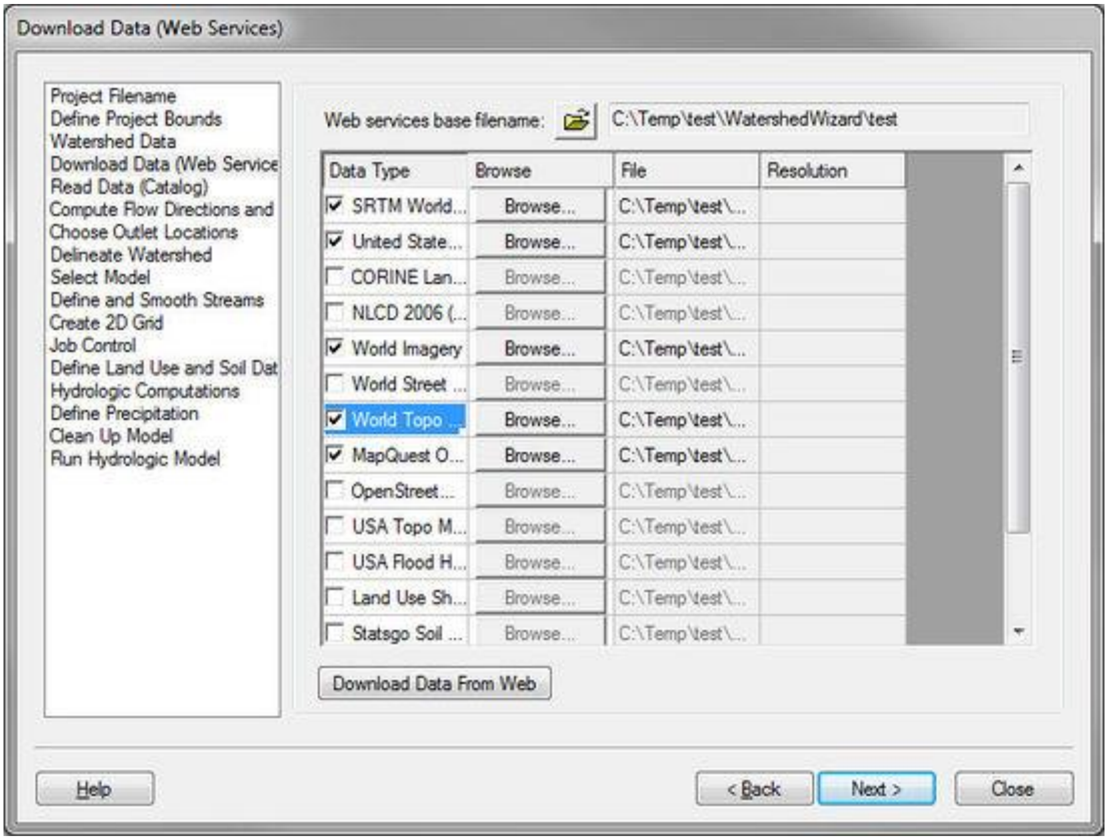
Help

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 selected.

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

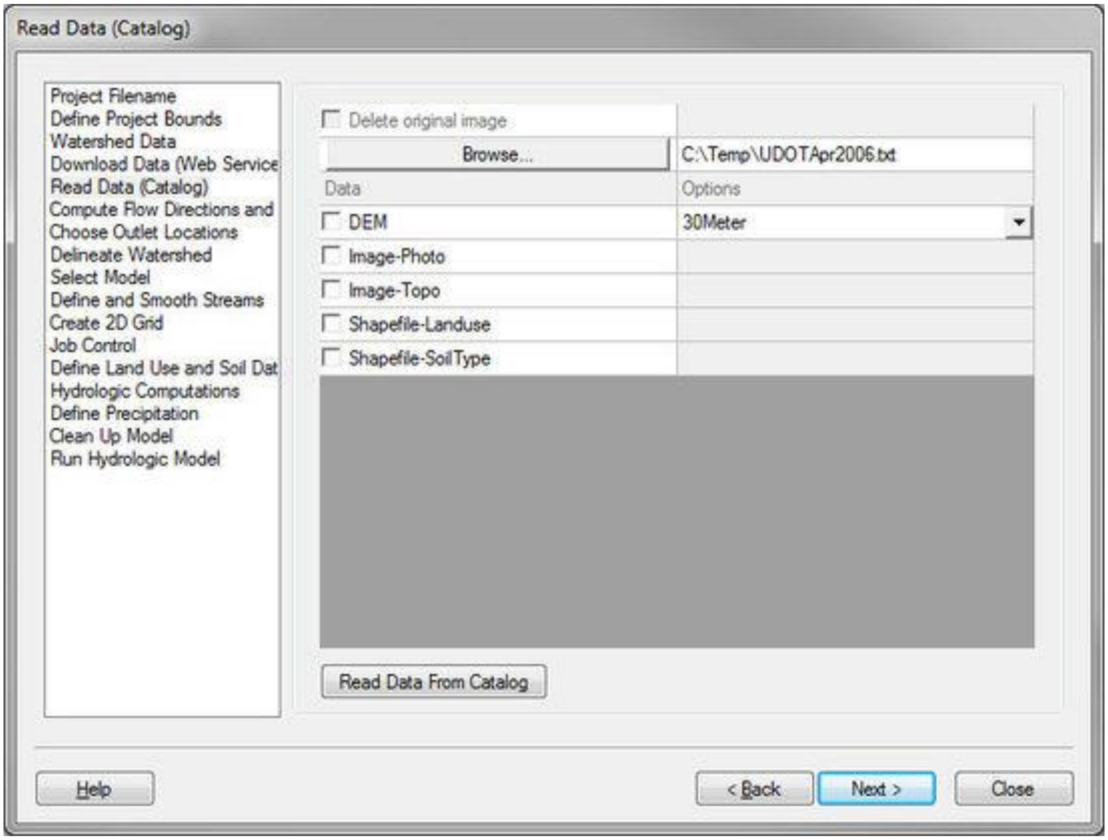


The Download Data (Web Service Client) step is used to download data over the internet using the WMS web service client if a web service is available. NED and SRTM data may be downloaded from this dialog, but coverage may not exist or download times may be slow for the hydrologic modeling area. The NED 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 the selected modeling area from the web service providers.

Read Data

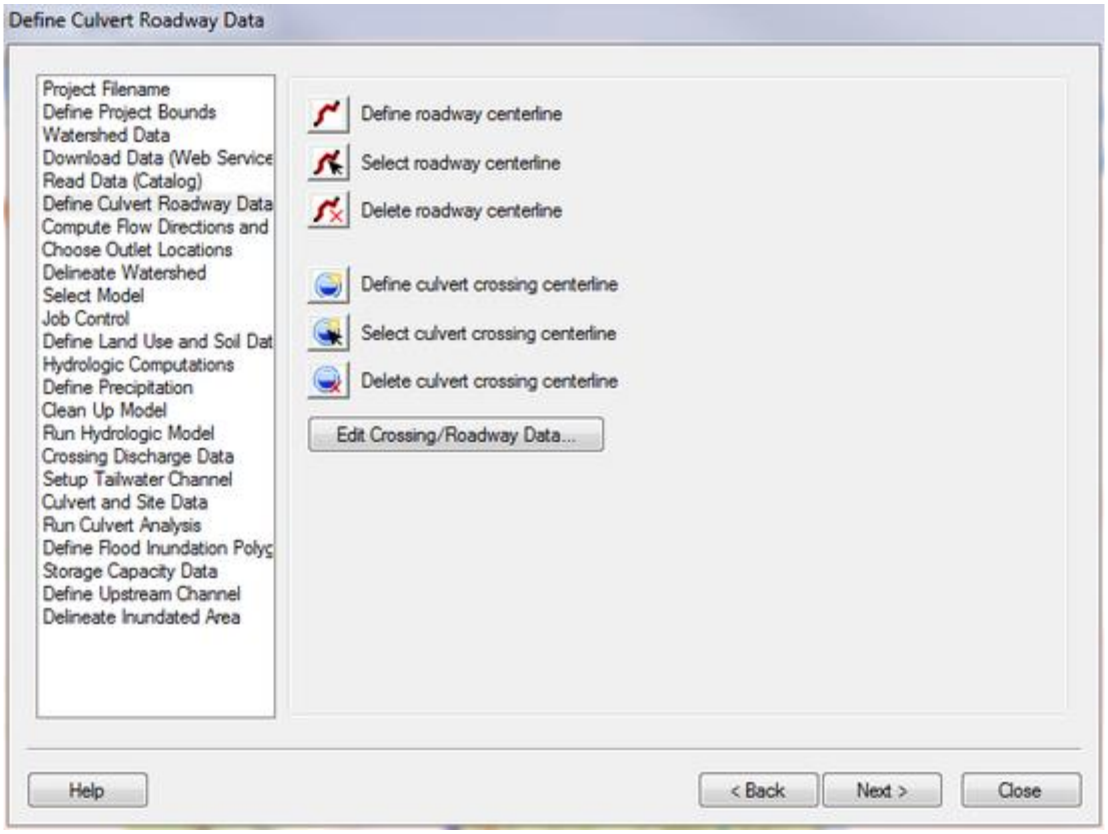


The Read Data (Catalog) from catalog step is used to select and read data from a WMS catalog.

Help

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

Define Culvert Roadway Data



The Define Culvert Roadway Data step is used to define the culvert and roadway centerlines as arcs and the data associated with these structures.

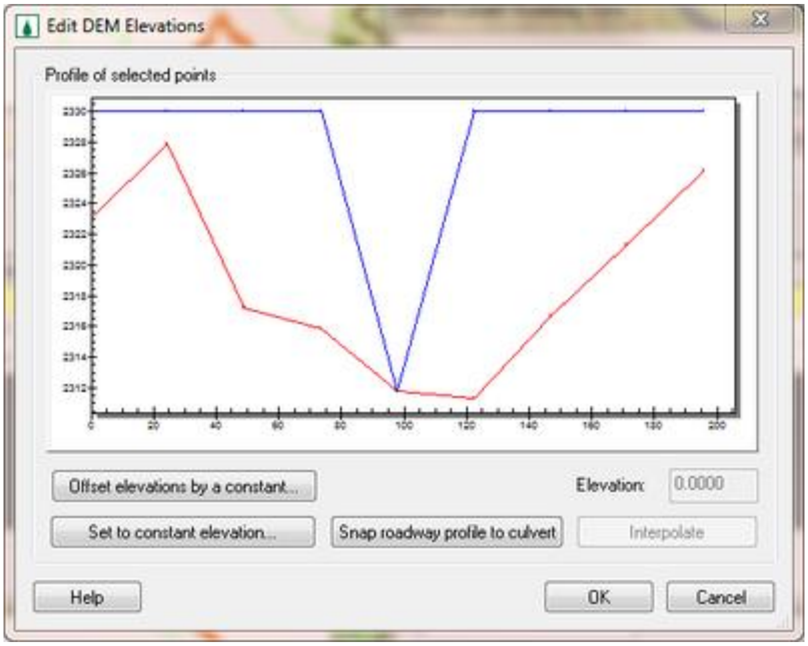
Help

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 clicking 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.

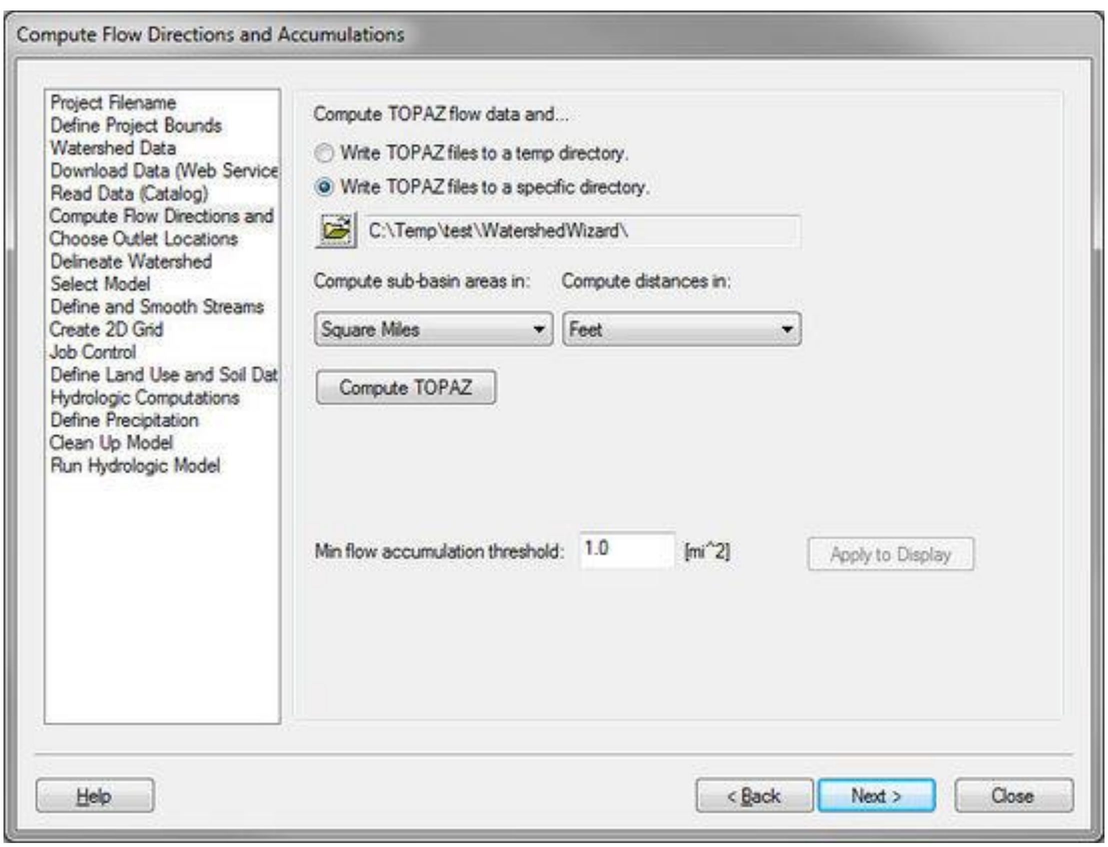
For the roadway attributes, a button exists that brings up the roadway elevation profile, as shown here:



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



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

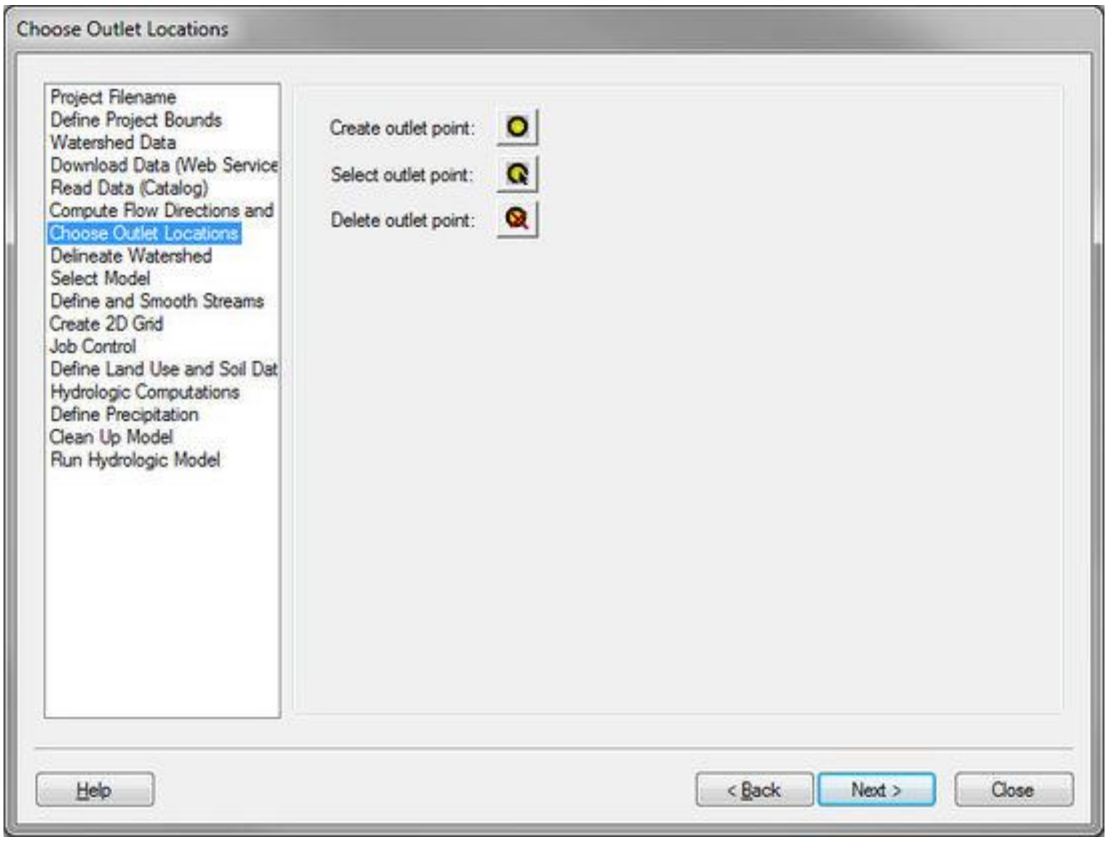
Help

Compute TOPAZ flow data and... – Choose either to write the TOPAZ output to a specific directory or to the temp directory. Most frequently, write the TOPAZ output to a temp directory.

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

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

Choose Outlet Locations



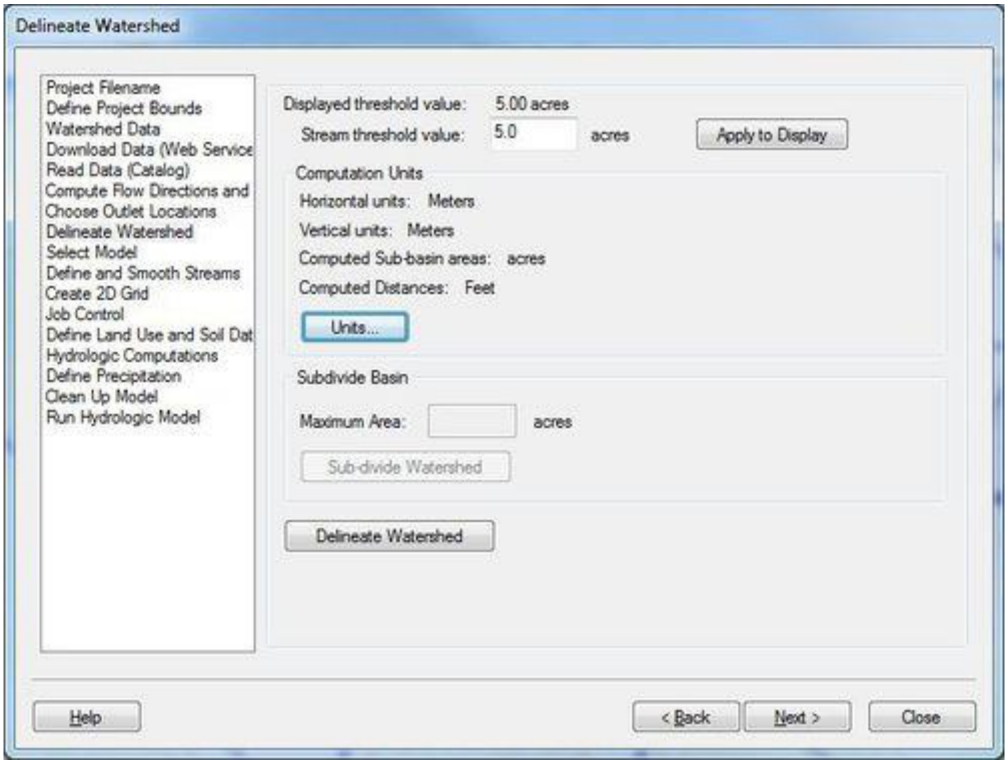
The Choose Outlet Locations step is used to define, move, or delete outlet locations.

Help

Create , **Select** , and **Delete** outlet point buttons – To create an outlet point, select the tool 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.

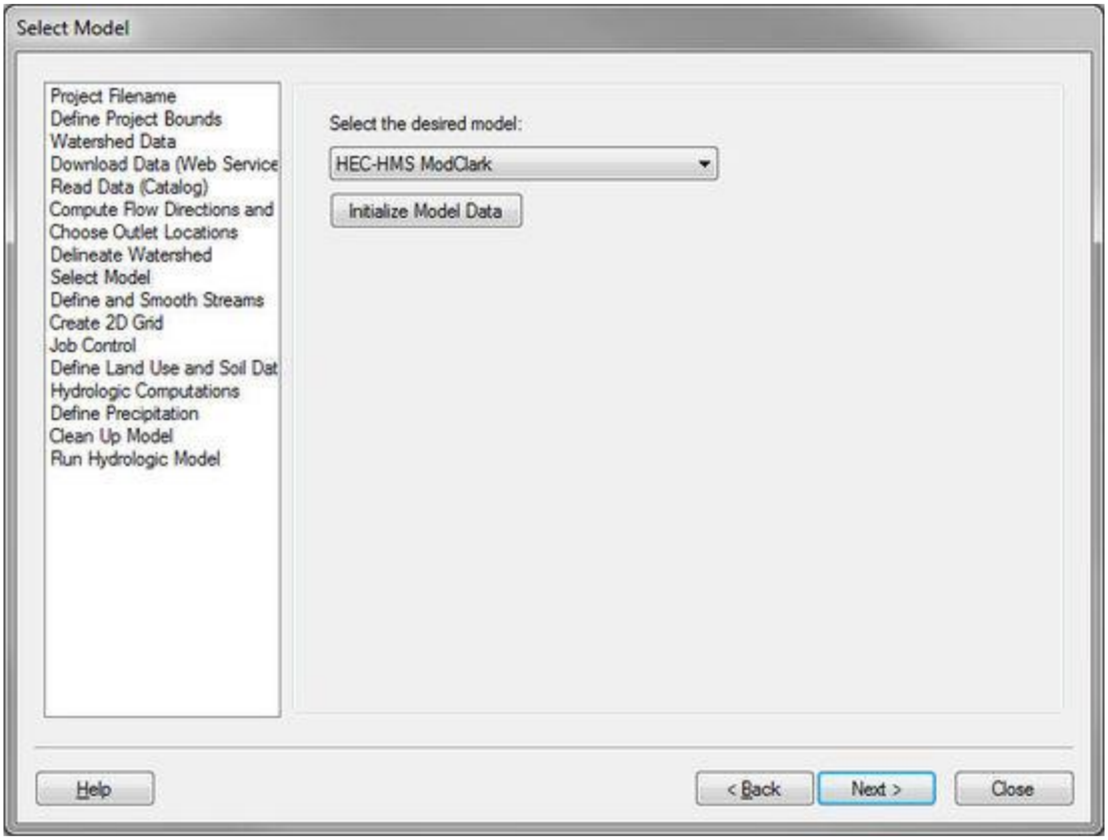
Units – This button is used to define the model and computation coordinates and units. Turn on the option to create a Tc coverage and change which data is computed by selecting the *Drain Data Compute Opts*... button in the *Units* dialog.

Sub-divide Watershed – This button subdivides the 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

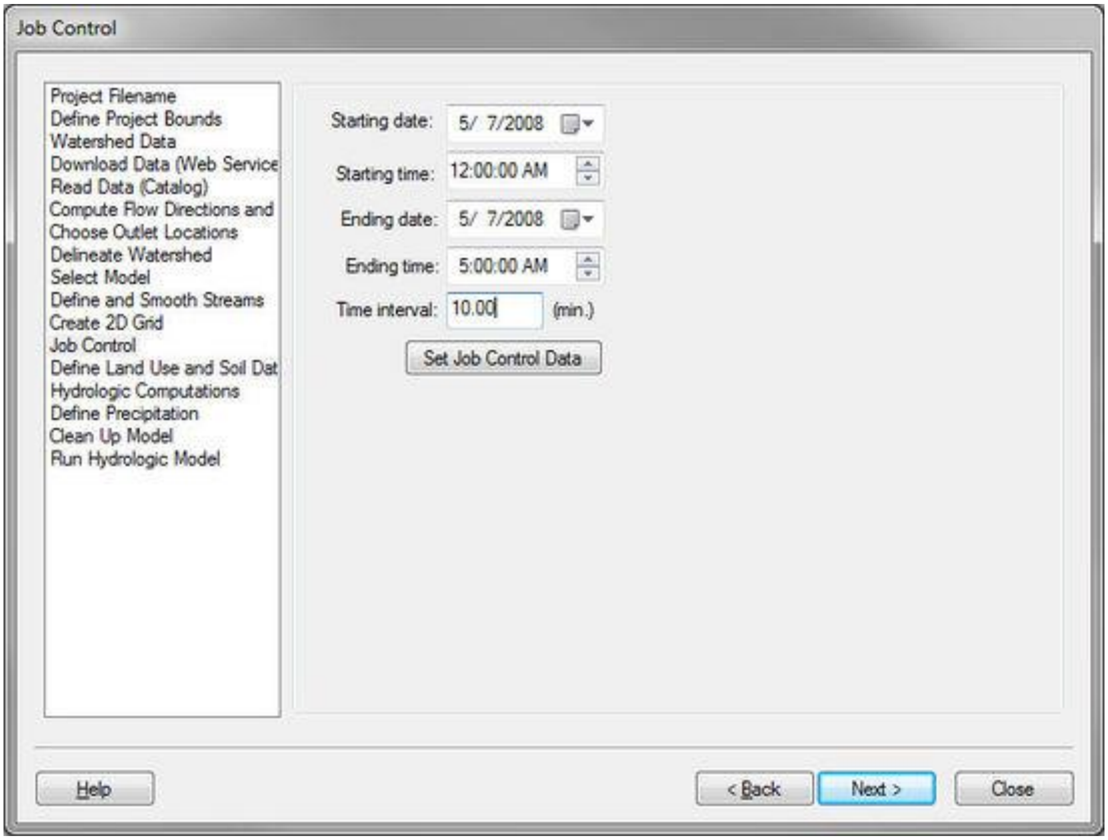


The Select Model step is used for defining which model is running with the delineated watershed.

Help

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

HMW Job Control

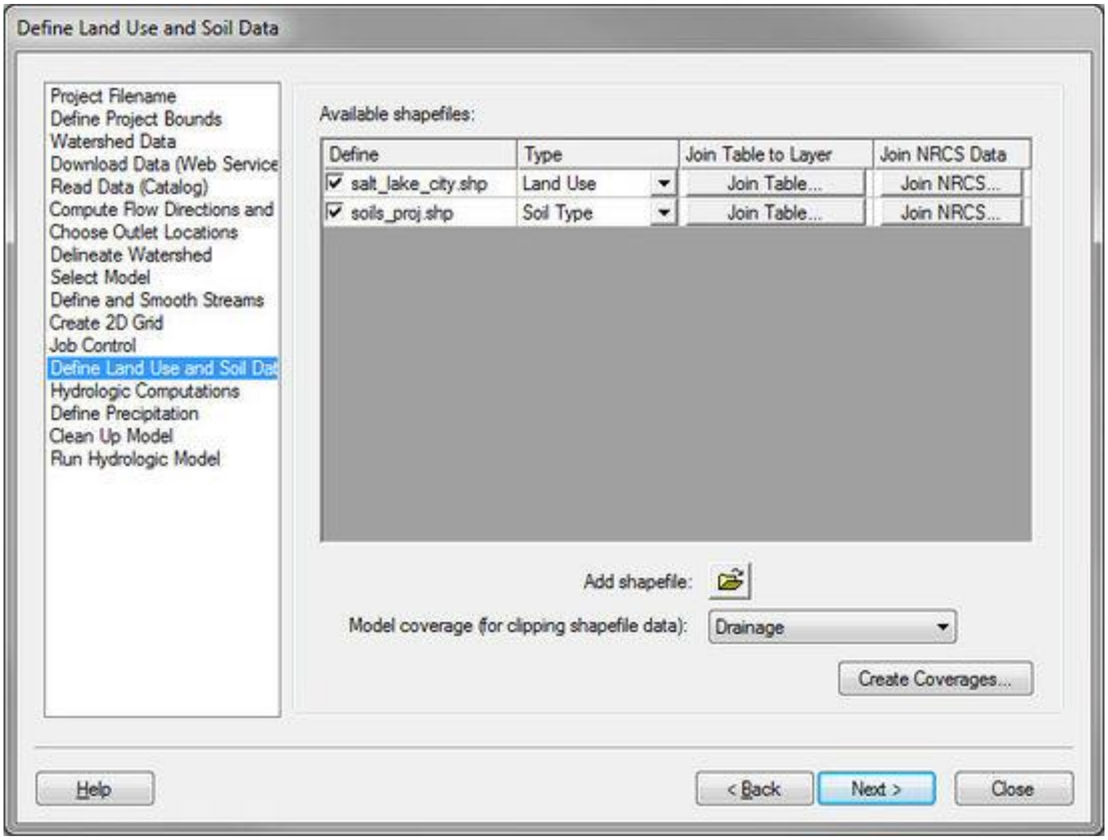


The Job Control step is used define the time parameters for running the model. Define the start and end time and date and select **Set Job Control Data** to set the job control parameters for the selected model.

Help

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

Define Land Use and Soil Data

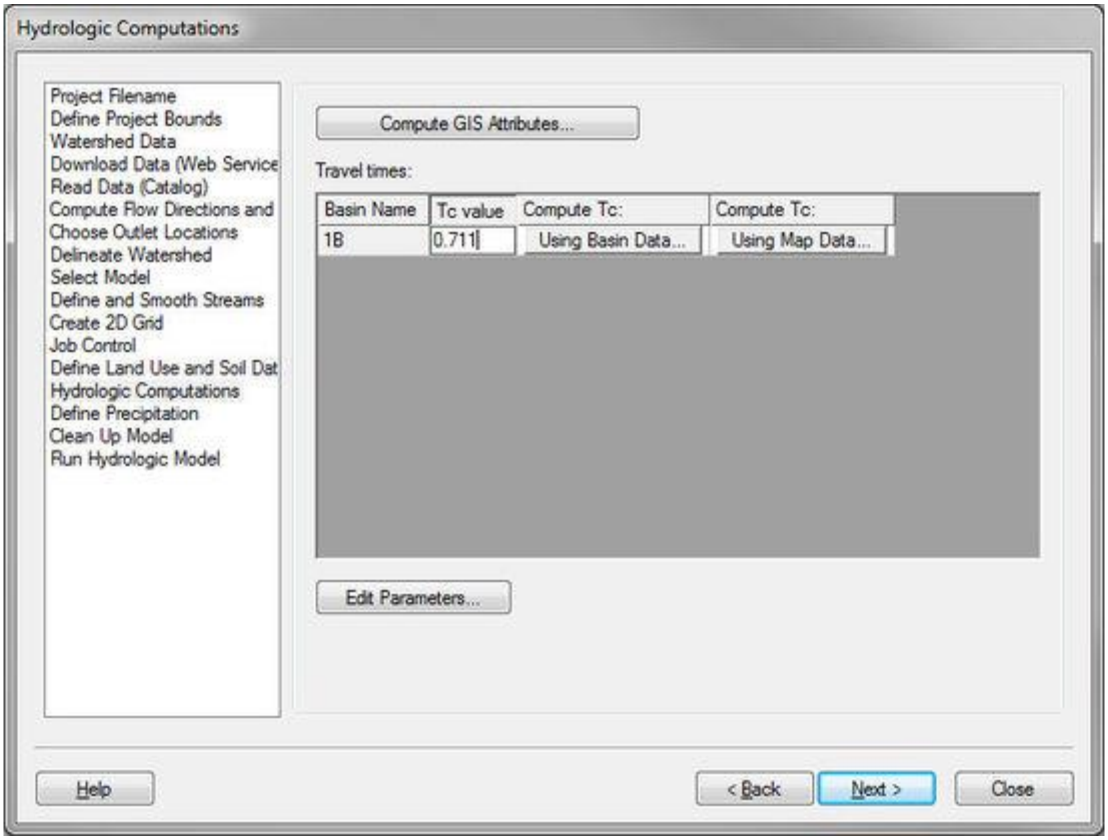


The Define Land Use and Soil Data step converts GIS Module shapefiles to data in the WMS Map module. WMS uses the boundary of the watershed to clip the shapefile data for the selected files.

Help

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

Hydrologic Computations



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

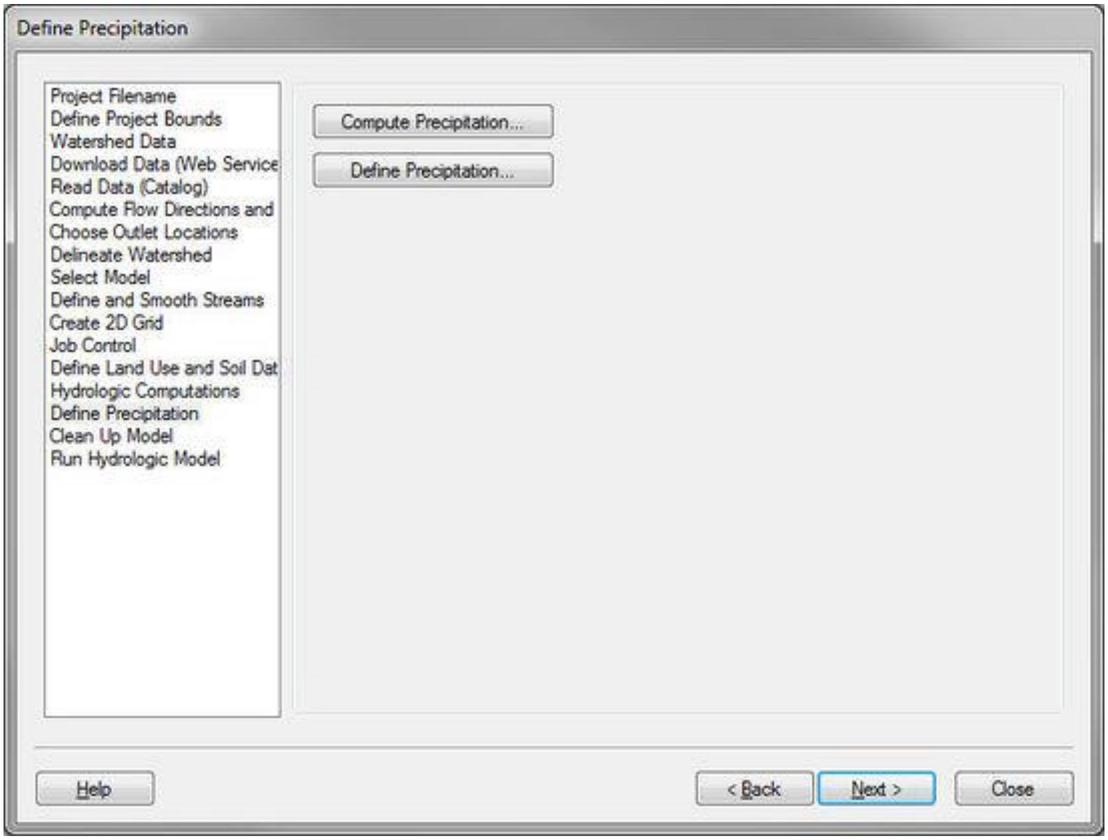
Help

Compute GIS Attributes – Depending on the model being used, this button brings up a dialog where attributes are computed from the land use and/or soil type data.

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

Edit Parameters – This button brings up the edit parameters dialog for the model where modeling parameters are changed for all the sub-basins in the watershed.

Define Precipitation



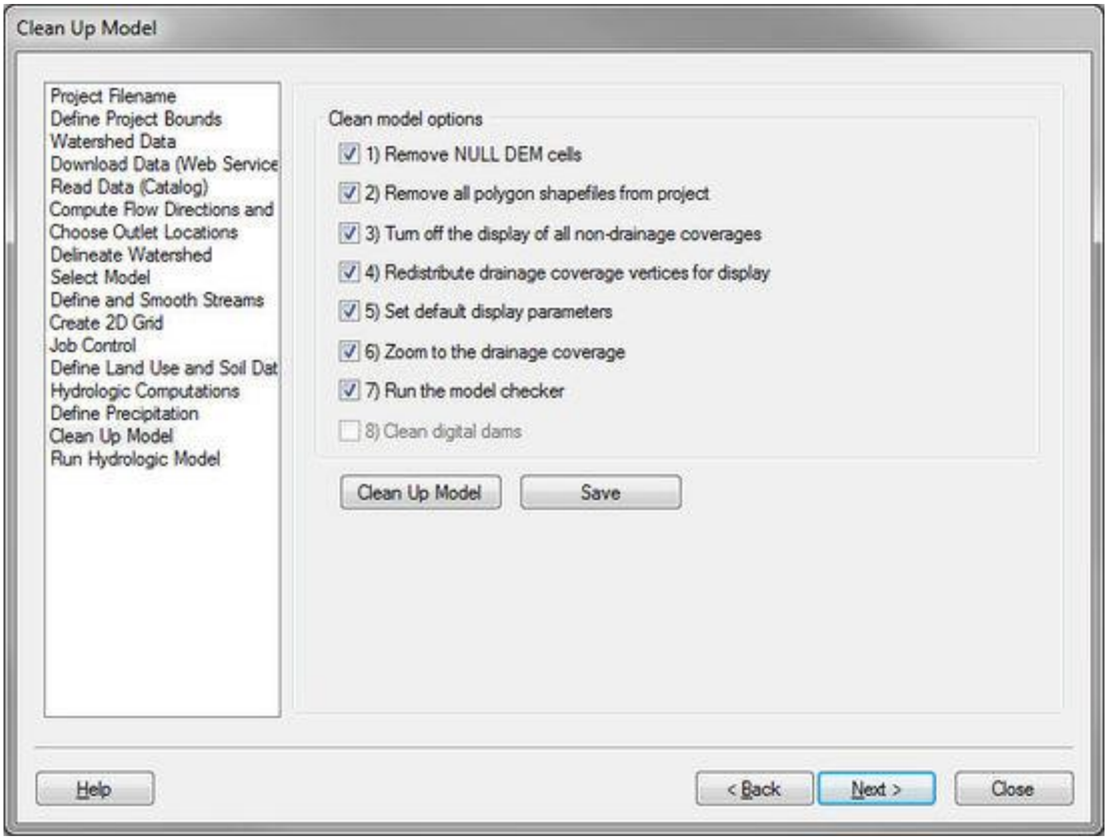
The Define Precipitation step is used to compute and define precipitation for the model.

Help

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

Define Precipitation – This button allows defining precipitation for the selected model.

Clean Up Model



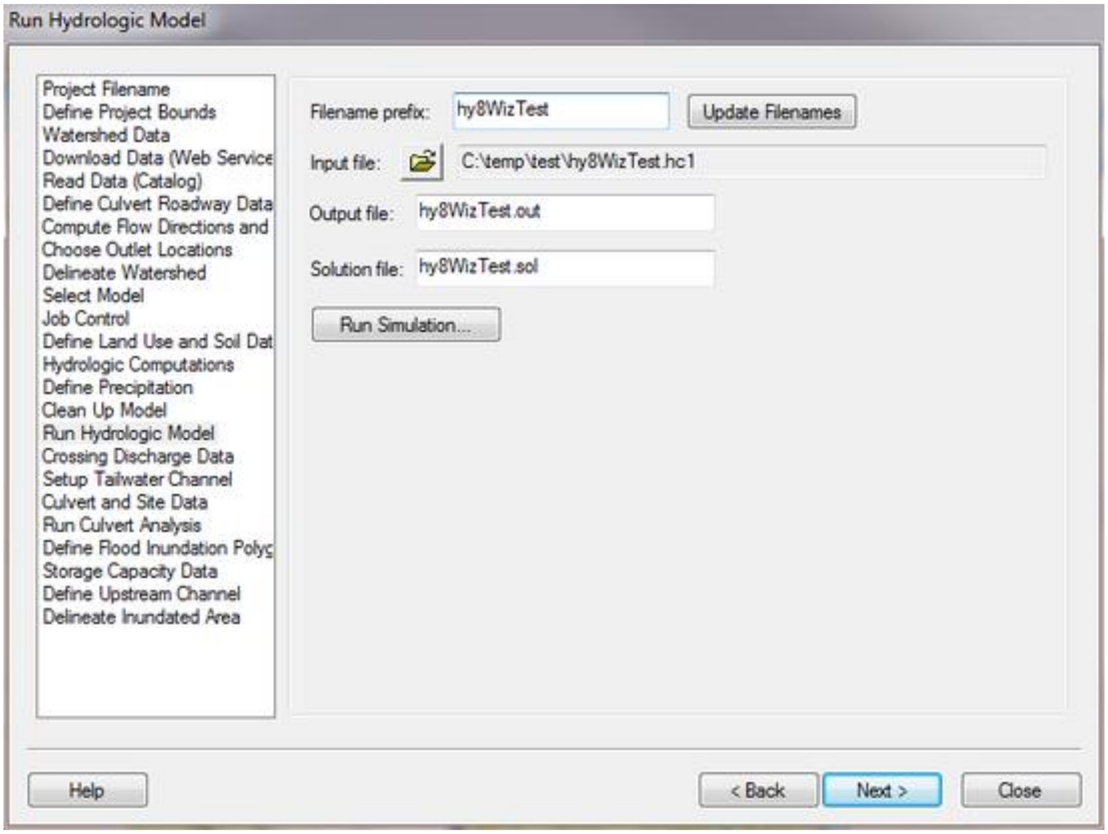
The Clean Up Model step is used to clean up the model by doing tasks that are typically done when the model is finished.

Help

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

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

Run Hydrologic Model

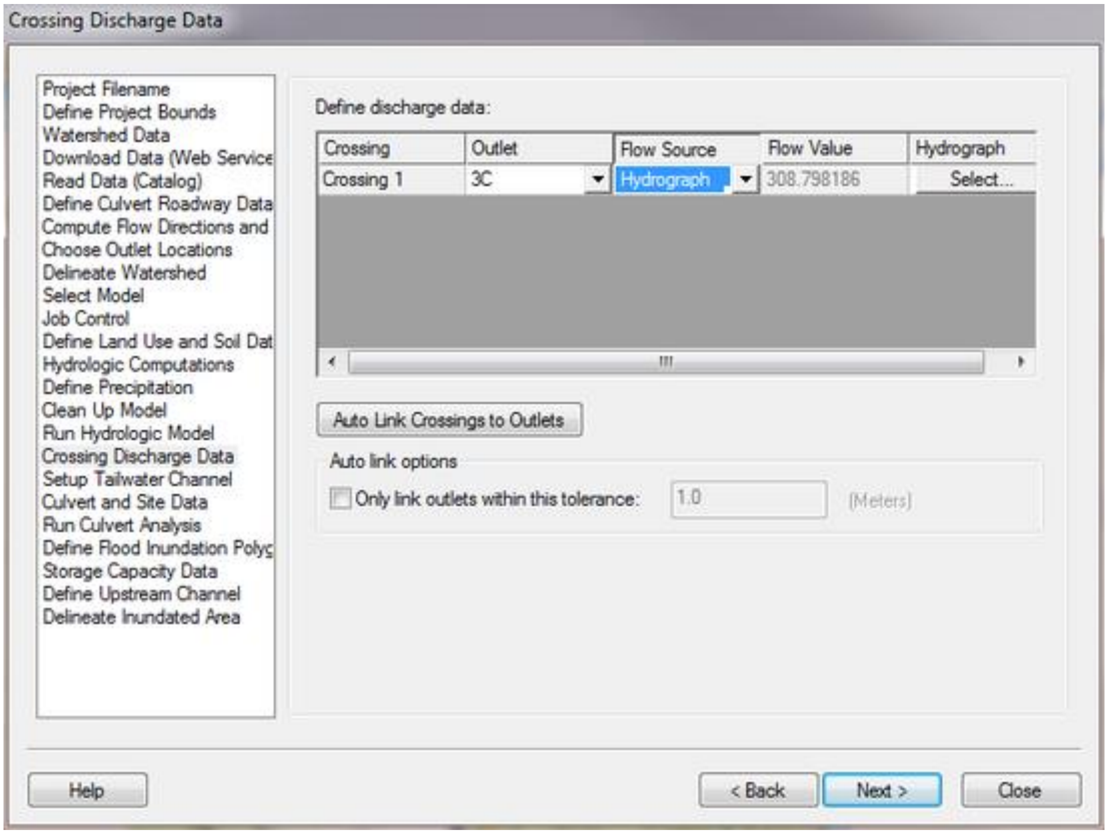


The Run Hydrologic Model step is used to run the hydrologic model and read the resulting hydrograph.

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



The Crossing Discharge Data step allows pulling a hydrograph from the hydrologic simulation, use the peak discharge from the hydrograph, or enter a hydrograph or peak discharge.

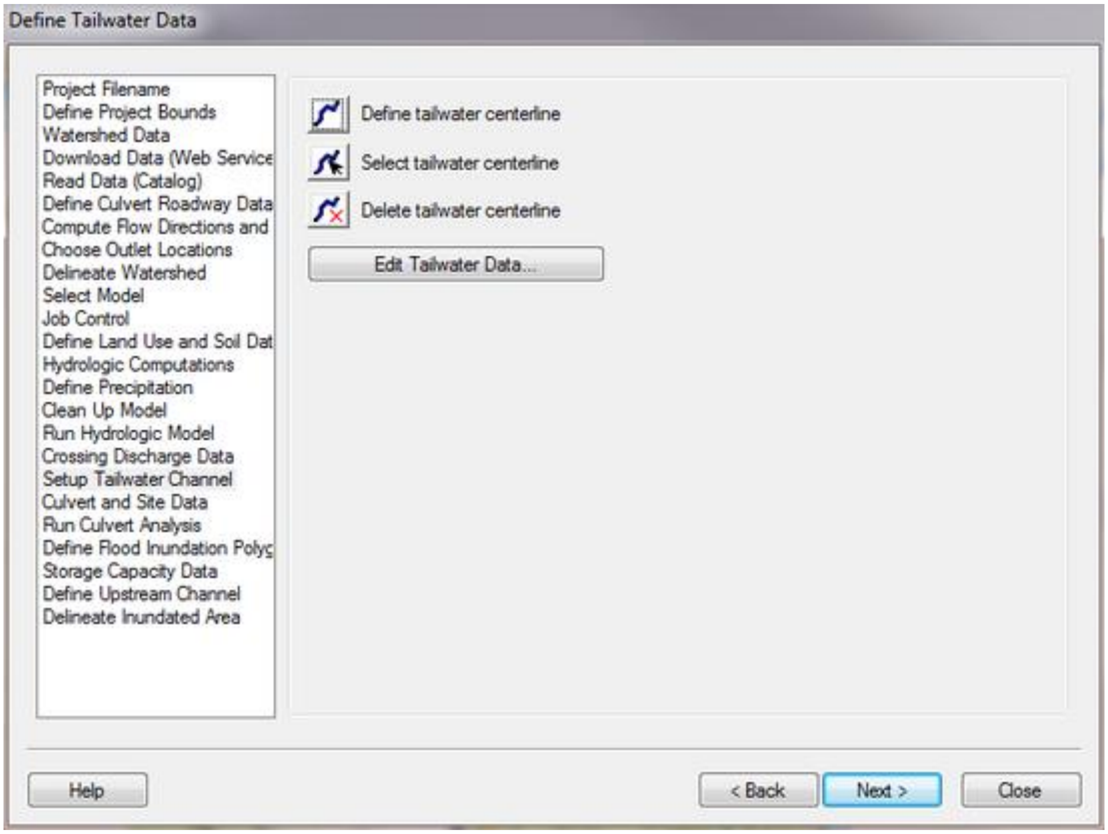
Help

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



The Setup Tailwater Channel step is used to allow defining 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. 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.

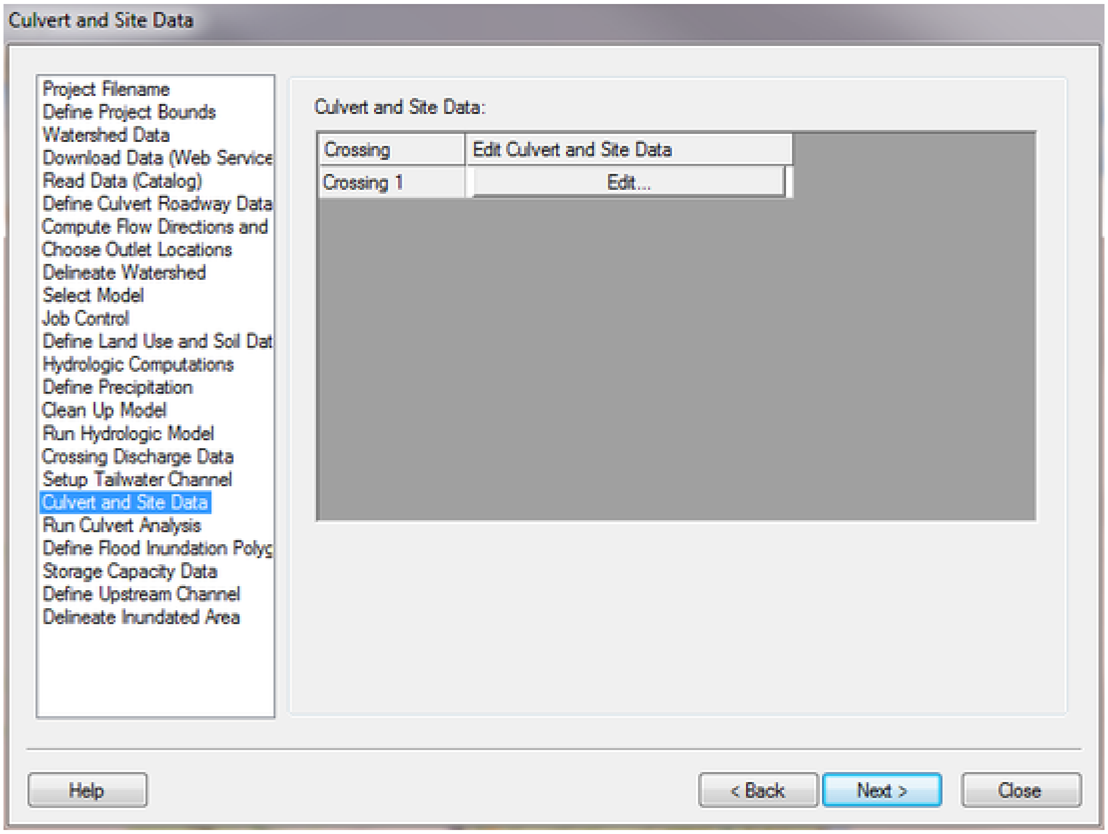
If no tailwater arcs are selected, clicking on the **Edit Tailwater Data...** button selects all the tailwater arcs in the HY8 coverage and brings up the arc attributes for these arcs. If tailwater arcs are selected, only the selected tailwater arc attributes are shown. These attributes are shown in 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. Enter values for other parameters such as bottom width, side slopes, and roughness.

If the irregular channel type is selected, a button that allows entering 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



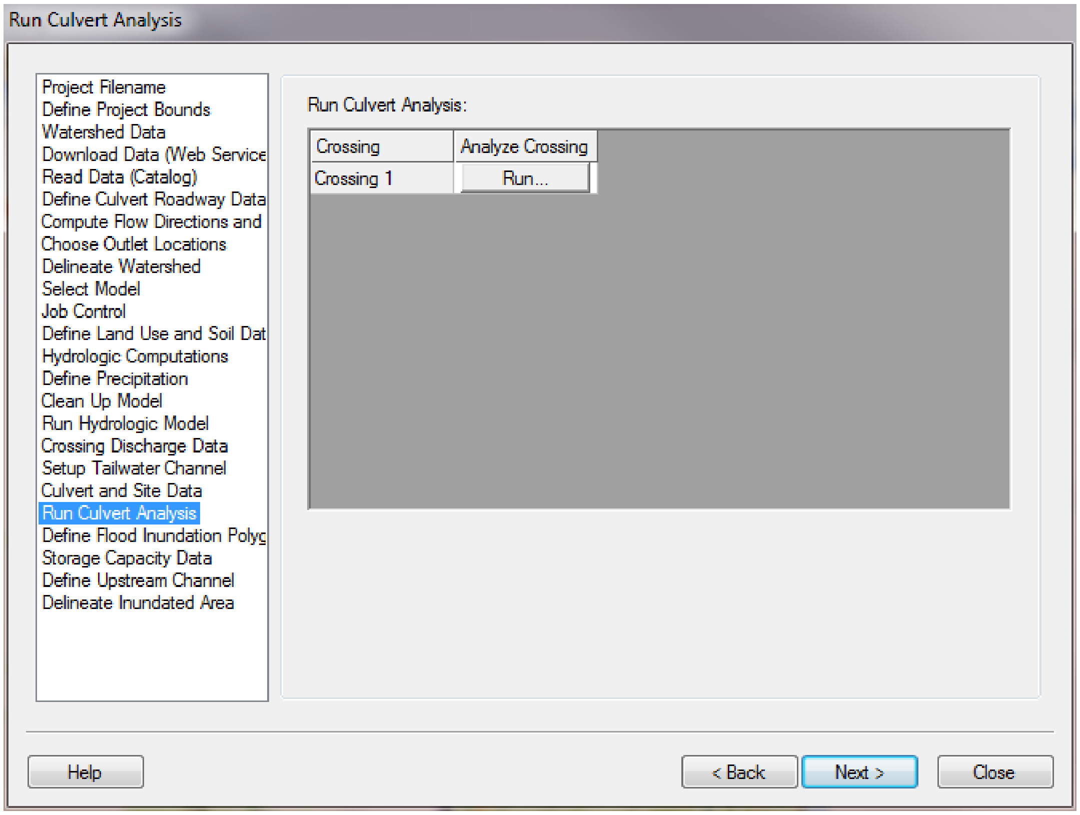
The Culvert and Site Data step is used to finish setting up the culvert data by defining the data in the *HY-8 Crossing Data* dialog. This step shows a spreadsheet with a listing of each culvert crossing. A user can edit the culvert and site data associated with each crossing by clicking on the **Edit** 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

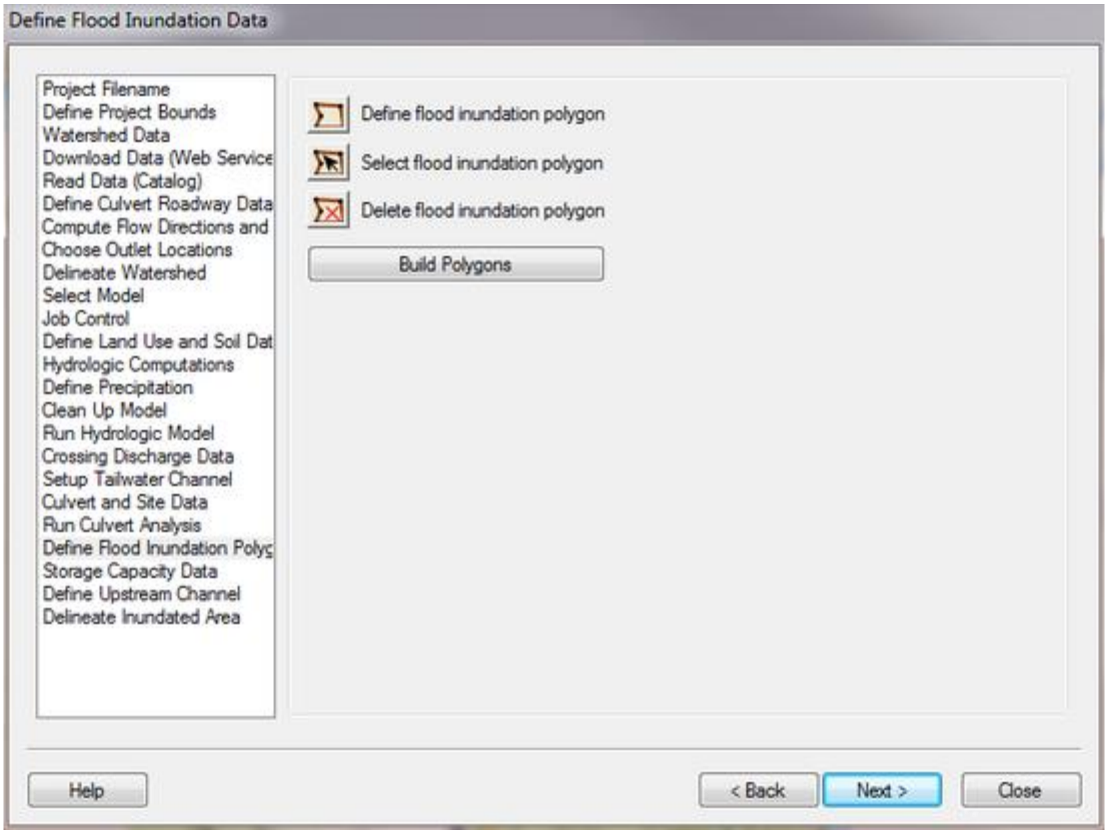


The Run Culvert Analysis step is used to run the culvert crossing and view the results of the culvert analysis.

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

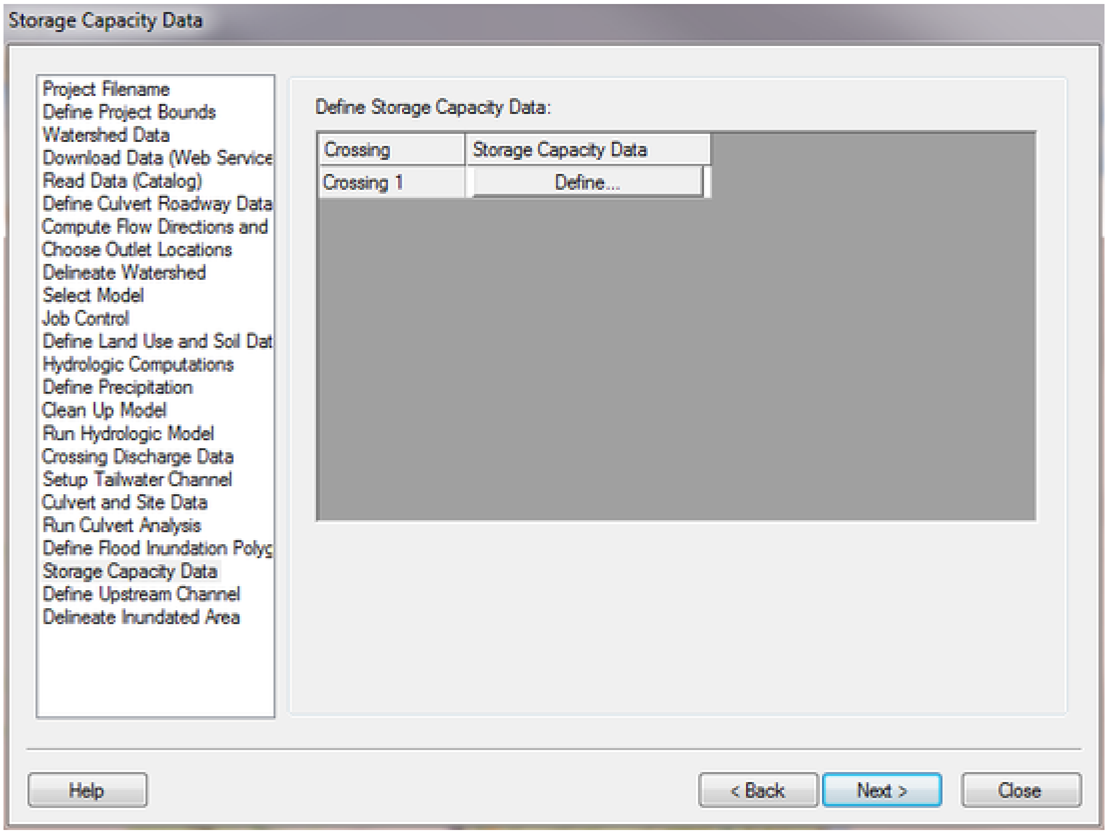


The Define Flood Inundation Polygon step is used to define a flood inundation polygon if a drainage coverage with a polygon defining the inundation area does not already exist. It's necessary to define a flood inundation polygon for any culvert crossings with no outlets (and no basin polygons) assigned 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 creating arcs that represent the flood inundation boundary. Normally, use the roadway as one of the arcs and build arcs for the rest of the inundation boundary. If selecting **delete flood inundation polygon** then clicking 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 to use all the arcs to build polygons.

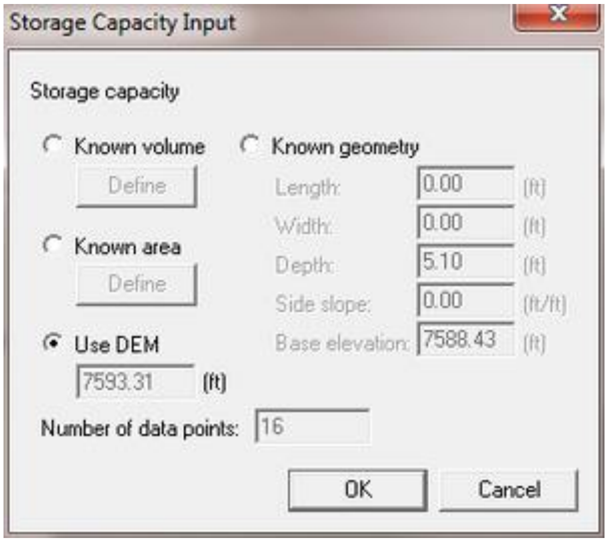
Storage Capacity Data



The Storage Capacity Data step is used to define storage capacity data and to route the hydrograph at the outlet through the culvert.

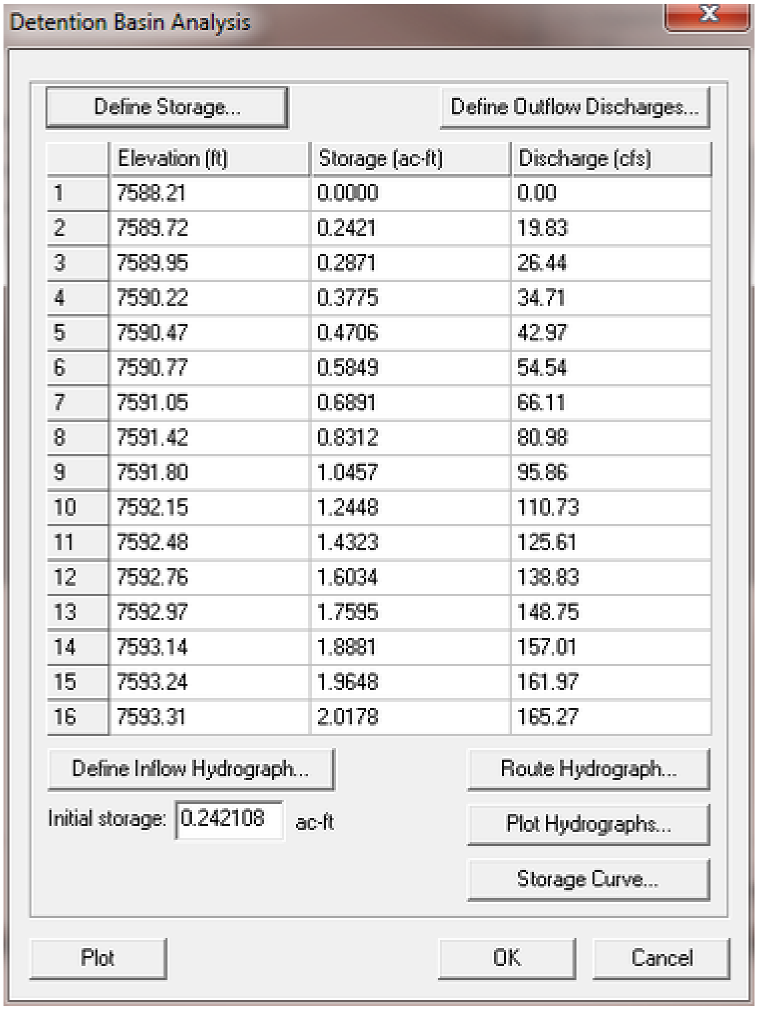
Help

This step in the wizard lists all the crossings in the HY-8 coverage and allows defining 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:



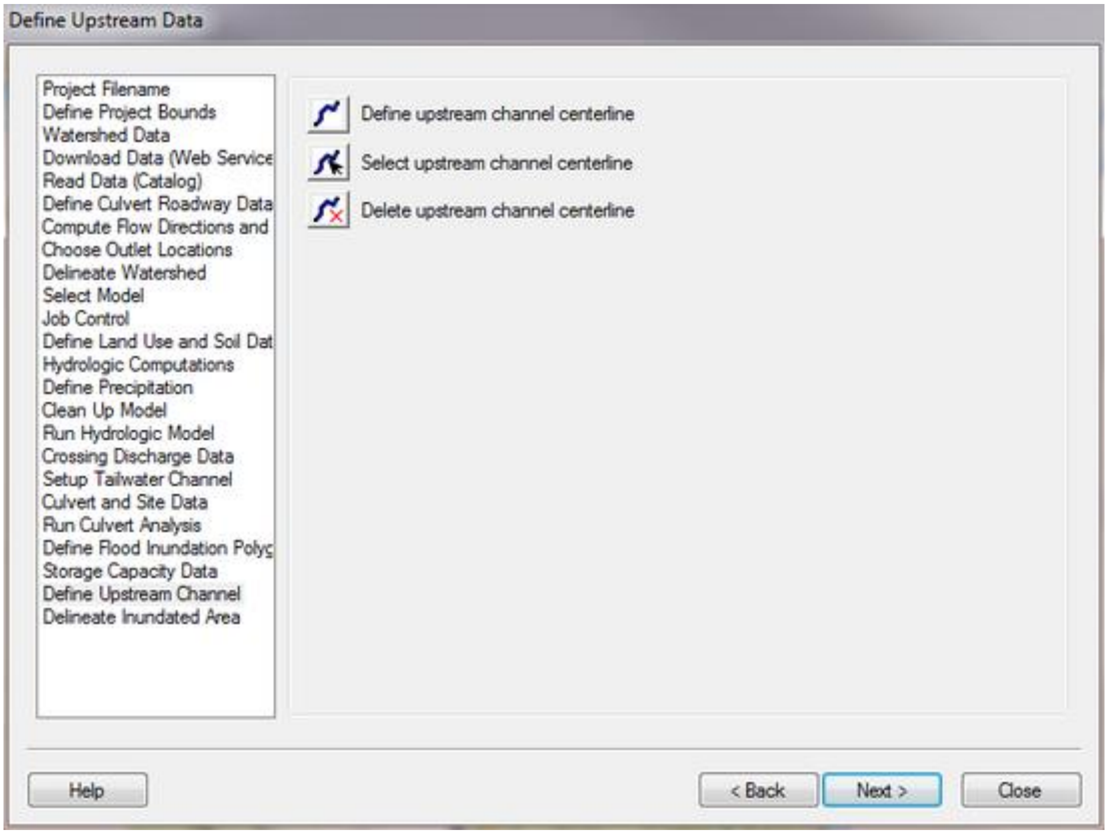
In this dialog, select the option to use the DEM (this is the default if a DEM exists) to compute the volumes at each water surface elevation. Alternatively, enter the geometry of the basin upstream of the culvert or an elevation-area curve. If the *Known Area* option is selected, manually enter the 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 viewing and editing the routed hydrograph and its associated data:



	Elevation (ft)	Storage (ac-ft)	Discharge (cfs)
1	7588.21	0.0000	0.00
2	7589.72	0.2421	19.83
3	7589.95	0.2871	26.44
4	7590.22	0.3775	34.71
5	7590.47	0.4706	42.97
6	7590.77	0.5849	54.54
7	7591.05	0.6891	66.11
8	7591.42	0.8312	80.98
9	7591.80	1.0457	95.86
10	7592.15	1.2448	110.73
11	7592.48	1.4323	125.61
12	7592.76	1.6034	138.83
13	7592.97	1.7595	148.75
14	7593.14	1.8881	157.01
15	7593.24	1.9648	161.97
16	7593.31	2.0178	165.27

Define Upstream Channel

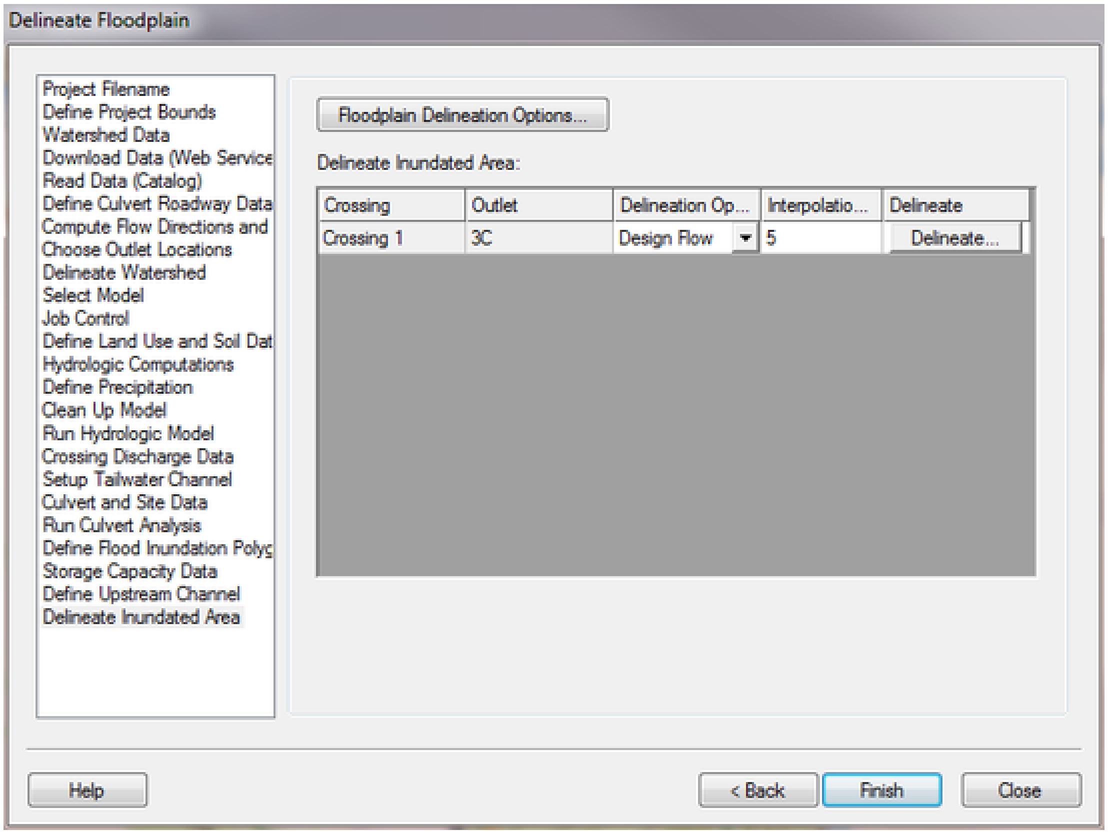


The Define Upstream Channel step is used to define an arc representing the upstream channel.

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



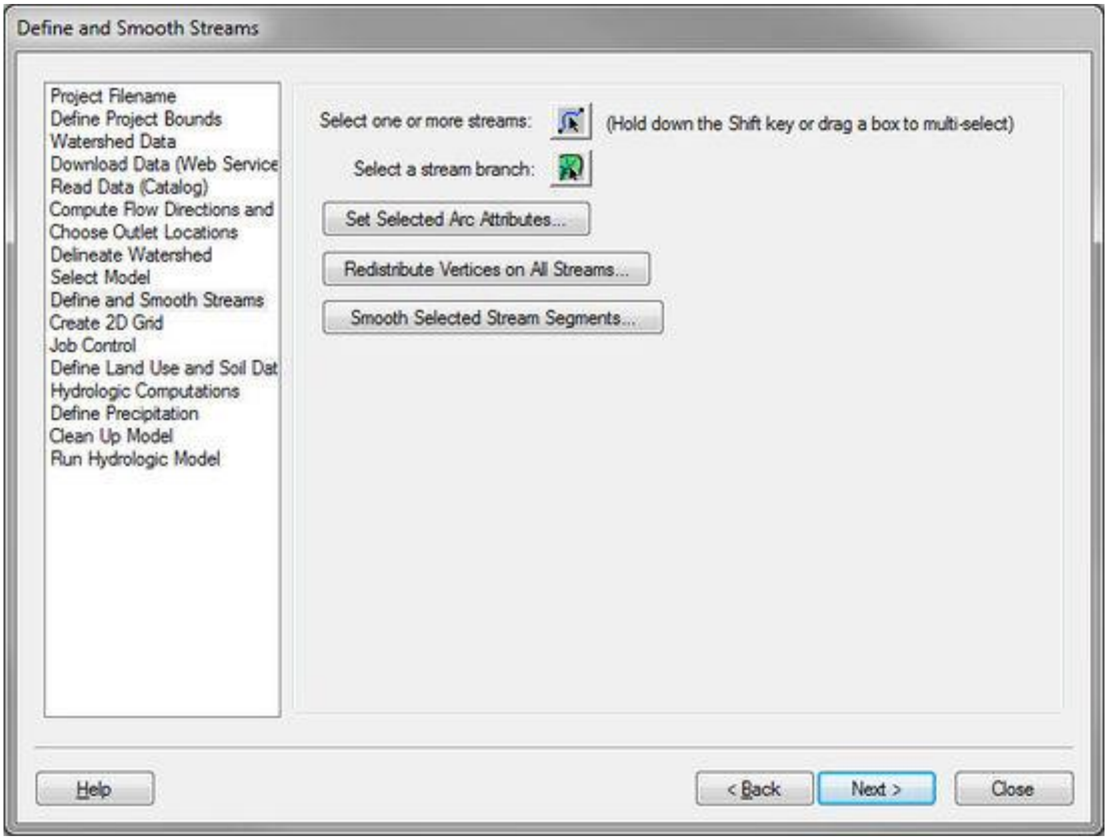
The Delineate Inundated Area step is used to delineate the floodplain corresponding to the culvert's design flow or hydrograph at each culvert crossing.

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 it is possible to edit this interpolation interval. When the **Delineate** button is pressed, WMS creates a new 2D scattered dataset 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



When building a GSSHA model, define the parameters for and smooth all the streams in the 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, 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 setting the attributes for all the selected arcs.

Redistribute Vertices on All Streams – Selecting this button will first switch the 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 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, 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.

8. Importing/Exporting Data

File 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 *File Import Wizard* .

The *File Import Wizard* enables importing many different types of data into WMS. The *File 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

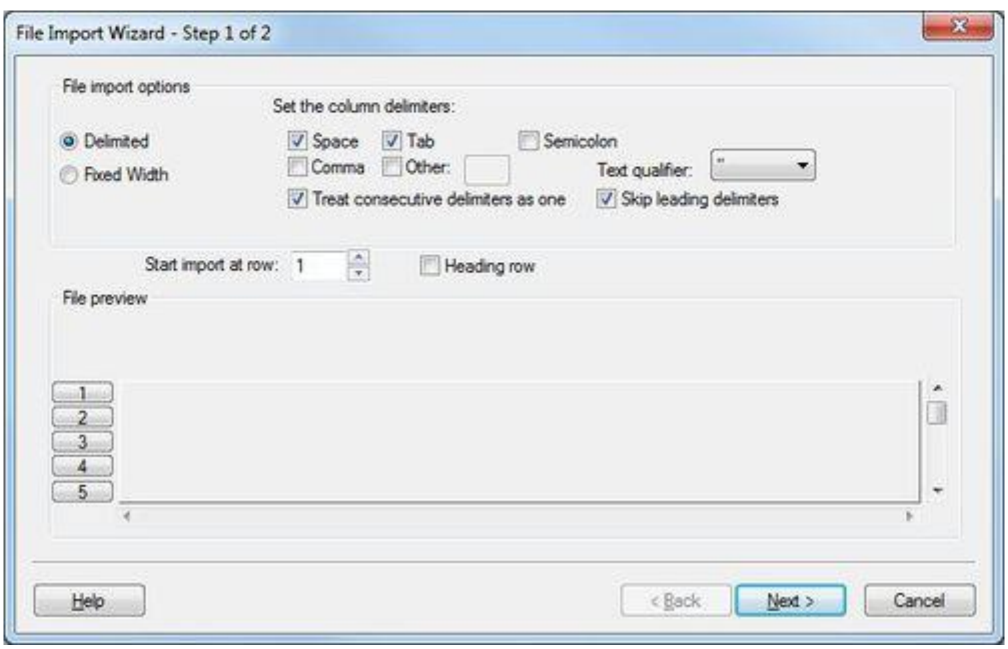
Step 1 – File Outline

The first step in the wizard allows delimiting the data into columns. Two options exist to delimit the data.

•Delimited

•Fixed Width

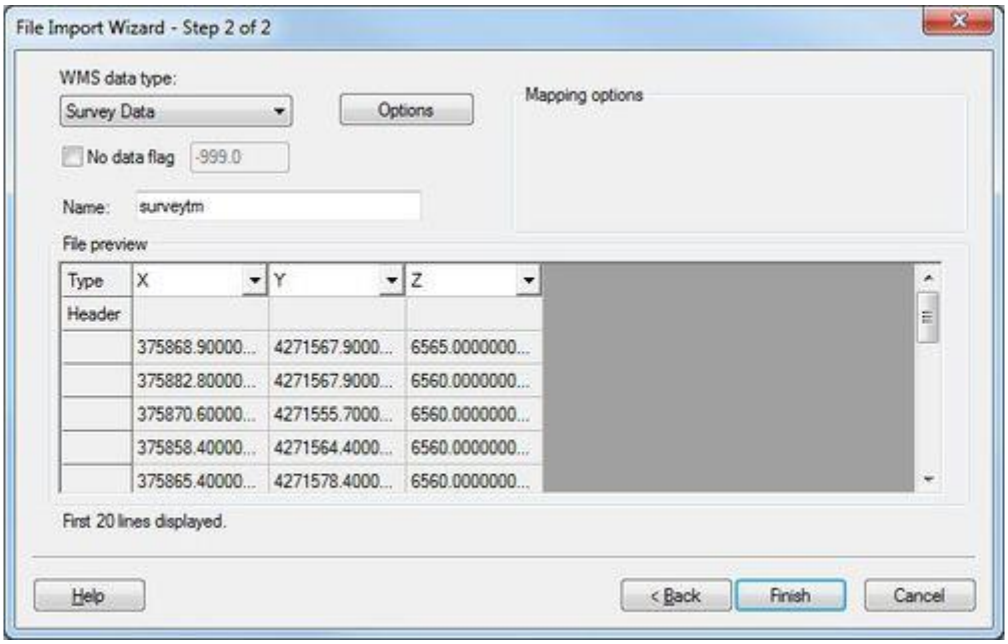
For the *Delimited* option, typical delimiters are included as well as an option to specify a delimiter. Columns can also be specified with a fixed width by clicking on the ruler bar or the window with the data. Break lines can be dragged, and they can be deleted by double-clicking on the break line or dragging them off the screen. Specify the starting row the data will be imported. If the data has a row of headings, indicate such and WMS will use the headings in the next step to determine what kind of data each column represents.



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 allows picking what kind of data to import (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 progressing to the next step in the wizard. The name of each column is changed by editing the Header cell.



File Import Filter Options

In Step 2 of the *File Import Wizard* dialog, when selecting "Survey Data", "TIN Vertices", or "Catalog" *WMS data types* an **Options** button becomes active. Selecting this button will bring up the *File Import Filter Options* dialog.

- Filter Type
    - nth Point
    - Area
    - Grid

Supported Formats

The following types of data can be imported into WMS via the *File 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	
Dataset	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

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 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 Dataset (NED). The default at this site is Arc/Info grid, but modify the 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 creating the outlets, streams, and basins themes, reorder streams, rename attributes, and then export to a WMS/ArcView super file which will allow opening 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, 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 importing 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.

Import and Export GIS Files

The **Import GIS File** and **Export GIS File** commands in the *HEC-RAS* menu can do the following:

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, a prompt will ask for a file name to save the cross section database to.

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 the project in HEC-RAS
2. Select *File* | **Export GIS Data...** . Select the data 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 the GIS data was exported in step 2 (it is saved to be in the same folder as the 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
- Managing Cross Sections

8.1 File Support

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 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 Dataset Files
- Binary Dataset 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 *File 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. For more information see the article 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. For more details see the article 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	*.shp	

Shapefile		
ArcView®/ WMS Superfile	*.sup	Besides the tools in WMS for editing data is imported from shapefiles, a special extension for ArcView® 3.3 and earlier has been created which allows creating the outlets, streams, and basins themes, reorder streams, rename attributes, and then export to a WMS/ArcView® super file which will allow opening all three themes by importing the super file. This extension may be available on the WMS website and has a separate document describing its usage. In order to be activated in ArcView®, 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 it's necessary 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 specifying 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 WMS 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	

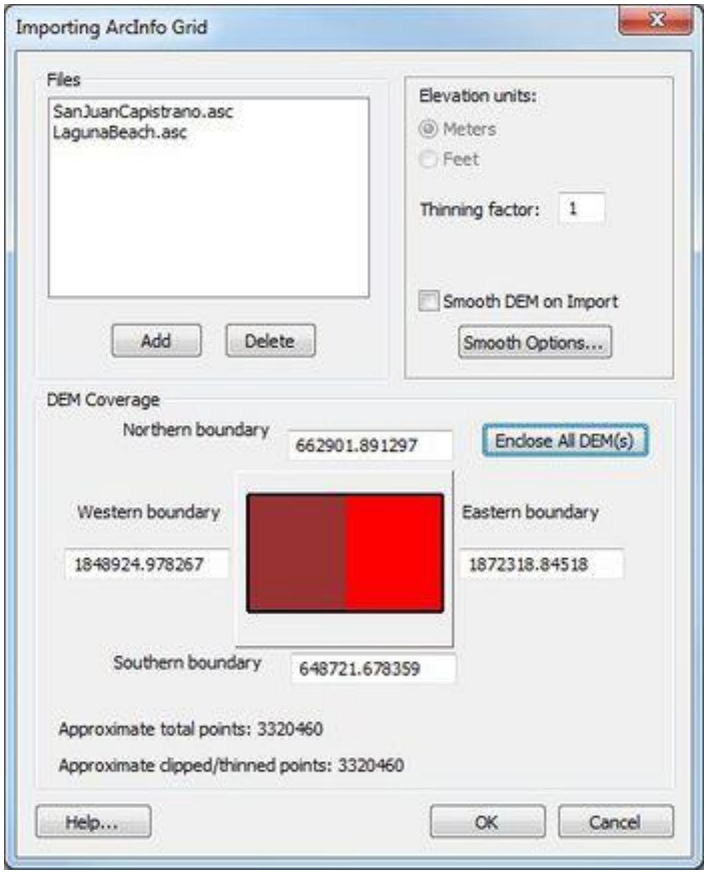
GRASS Watershed Grid	*.ggdd	
XYZ Data→TIN Verticies	*.xyz	Choosing this option allows importing a space-delimited file containing x, y, and z coordinate values. The **Triangulate** command in the *TINs* menu can then be used to create a TIN from the xyz data. The file must be space delimited in order 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 having access to data from WES, contact Aquaveo 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 there is additional information please contact the developers for help in implementation.
Intergraph DTM	*.dtm	This file format is no longer supported. If interested in using this file format, contact the WMS developers.
JPEG Image	*.jpg *.jpeg	JPEG image format file.
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 modify the 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

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 **Open** command in the *File* menu. This will bring up the *Importing ArcInfo Grid* dialog.



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 dataset 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 dataset */
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"	/* Dataset 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 dataset */
/* 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 Dataset 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	TINs 2D meshes 2D grids 2D scatter points

		mesh3d grid3d scat3d	3D meshes 3D grids 3D scatter points

Card Type	**BEGSCL**
Description	Scalar dataset 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 datasets 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 dataset 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 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.			
Field	*Variable*	*Size*	*Value*	*Description*
1	id	4 byte int	1 2	TINs Boreholes

			3	2D meshes
			4	2D grids
			5	2D scatter points
			6	3D meshes
			7	3D grids
			8	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 dataset.			
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 understanding the pathnames in DSS files, it is easier to understand how to use DSS files. DSS references datasets, 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:

/A/B/C/D/E/F/

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/01JAN1995/1DAY/OBS/

Related Topics

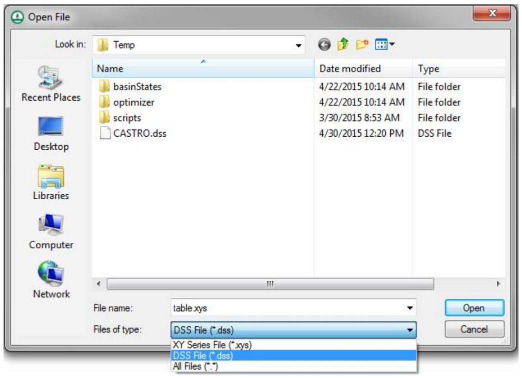
- DSS File I/O
- WMS Non-native Files
- DSS Interface

External Links

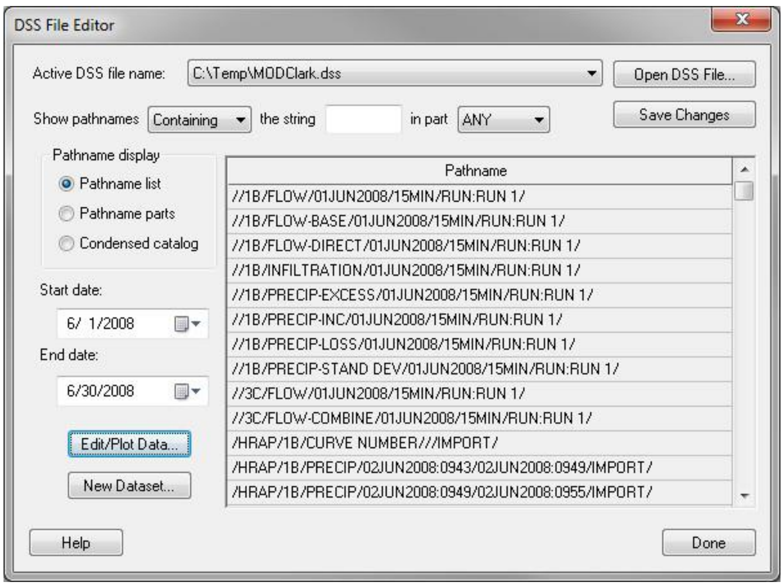
- HEC-DSS site

DSS File I/O

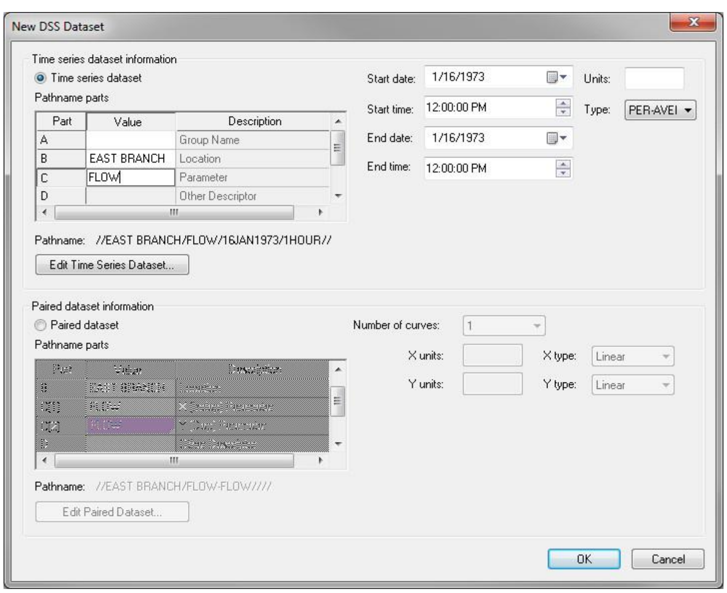
DSS files can be imported and exported from the *XY Series Editor* (using the **Import/Export** buttons) or from the **Open** command in the *File* menu. Selecting the **Import** command in the *XY Series Editor* will bring up the *Open File* dialog, which will have an option for reading a DSS file:



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 exporting the selected xy series to a DSS file using the *New DSS Dataset* dialog:



Related Topics

•DSS Files

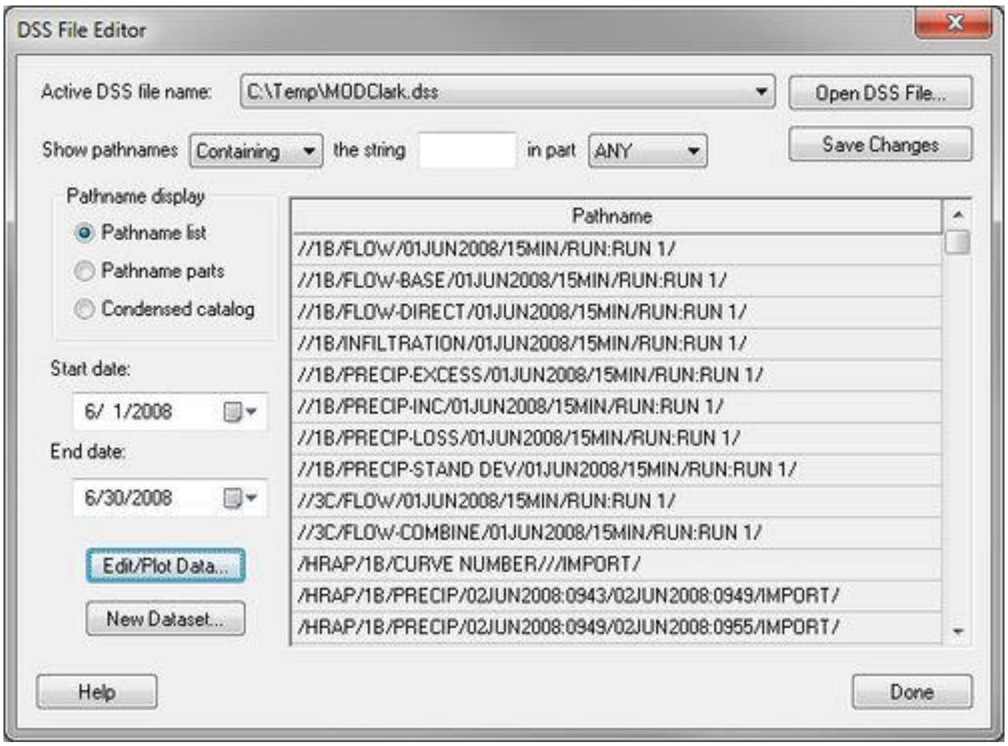
•XY Series Files

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 reading 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. That's what these options are for.

The edit field starts out with nothing in it. As entering something in this field, only those pathnames containing what has been typed are displayed. The search is case insensitive.

Pathname Display Group Box

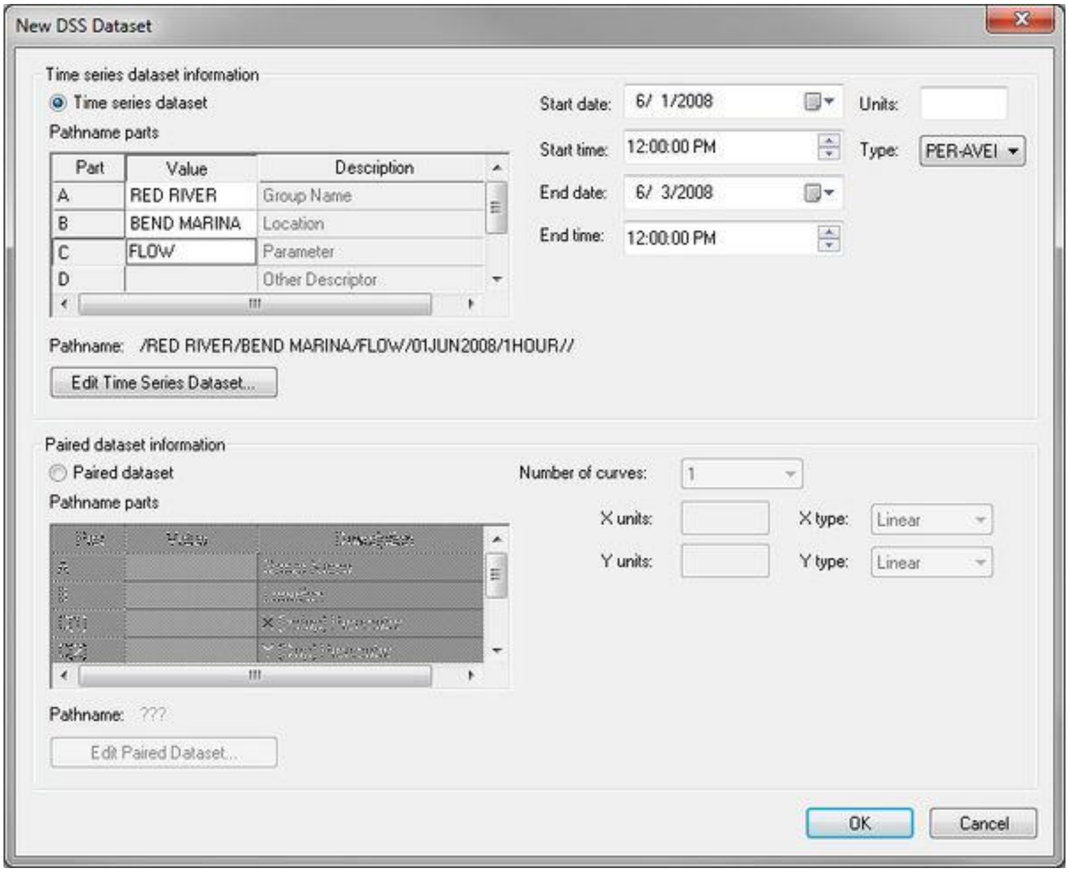
This radio group has options for different ways of displaying the pathname. The default will be the *Pathname list* option. This just displays the pathnames normally, with the "/" in between different parts. If the *Pathname parts* option is selected, WMS displays each part of the pathnames separately. The *Condensed catalog* option condenses the dates for different pathnames into a single pathname.

Edit/Plot Data Button

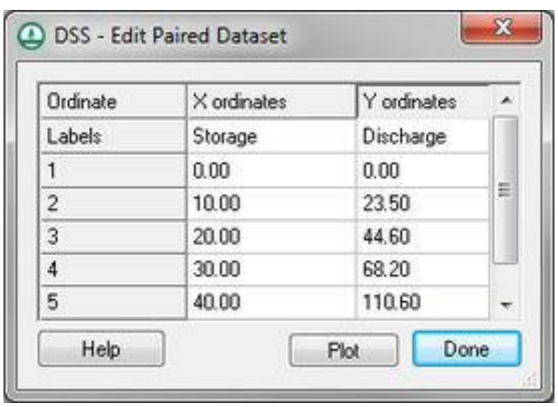
This button brings up the data from the DSS file for the selected pathname in the *XY Series Editor* which allows editing and plotting the data.

New Dataset Button

This button brings up the *New DSS Dataset* dialog which allows entering the pathname and other information for a new dataset:



Selecting the **Edit Time Series Dataset** button brings up the XY Series editor which allows editing 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 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. 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

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.

Figure 1. Hydrograph File Format

HYDROGRAPH
name starttime interval numordinates
flow-1 flow-2 flow-3 ... flow-10

...
...
...
flow-n

Figure 2. Sample Hydrograph File

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

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.

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.

Figure 1. The Image File Format

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 2. Sample Image File

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

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.

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. Create tables with the currently required data. The format of the table is shown in Figure 1, and a sample file in Figure 2.

Figure 1. Curve Number Land Use File Format

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 2. Sample Curve Number Land Use File

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

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.

Figure 3. Green & Ampt Land Use File

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 4. Sample 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

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. 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.

Figure 5. CASC2D Attribute Mapping File Format

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 6. Sample CASC2D Attribute Mapping File

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

Related Topics

- File Import Wizard
- WMS Native Files

Read Stage File

Water levels are used in WMS as scattered datasets and can be imported as scattered dataset (x, y, z). Files can be in a simple delimited format as shown here within a text file or spreadsheet and imported through the *File 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 dataset 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 wanting to bring forward the data from earlier versions of WMS then 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
- File 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 shapefile 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 always move them around together if copying or moving them to a new directory.

Only one theme, or type of feature, can exist in a shapefile. For example, it is not possible to store points and polygons in the shapefile, or streams and basin boundaries, so it 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.

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.

Figure 1. Soil Type Runoff Coefficient File Format

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 2. Sample Soil Type Runoff Coefficient File

1, "Highly impervious", 1.0
2, "Slightly pervious", .9
3, "Moderately pervious", .75
4, "Highly pervious", .5

Green & Ampt mapping file definitions and an example are given in Figure 3 and Figure 4.

Figure 3. Green & Ampt Soil Table File Format

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 4. Sample Green & Ampt Soil Table File

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

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, save the 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. 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.

Figure 1. The XY Series File Format

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 2. The Sample XY Series File

XY3 1 241 0 6 0 0 0 0 0 typeI-24hour
0.00000
0.00174
.
.
1.0000

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.

Card Type	**XY3**
Description	Defines a curve with a list of Y values. The x values are defined by a beginning value, an increment, and a bias.
Required	NO
Format	XY3 id n x1 incx biasx dx dy rep begc name y_1 y_2 . . y_n
Sample	XY3 1 10 0 1 0 0 0 0 0 head 0.0 2.0 7.0 8.0 9.5

'Field	*Variable*	Value	*Description*
1	id	+	The id of the XY series.
2	n	+	The number of point in the series.
3	x1	"+/-"	The first x value.
4	incx	"+/-"	The increment in x used to compute the next x value.
5	pcx	+	The per cent change in x used to compute subsequent x values. Expressed as a

			decimal, i.e., 0.05 = 5%.
6	dx	0,1	A flag defining whether the x values listed are to be interpreted as incremental (dx=1) or absolute (dx=0).
7	dy	0,1	A flag defining whether the y values listed are to be interpreted as incremental (dy=1) or absolute (dy=0).
8	rep	0,1	A flag defining whether the xy series is to be interpreted as cyclic (repeating)
9	begc	"+/-"	The x value in the series where the cyclic portion of the curve begins. Value is ignored if rep=0.
10	name	str	The name of the series
11	y	"+/-"	The y values of the points defining the curve. Repeat n times.

Related Topics

•XY Series Editor

•DSS Files

•WMS Native Files

9. Appendix

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 or by contacting a sales representative. Prices and other information about renewing your license are available on the WMS Pricing Page . The latest update to WMS can be downloaded here .**

WMS 10.1.7 (Beta) - Release January 18, 2016

•Initial Release!!

Intermediate Release 10.0.12 – February 8, 2015

•08570 - Duplicated 1D 2D BCs and Links Coverages have "linked" BC Attributes

•08579 - Nodes not Renumbered in Merged Coverages

•08474 - WMS crashes after right clicking on Terrain Data sub menu item.

•08400 - Link Icon not Created in the Correct Location for 1D Schematic

- 08421 - TOPAZ is reporting error about elevation values that is inaccurate
- 08131 - Problem saving HMS rainfall from WMS 10.0
- Other RegWiz Problems

Intermediate Release 10.0.11 – September 16, 2015

- 8132 - current projection has changed to object projection in basic HMS modeling tutorial
- 8134 - Edit | Current Projection needs to be replaced in the tutorials
- 8182 - WMS not exporting rainfall correctly for MODCLARK gridded precipitation
- 8179 - SMPDBK Flood Conversion
- 8124 - TC values not calculated
- 7996 - Strange error message when creating duplicate solution names in the project explorer
- 8022 - Bogus Flow Values message after computing basin data twice
- 7995 - Crash when creating new HY-12 Structure
- 8001 - .xys file Not Importing
- 8053 - HEC HMS Channel Routing Parameters not exported
- 8007 - WMS Crashes When Opening flood.wpr.
- 8013 - WMS Crashes When Opening Run1.wms
- 8014 - WMS crashes when clicking River Tools | Map -> 1D Schematic in the main menu.
- 8004 - MODRAT Interface Schematic - Tutorial needs updating for changes in the detention basin calculator
- 8020 - Add option to calibrate to Total Suspended Solids (TSS) at interior points in the GSSHA interface
- 0542 - Training slides need to be updated to discuss concepts intoroduced by the new DEM Delineation tutorial
- 4052 - Interface to create automated calibration files when the model is calibrated to one or more internal points
- 4762 - Replace WMS FHWA calculators with the Hydraulic Toolbox calculators (40 hours)
- 4833 - The project explorer needs an horizontal scroll bar and problems with the vertical scroll bar
- 5900 - Check into reading data into WMS from the national map viewer
- 5898 - Is there a way WMS could download the data on the national map viewer?
- 6120 - Test HMS 4.0 with WMS 9.1
- 7987 - NCDC Climate data web services used in WMS are being phased out
- 7597 - WMS dev can't access NSS_v6.mdb
- 8002 - Problem importing DEM in WMS 10.0 and later versions
- 8003 - Problem assigning index maps to mapping tables when there are different scenarios
- 7906 - Box culvert dimensions save incorrectly
- 7943 - Backwater cards not saving
- 7907 - GSSHA depth grid not saving/exporting correctly for import into ArcMap

Intermediate Release 10.0.10 – May 6 2015

•Merged revision(s) 35575 from devproj/trunk/Dev: Fixed issue 0008002: Problem importing DEM in WMS 10.0 and later versions. This problem was caused by SVN revision 33257. Revision 33257 changed how we handle the Y-resolution so it's not a...

Intermediate Release 10.0.9 – Feb 23 2015

•Merged revision(s) 33350 from devproj/trunk/Dev: * Fix problem with assigning GSSHA lake cells when the lake outlet node is downstream from an embankment edge on the grid ....

•Merged revision(s) 32899 from devproj/trunk/Dev: Fix problems with WMS GSSHA tutorial tests--when running multiple GSSHA scenarios, the mapping tables were using the same memory in each of the scenarios instead of making a new copy for each...

•Merged revision(s) 35539 from devproj/trunk/Dev: Fixed issue 0007943: Backwater cards not saving. See bug report for information about bugfix. ....

•Merged revision(s) 35516 from devproj/trunk/Dev: Fixed issue 0007907: GSSHA depth grid not saving/exporting correctly for import into ArcMap. See bug report for more information. ....

•Merged revision(s) 35514 from devproj/trunk/Dev: Fixed issue 0007906: Box culvert dimensions save incorrectly ....

•Merged revision(s) 35355 from devproj/trunk/Dev: Fixed issue 0007959: Multiple Items Being Selected When Creating New Materials in the Materials Data Dialog. See bug report for more information. ....

•Merged revision(s) 35321 from devproj/trunk/Dev: Fixed a problem with the rainfall grid and other grids (land use, soil type) not being displayed if the display option to display these grids is turned on. Also the option to display these g...

•Merged revision(s) 35294 from devproj/trunk/Dev: Fixed a problem with the grid control (spreadsheets) not copying/pasting values correctly after you have clicked on a toggle or a button. ....

•Merged revision(s) 34934 from devproj/trunk/Dev: Added dao.dll to WMS installation to fix a problem with NSS; Changed the bin.bat file so this file is also included in the 64 bit version. ....

•Merged revision(s) 34894 from devproj/trunk/Dev: Fixed issue 0007849: Selecting TIN Dataset Properties or View values Causes Crash: Bring up error message if no vertices exist that are associated with the selected dataset. ....

•Merged revision(s) 34885 from devproj/trunk/Dev: Fix a problem with not saving the GSSHA Overbank Flow and the Overland Backwater options in the GSSHA channel routing options dialog. We were not saving these values out correctly when the u...

•Merged revision(s) 34880 from devproj/trunk/Dev: Fixed issue #0007877: Problem computing storage capacity curve. If a user sets the base elevation to be below the bottom-most elevation in the basin, the hydraulic toolbox would always be in...

•Merged revision(s) 34342 from devproj/trunk/Dev: Get the CORINE Land Cover Europe dataset working (they added "2000" onto the end of the name...this caused it not to work). ....

•Renamed all the model build files to get rid of the "_" that was causing errors with the automated builds.

•Updated version numbers for WMS, HY8, and Hydraulic Toolbox (associated with WMS 10.0)

Intermediate Release 10.0.8 – Jan 28 2015

•Merged several revisions...some changes from revision 32779 and previous related revisions to convert the GSSHA model pointers to shared pointers. * Also added the Tikhonov option for the GSSHA model that was added to the dev version. * A...

•0007561: Renaming a Project Explorer item fails if mouse moved

---

•Update HY12, Hydraulic Toolbox, and HY8 version numbers so these are updated when you install WMS.

Intermediate Release 10.0.7 – Nov 13 2014

•Merged revision(s) 33893 from devproj/trunk/Dev: Commented out the ASTER web service from our program-defined options since it's no longer reliable (when we upgrade to Global Mapper 16 we will add the ASTER GDEM v2 web service in place of t...

•Merged revision(s) 33870 from devproj/trunk/Dev: Fixed crash associated with issue 0007666: Save/Export HMS model files crashes WMS See bug report for more information. ....

•0007574: Saving static image does not replace existing file.

•0007574: Saving static image does not replace existing file.

•Merged revision(s) 33360 from devproj/trunk/Dev: Fixed problems with WMS corrupting files after changes to the map, TIN, or scatter data because of the changes to the project-on-the-fly code. ....

•Merged revision(s) 33268 from devproj/trunk/Dev/WMS/APP/source/Hydrology/GSSHA/casc2d.cpp: Fixed issue 0007564: Arc Info Grid Is Different Than Original Grid See bug report for more information. ....

•Merged revision(s) 33248 from devproj/trunk/Dev: Work on issue 0007634: CE-QUAL-W2 Problems: See bug report for more information. ....

•Merged revision(s) 33189-33190 from devproj/trunk/Dev: Fixed issue 0007594: Link/Node data results displayed at -999.99 for nodes within GSSHA reservoirs. See bug report for more information. .... Fixed issue 0007594: Link/Node data re...

•Merged revision(s) 33054 from devproj/trunk/Dev: Fixed some problems with showing HGL/EGL Plots in the HY12 interface. Some nodes were not shown as part of the plot if they had gutter inlets but no access holes. Also, if you selected mult...

•Merged revision(s) 33039 from devproj/trunk/Dev: Fixed a problem with contours not showing up, caused by SVN check-in 33002 ....

•Merged revision(s) 33008 from devproj/trunk/Dev: Fixed issue 0007589: WMS crashes when mapping land use shapefile to coverage in Hydrologic Modeling Wizard See bug report for more information. ....

•Merged revision(s) 33002 from devproj/trunk/Dev: Fixed issue 0007324: WMS crashes when selecting a DEM in the Project Explorer Cleaned up some functions that had potential issues. ....

Intermediate Release 10.0.6 – Aug 6, 2014

•Merged revision(s) 32988 from devproj/trunk/Dev: Fix issue 0007432: GSSHA index map not recognized by mapping tables if deleted and recreated Also fixed other related issues and crashes related to deleting index maps and other GSSHA model d...

•Merged revision(s) 32947 from devproj/trunk/Dev: Fixed contour label issues and Bug 0007456: Placing One or More Contour Labels Inside of a Triangle Causes WMS to Crash

---

•Merged revision(s) 32912 from devproj/trunk/Dev: Fixed bug 0007478: WMS 10.0 crashes when deleting rainfall depth in GSSHA precipitation dialog This was a simple fix, just needed to check on the return value of GetItemData to make sure it w...

•Merged revision(s) 32906 from devproj/trunk/Dev: * Fix problems with mapping multiple shapefile attributes to a single coverage. You can map multiple shapefiles to a single coverage in WMS, but apparently not in SMS or GMS. Why not in SMS...

•Merged revision(s) 32869 from devproj/trunk/Dev: Update version of GSSHA we use with WMS 10.0 ....

•Merged revision(s) 32863 from devproj/trunk/Dev: Fixed issue 0007469: Problem with TR-55 tabular hydrograph method ....

•Merged revision(s) 32510 from devproj/trunk/Dev: Fixed a problem with the GSSHA model checker finding incorrect errors. ....

•Merged revision(s) 32125 from devproj/trunk/Dev: Fix problems with exporting rain gage information to HMS 4.0. The rain gages need to be ALL CAPS in the HMS meteorology file so they are in the same (upper) case as the gages in the HMS gage...

•Merged revision(s) 32016 from devproj/trunk/Dev: Removed unused dialogs and fixed some dialogs with radio buttons so the radio buttons work correctly. ....

•Ignore files

•Update All FHWA Projects to Use Boost 1.55

•Merged revision(s) 31669 from devproj/trunk/Dev: Fix Filename Issues in HY12: change how file names are managed in the HY12 interface-- -clean up how all the filenames are handled with HY12 to make sure everything is saved to the project di...

•Update Hydraulic Toolbox 4.2 Version number for update

•Merged revision(s) 31577 from devproj/trunk/Dev: Fix "ripraap" spelling errors to "riprap" ....

•Merged revision(s) 31571-31573 from devproj/trunk/Dev: Add 32-bit gssha.exe to WMS bin directory. .... Add 64-bit gssha.exe to WMS bin directory. .... Add 32 and 64-bit gssha.exe to WMS bin directories. ....

•Updated the HydraulicToolbox/Desktop Reference Guide.pdf based on the latest version from Bart

---

•Merged revision(s) 31486 from devproj/trunk/Dev: HY12 Redesign: Interface Bug Fixes: Remove Extra Lines in the HY12 Properties Dialog ....

•Merged revision(s) 31448 from devproj/trunk/Dev: Fixed issue 0007264: WMS crashes when editing GSSHA embankment arc: See bug report for more information. ....

Intermediate Release 10.0.5 – July 17, 2014

•Update HY8 7.31 Version Number for WMS 10.0 release update.

Intermediate Release 10.0.4 – June 12 2014

•Update HY8 version number so the HY8 build gets updated with the WMS 10.0 beta release.

•Update HY8 so it no longer has AOP analysis included in the interface (the code is still there, we just took out AOP).

•Merged revision(s) 31141 from devproj/trunk/Dev: Implement Inset model and related boundary conditions. ....

•Merged revision(s) 30939-30940 from devproj/trunk/Dev: Changed the SWMM Sanitary Code so it is a separate module from the normal SWMM module. .... Reverted revision(s) 30939 from devproj/trunk/Dev/WMS/APP/source/wmspass.cpp: Changed the...

•Merged revision(s) 30820 from WMS/WMS_10.0/trunk: Rename the Hydraulic Toolbox for testing ....

Rename the Hydraulic Toolbox for testing

•Updated HY12, HY8, and Hydraulic Toolbox solution filenames.

•Updated the WMS 10.0 Release solution name and version numbers and changed the name of the release executables.

•Branch WMS 10.0 from Dev

•Try to fix errors

•Identify specific header files that are including too much - Most WMS files using shared/mfc/mfcstuff.h.

•Update HY8 version number to update build that's installed with installation.

•Fix problems with HY8 bringing up the wrong dialogs from WMS. The indices associated with the command line arguments were wrong so I updated them.

•0007083: Warning Notification for Licenses Expiring Soon - fix null crash when no license exists.

•Fix problem from GUI tests that caused HMS to not run when writing gridded rainfall data.

•WMS::HY8::AOP Unit Test Fix

•Fixed a spelling error in the GSSHA dialogs.

•WMS::HY8::AOP Made more fixes to AOP, based on results of going through an example

•WMS::HY8::AOP Bug fixes

•WMS::HY8::AOP Bug fixes

•WMS::HY8::AOP Ripped out the Rock/Sediment gradations and put in a generic one. Also updated the culvert 'thalwegs' to include embedment depth and listed the steps. added file I/O and reports for the gradations. Also an embedment check.

•AHGW SourceGrid license, and Boost license.

•Added an attempted fix for a shared code SNAFU that messed up units when reading in previously-saved projection information from the WMS project file.

•Fixed the MODRAT interface so WMS writes the correct output filenames and so MODRAT writes/WMS reads the output files correctly. See bug report 0007166 and SVN check-in 30366 for related changes

•Fixed GUI test problem with WMS not writing out the HMS version number, which apparently is required when running HMS 4.0 or later.

•Upgrade to Boost 1.55 and signals2

•kmz.h missing xmscore::Pt3d prefix.

•Add DLL's to WMS 64-bit version and update DLL and EXE file descriptions in .bat file

•Fixed a bug with WMS crashing when shutting down after pulling the tree window off the mainframe, resizing, and then re-attaching the window.

Fixed issue 0007129: Problems Scrolling Side to Side in project Explorer (See bug report for more information)

•Attempt to fix issue 0007175: Problem running NSS models in WMS Dev

•Missing forward declaration for display atts (EDispAttLine). Can cause compile errors elsewhere without this.

•Added an error message when delineating basins and no outlets exist.

•Fixed issue 0007162: Delineate Basins Wizard should delineate the watershed - not launch the Hydrologic Modeling Wizard (See bug report for more information)

•Fixed bug 0007166: Error running simulation when changing MODRAT executable file (See bug report for more information)

•Moved SetState to protected and switched all calls to it to SetStatePropogate like in SMS.

•0003072: Setup mesa for 64-bit - Get Mesa from its own 32 or 64-bit folder.

•Fixed issue 0007168: GSSHA Rainfall Time Inconsistency (See bug report for more information)

•Fixed issue 0007176: Problem saving GSSHA Storm Drain coverage (See bug report for more information)

•0007083: Warning Notification for Licenses Expiring Soon - Show days remaining after 5 if both licenses will expire.

•WMS::HY8 Allow the user to select the errors.

•0007083: Warning Notification for Licenses Expiring Soon

•Fixes for project explorer state not matching displayed state when projects are loaded in GMS & WMS.

•WMS::HY8 change version to force a build

•Adjustments to fix bugs discovered in AOP

•Add GSSHA options for outputting point overland depth and water surface elevation information.

•Common GIS module - Make sure all 3 programs read files as defined in the design doc - Hide new GIS. General GM file type. World file and zip file readers/complainers. Fix CanGlobalMapperOpenFile. Move GIS to shared.

•Added static and rating curve lake options to detention basins defined at feature nodes and polygon lakes. * Added a rating curve spreadsheet dialog so users can input and edit rating curves for detention basins/lakes. * Added map and GSS...

•Common GIS module - Make sure all 3 programs read files as defined in the design doc - GMS, SMS, and WMS. Shared GM reader. Only jpg so far.

•Force a rebuild on HY8

•WMS::HY8 Add some commandline arguments to allow HY-8 to disable crossing input

•WMS::HY8 Add some commandline arguments to allow HY-8 to disable crossing input

•Remove test code for calls to ComputeClippedPoints function.

•Committed changes to demio.cpp for debugging release 32 version.

• Separated out two dialog files from flineatt.cpp (I need to modify them and the code's all messed up) and modified seGetSelectedFeaturePoints to use a vector instead of malloc'ing memory. Cleaned up some code. Fixed a problem with feature...

•Added more GIS mapping for the dynamic model interface.

•BUGFIX: Fix to state functionality in shapefile ETreeItems for WMS

•Fixed problem with GSSHA crashing when exporting stochastic data files.

•Fix for WMS ETreeItem state based on messages from Chris and comparison with WMS 9.1

•Moving tree_enum and troot_enum into xms namespace for SMS and WMS.

•Allow HY-8 to run with an exceptionally tall roadway in commandline. Still need to allow it to be entered, get past the data checks, and modify the interface for it.

•Added arc-specific OVERBANK and BACKWATER channel card options to GSSHA file I/O and user interface. Check to see if these options are set on any of the arcs when entering the project channel data dialog and when exporting the GSSHA projec...

•Fixing WMS Checkboxes I think.

•Clean up lots of code (move classes out of globals.h) * Fix a problem with the stream smoothing dialog that does not display a plot if one of the groundwater datasets is not defined.

•Fix for BUG# 0007125

•Update for fix to bug 0006856: WMS hangs when reading project

•Unification of tree icons across products.

•Update the EPA SWMM executable file directory to the default installation folder for the new version.

•Added code to read/write the Depth Varying Overland Flow Roughness mapping tables (GSSHA), ROUGH_EXP to/from the GSSHA .cmt file.

•Common GIS module - Make sure all 3 programs read files as defined in the design doc - GMS has optional new GIS, image reader for FILE_JPG to FILE_KMZ_VECTOR, default GM reader, merge GMS into shared file types, adapt unit tests to merged t...

•WMS::HY8::AOP Updates for bugs that were noticed in a sprint meeting; Also update of the version number to force a build.

•Fixed root display oder for WMS.

•Cleaned up some more GSSHA code.

•Moved ETreeItem state functionality to be all new state functionality from SMS. All statements within #ifndef statements for the new state functionality should either be commented out, deleted, or migrated to the new state functionality. C...

•Fix Errors in WMS build

•Cleaned up code in casc2d.h (removed headers and moved stuff to .cpp files) and made GSSHA parent/child model dialog resizable.

•WMS::HY8::AOP Fix an bigger issue with Units, also update the version number and force a new build

•WMS::HydraulicToolbox Update version number to force a new build/install

•Update NSS version number and database.

•Common GIS module - Make sure all 3 programs read files as defined in the design doc - Change WMS to not use new GIS, but use new common enumeration.

Added GSSHA model check to determine if cells containing rating curve BC's have adjacent cells marked as receiving cells.

•Fixed a problem with too many options showing up in the Compute Topaz step of the Hydrologic Modeling Wizard. Needed to reset the combo box before it's set up.

•WMS::HY8::SRH Add new commandline arguments for HY8, to run within SMS. Allow SMS to call for a crossing just knowing the GUID, not the index, if it doesn't exist, create one.

Intermediate Release 9.1.12 – May 6, 2014

1. Fixed the WMS 2D grid cell traversal algorithms to handle line segments that begin and end outside the bounds of the 2D grid or the active cells on the 2D grid.
2. Fixed the HY8 Crossing Discharge Data dialog to get the hydrograph whether it's defined at the outlet or at the basin associated with the outlet.
3. Fixed WMS so circular stream arcs do not cause WMS to crash when selecting File | New or when shutting down.
4. Fixed a problem where Mapping Shapefile causes crash
5. Fixed a Crash when reading in a map file
6. Fixed a bug with reading tailwater curve ratings of a different length
7. Fixed some reading issues
8. Fixed an issue to Update Culvert Info if reading an older file

Intermediate Release 9.1.10 – November 22, 2013

WMS 9.1.10 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 . The following revisions have been made to this version:

•26787; Author: cdickson; Date: Thursday, November 21, 2013 9:09:05 AM; Message: Web form option for getting license from the license manager - WMS9.1

•26719; Author: csmemoe; Date: Monday, November 18, 2013 2:47:37 PM; Message: Removed failing test assert that isn't necessary. Only needs to test vertical units. (See SVN revision 24750)

•26713; Author: csmemoe; Date: Monday, November 18, 2013 12:06:36 PM; Message: Merged revision(s) 26712 from devproj/trunk/Dev: Changed WMS so it reads and saves whether each coverage is visible.

•26557; Author: csmemoe; Date: Monday, November 11, 2013 9:43:46 AM; Message: Merged revision(s) 26550 from devproj/trunk/Dev/WMS/APP/source/display/mapdraw.cpp: Fixed problems with framing image and getting max/min world coordinates when a rain gage coverage exists. Also cleaned up some of the setmap code to specify const parameters a little better.

•26542; Author: csmemoe; Date: Friday, November 08, 2013 11:27:03 AM; Message: Merged revision(s) 26541 from devproj/trunk/Dev: Fixed some issues with the TR-55 computation dialog caused by my check-in yesterday--the incorrect cells were getting set to read-only. Also the Fp value was not showing up in the dialog. Cleaned up some code.

•26535; Author: csmemoe; Date: Thursday, November 07, 2013 5:46:46 PM; Message: Merged revision(s) 26534 from devproj/trunk/Dev: Try to fix a problem with the TR55 window crashing in the dev release version when it first comes up.

•Revision: 26508; Author: csmemoe; Date: Wednesday, November 06, 2013 5:34:24 PM; Message: Merged revision(s) 26505 from devproj/trunk/Dev: Fixed text attribute strings in compute GIS attributes dialog.

•Revision: 26363; Author: csmemoe; Date: Wednesday, October 30, 2013 6:10:51 PM; Message: Merged revision(s) 26361 from devproj/trunk/Dev: Fixed problems importing HEC-RAS geometry files-did not read Manning's roughness correctly from these files. Also cleaned up lots of the shared/rivmod code.

•26237; Author: csmemoe; Date: Friday, October 25, 2013 2:37:18 PM; Message: Update WMS Release Version number

•Revision: 26226; Author: csmemoe; Date: Friday, October 25, 2013 10:27:42 AM; Message: Fixed issue 0006656: Helpstrings for "Compute Time Area Curve" button and "Compute Parameters – Basin Data" button are switched

•26216; Author: csmemoe; Date: Thursday, October 24, 2013 5:57:48 PM; Message: Merged revision(s) 26137 from devproj/trunk/Dev: WMS::HY-8 Fix for help PDF Help (replaces temporary check)

•Revision: 26215; Author: csmemoe; Date: Thursday, October 24, 2013 5:55:15 PM; Message: Merged revision(s) 26205 from devproj/trunk/Dev: Fixed problems with plot menus not coming up in HY12 plot dialogs and in detention basin storage capacity curve dialog.

•26019; Author: csmemoe; Date: Tuesday, October 15, 2013 1:48:30 PM; Message: Merged revision(s) 26017 from devproj/trunk/Dev: Fixed issue 0002019: Problem reading HEC-RAS solution for WMS Dev and HEC-RAS 4.1, fixed a problem running HEC-RAS from WMS, changed the stochastic modeling code so it does not require user interaction, and fixed stochastic modeling code so HEC-1 runs do not depend on the length of the path name to the HEC-1 input file. Also fixed a crash when running floodplain delineation after a stochastic run and a crash when exiting WMS.

•25976; Author: csmemoe; Date: Friday, October 11, 2013 1:47:00 PM; Message: Merged revision(s) 25975 from devproj/trunk/Dev: Fix problem with joining SSURGO data when you have "spatial" somewhere in the path of the shapefile other than under the base directory containing the shapefile data.

•25968; Author: csmemoe; Date: Friday, October 11, 2013 9:57:08 AM; Message: Merged revision(s) 25946, 25967 from devproj/trunk/external_libs:

a) Add user_online_sources.xml file to specify online sources that are not available in Global Mapper by default (Added FEMA flood map web services).

b) Added FEMA flood maps to list of possible web services.

c) Merged revision(s) 25965 from devproj/trunk/Dev:

d) Added FEMA flood maps to list of possible web services.

•25940; Author: csmemoe; Date: Thursday, October 10, 2013 9:55:26 AM; Message: Merged revision(s) 25935 from devproj/trunk/Dev: Added support for Web Feature Services (WFS) to global mapper web services. Handle reading vector data from OGC WFS services.

•25911; Author: csmemoe; Date: Tuesday, October 08, 2013 4:04:38 PM; Message: Merged revision(s) 25909 from devproj/trunk/Dev: Fixed crashes when exporting riprap profiles to a report.

•25892; Author: csmemoe; Date: Tuesday, October 08, 2013 9:50:54 AM; Merged revision(s) 25876 from devproj/trunk/Dev/WMS/APP/source/display/displayopt.cpp: Added option to display cross section ID's and display options.

•25803; Author: csmemoe; Date: Wednesday, October 02, 2013 9:25:47 PM; Message: Fixed an error caused by a merge

•25798; Author: csmemoe; Date: Wednesday, October 02, 2013 5:45:55 PM; Message: Merged revision(s) 25797 from devproj/trunk/Dev/WMS/APP/source/Hydrology/GSSHA/casc2d.cpp: Fix a problem with reading solution files when not running a calibration and maintain code that correctly reads calibration solution files.

•25657; Author: ejones; Date: Thursday, September 26, 2013 11:12:38 AM; Message: Merged revision(s) 25656 from devproj/trunk/Dev/WMS/APP/bin/modrat: Fixed a bug in Modrat with generating a hydraulic parameter table used for channel routing

•25518; Author: csmemoe; Date: Wednesday, September 18, 2013 3:32:23 PM; Message: Merged revision(s) 25517 from devproj/trunk/Dev: Cleaned up some code in ocmodels.cpp-determining what Tree Mapping does.

•25487; Author: csmemoe; Date: Tuesday, September 17, 2013 4:20:31 PM; Message: Fixed a problem with images that were not displayed being checked in the project explorer window.

•25240; Author: csmemoe; Date: Friday, September 06, 2013 9:25:10 AM; Message: Merged revision(s) 25239 from devproj/trunk/Dev: Fixed issue 0006424: WMS not saving user defined DEM names

•25226; Author: csmemoe; Date: Thursday, September 05, 2013 3:04:31 PM; Message: Merged revision(s) 25223 from devproj/trunk/Dev/WMS/APP/source/Hydrology/GSSHA/casc2d.h: Fixed issue 0006481: Error Opening Summary File in Gssha Calibration (tut 57) and cleaned up lots of code related to solutions, etc.

•25207; Author: csmemoe; Date: Wednesday, September 04, 2013 4:54:01 PM; Message: Merged revision(s) 25206 from devproj/trunk/Dev: Fixed issue 0006287: Difference between 32 and 64 bit version in tut63. Also added capability to print out link/node names in detailed report file for selected links nodes.

•25139; Author: csmemoe; Date: Tuesday, September 03, 2013 8:21:44 AM; Message: Updated the HY-8 7.30 version number.

•25012; Author: csmemoe; Date: Monday, August 26, 2013 10:48:44 AM; Message: Update WMS 9.1.x version number.

•25011; Author: csmemoe; Date: Monday, August 26, 2013 10:47:16 AM; Message: Update hydraulic toolbox 4.0.x version number.

•24964; Author: csmemoe; Date: Friday, August 23, 2013 1:44:47 PM; Message: Merged revision(s) 24951 from devproj/trunk/Dev/WMS/APP/source/Feature/fcoverage.cpp: Added support to automatically map the global land use and soil shapefile data to map module feature object data.

•24933; Author: csmemoe; Date: Thursday, August 22, 2013 6:35:07 PM; Message: Merged revision(s) 24932 from devproj/trunk/Dev: Fixed problem with virtual earth map locater tool showing up in the ocean if you have the Hydrologic modeling wizard up.

•24925; Author: csmemoe; Date: Thursday, August 22, 2013 4:28:55 PM; Message: Merged revision(s) 24924 from devproj/trunk/Dev: Fixed WMS so it sets the main channel slope if the string "main channel slope" or other possible names are found in the NSS parameter description. ....

•24912; Author: csmemoe; Date: Thursday, August 22, 2013 2:58:36 PM; Message: Merged revision(s) 24908 from devproj/trunk/Dev: Added USA Topo maps to the main online data dialogs so you don't have to click on the Advanced button.

•24911; Author: csmemoe; Date: Thursday, August 22, 2013 2:56:01 PM; Message: Merged revision(s) 23041 from devproj/trunk/Dev: Fixed problem with web services that did not allow downloading data from http://xmswiki.com/xms/WebServiceCatalog

•24886; Author: ejones; Date: Thursday, August 22, 2013 11:50:04 AM ;Message: Fixes for Modrat Cross Section WorkMerged revision(s) 24871-24883 from devproj/trunk/Dev/WMS:

e) Rectangular cases are now matching closely!

f) Fixes for Modrat Cross Section Work

g) Fixes for Modrat Cross Section Work

h) Fixes for Modrat Cross Section Work

•24862; Author: csmemoe; Date: Wednesday, August 21, 2013 5:21:42 PM; Message: Fixed WMS 9.1 to be able to read online images with colons and other unacceptable characters in the filename.

•24859; Author: csmemoe; Date: Wednesday, August 21, 2013 5:08:10 PM; Message: Handle online images with unauthorized characters in the filename (\/:*?"<>|) and handled exceptions in online images without crashing.

•24848; Author: csmemoe; Date: Wednesday, August 21, 2013 2:35:31 PM; Message: Fixed a crash when getting data from map or when using the Get Data tool if you select an option from the "Advanced" button in the Online Data dialog. Also fixed a problem with characters that cannot be used in filenames when using this button.

•24747; Author: csmemoe; Date: Monday, August 19, 2013 3:00:59 PM; Message: Merged revision(s) 24743 from devproj/trunk/Dev: Fixed problem that caused vertical units to change when selecting the Virtual Earth map locater tool. See issue 0006240: After setting the current projection vertical units to feet from meters (Tutorial 62)

•24683; Author: csmemoe; Date: Thursday, August 15, 2013 5:51:31 PM; Message: Merged revision(s) 24682 from devproj/trunk/Dev: Updated QuickPDF Unlock Code to permanent license code (until we upgrade global mapper and there's a new version of QuickPDF included with GM).

Intermediate Release 9.1.7 – August 13, 2013

WMS 9.1.7 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 . The following known bugs have been fixed in this version:

Id-Date Submitted-Updated-Summary

•6464-7/31/2013-8/7/2013-Hydrograph timing messed up when exporting HEC-HMS hydrographs with a SWMM input file

•6456-7/26/2013-8/7/2013-Problem reading NRCS data from soil files

•6471-8/2/2013-8/5/2013-Crash when saving TIN to grid format

•6463-7/31/2013-8/5/2013-wmsdss.exe crashes when reading HEC-HMS solution

Intermediate Release 9.1.6 – July 18, 2013

WMS 9.1.6 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 . The following known bugs have been fixed in this version:

Id-Date Submitted-Updated-Summary

•5001-4/19/2012-6/25/2013-The HSPF dialog doesn't close after you define the parameters for Snow and Water.

•6219-5/9/2013-6/18/2013-"In Tutorial 10, HEC-1 is unable to open input file."

•6129-4/11/2013-6/18/2013-Problem running HEC1 on computers at NHI training course

•6165-4/24/2013-6/18/2013-HEC-1 File Name and Path limitations (from Scott Hogan @ FHWA)

•6068-3/21/2013-6/18/2013-HEC-1 Fails to load input files with a filepath >80 characters

•6314-6/6/2013-6/14/2013-Bug computing storage from DEM when vertical units are in meters

•6053-3/18/2013-6/13/2013-Two Time Computation Saving Permissibility

•6204-5/6/2013-6/13/2013-Intermediate tests failing

•6286-5/28/2013-6/13/2013-Default Selection in 64-bit version HEC-1 Unit Hydrograph Methods window Bug (Watershed Modeling - MaricopaNSS)

•6250-5/17/2013-6/12/2013-Lines extend when you select the Hydrological modeling module (Tut 15)

•6289-5/28/2013-6/11/2013-Crash when mapping shapefile to features

•6239-5/16/2013-6/11/2013-The Peak & Volume Boxes are read only when they should be editable (Automated Calibration Tutorial 58)

•6057-3/19/2013-5/7/2013-Hydrolic Modeling Dialog Crashes on NULL Values

•6130-4/11/2013-5/3/2013-Rational Method Interface tutorial has problems

•6201-5/3/2013-5/3/2013-WMS Crashes in Tutorial 4 Section 6.1

•6184-4/29/2013-5/3/2013-Problem renumbering storm/tile drains in WMS 9.1

•6178-4/29/2013-5/3/2013-Documentation missing step (W_20)

•6193-5/1/2013-5/2/2013-Problems with the units in tutorial 36

•6194-5/2/2013-5/2/2013-"Message box pops up that says ""Error: No links found"" after saving GSSHA project file (Tutorial 46)"

•6183-4/29/2013-5/1/2013-Problem assigning groundwater BC to streams in GSSHA model

•6182-4/29/2013-5/1/2013-"Problems reading, writing, and editing files for the Multi Layer Green and Ampt (MLGA) method in WMS 9.1"

•6166-4/24/2013-5/1/2013-Problem Trimming DEM

•6176-4/29/2013-4/30/2013-WMS crashes on changing Point Display step for DEM

•6117-4/9/2013-4/18/2013-Cryptic Error code message when online image servers are down

•6016-3/7/2013-3/13/2013-"Selecting ""Show Dates"" in User Hyetograph dialog changes time interval in dss files"

•6032-3/12/2013-3/12/2013-Problem reading ArcInfo and GRASS ascii flow directions and accumulations

•6027-3/8/2013-3/11/2013-Deleting a point takes a very long time

•6010-3/7/2013-3/8/2013-WMS crashes you try to view scatter dataset properties

•6011-3/7/2013-3/8/2013-Opening *.hdr file created in ArcGIS crashes WMS

•6012-3/7/2013-3/8/2013-WMS crashes when click OK on XY Series Editor when defining a hydraulic structure

•6000-3/6/2013-3/7/2013-"WMS reverts back to same ""All"" row when you change it in HMS properties dialog"

•6013-3/7/2013-3/7/2013-CE-QUAL top layer widths less than the widths of the Segments

•4496-12/15/2011-3/7/2013-"Unable to read a CAD file, probably Shared Code issue"

•4530-1/3/2012-3/7/2013-Problem getting online images when using remote desktop

•4531-1/3/2012-3/7/2013-Problem with registration of a static image created from an online image

•4538-1/4/2012-3/7/2013-The 'Setup All Precip Events' button does not work

•4580-1/12/2012-3/7/2013-WMS not responding

•4649-1/31/2012-3/7/2013-DEV - WMS freezes when converting online image to GeoTIFF

•4683-2/7/2012-3/7/2013-Oblique view zooming issue with GSSHA solution

•4690-2/8/2012-3/7/2013-Importing CMT file in WMS 8.4 does not import all data

•4694-2/9/2012-3/6/2013-Can't import more than 1 GeoTIFF DEM at a time

•4703-2/10/2012-3/6/2013-"Update ""Images"" menu in the Map Data menu items"

•4757-2/22/2012-3/6/2013-WMS fails to save a CAD file for hydrologic modeling tree data

•4761-2/23/2012-3/6/2013-Problem reading TIN datasets after computing using the data calculator

•4552-1/6/2012-3/6/2013-GSSHA makes computer run out of memory when running a constituent transport simulation

•5984-3/4/2013-3/6/2013-Problem with Getting Online Maps when only CAD data is defined

•5982-3/4/2013-3/5/2013-DXF and DWG files display problem: Thick lines

•5082-5/7/2012-3/5/2013-Channel Calculator issue

•5092-5/8/2012-3/5/2013-msvcr71.dll error when running wmstopaz.exe

•5518-9/26/2012-3/5/2013-Save project crash with GeoPDF with cropped collar

•5552-10/5/2012-3/5/2013-WMS is unable to write to the worldfile unless images are stored in the project directory

•5572-10/11/2012-3/4/2013-Check long term simulation tutorial and calibration tutorial to see if everything is working fine with WMS 9.0

•5601-10/18/2012-3/1/2013-Problem displaying flow paths and moving basin labels

•5495-9/18/2012-3/1/2013-Unable to download many of the soil and land use options in WMS

•5605-10/18/2012-3/1/2013-WMS is not creating frames for the .AVI animations for GSSHA solutions

•5713-11/28/2012-2/28/2013-"When adding new polygon, all previous colors are reset"

•5826-1/15/2013-2/28/2013-WMS doesn't create Routing Hotstart File when option is checked on

•1140-7/10/2009-2/27/2013-Join SSURGO data doesn't work when adding a custom attribute before joining

•5062-5/3/2012-2/27/2013-freeze when changing all arcs in a GSSHA coverage to a different type

•3952-8/10/2011-2/26/2013-Selected crossing/crossing name not saved

•4189-9/29/2011-2/25/2013-Support .las and .dtm file formats.

•5902-2/6/2013-2/25/2013-Problem reading solutions as 2D scattered data from HEC-RAS GIS files

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 . 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 . 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 . 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/ . 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

9. Fixed MODRAT/WMS bugs
10. Rewrite all recursive functions associated with basin delineation and basin data computation
11. 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)
12. 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)
13. Change "Compute Basin Data" function to only compute selected basin data parameters (may significantly reduce computation time for large datasets)
14. 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/ . 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/ . 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/ . 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/ . 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/ . 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/ . 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/ . 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/ . 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/ . 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 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 .

- 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

Publications Involving WMS

Monsef, H. A.-E. (2015, July) "A mitigation strategy for reducing flood risk to highways in arid regions: a case study of the El-Qseir–Qena highway in Egypt" in *Journal of Flood Risk Management* . DOI: 10.1111/jfr3.12190. [73]

Hopkinson, L.C., Sears, A. E., Snyder, M., O'Leary, E., DePriest, N., Quaranta, J.D., and Ziemkiewicz, P.F. (2015, December). "Simulating the hydrologic response when streams are incorporated in valley-fill design". DOI: 10.1080/17480930.2015.1105180. [74]

Xu, Haowen. (2015, October). "Prototype hydroinformatics-based system for supporting decision making in culvert design and monitoring." Master of Science thesis at the University of Iowa. [75]

Edsel B.D., et al. (2011). "Watershed Modeling and its Applications: A State-of-the-Art Review." *The Open Hydrology Journal,* 5: 26-50. [76]

Sadrolashrafi, S.S., et al. (2008). "Integrated Modeling for Flood Hazard Mapping Using Watershed Modeling System." *American Journal of Engineering and Applied Science,* 1: 149-156. [77]

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.

Ertuck, A., et al. (2006). "Application of Watershed Modeling System (WMS) for Integrated Management of a Watershed in Turkey." *Journal of Environmental Science and Health, Part A: Toxic/Hazardous Substances and Environmental Engineering,* 41(9): 2045-2056.

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. (April 2004). “Floodplain Risk Analysis Using Flood Probability and Annual Exceedance Probability Maps,” Ph.D. Dissertation, Brigham Young University. [78]

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.

Norman, N., Nelson, E., and Zundel, A. (2003). “An Improved Process for Floodplain Delineation from Digital Terrain Models,” ‘’ASCE Journal of Water Resources Planning and Management,‘’ 129(5): 427-436.

Omer, C.R., Nelson, E.J., and Zundel, A.K. (April 2003). “Impact of Varied Data Resolution on Hydraulic Modeling and Flood Plain Delineation,” *Journal of the American Water Resources Association,* 39(2): 467-475.

Green, J.I., and Nelson, E.J. (April 2002). “Calculation of Time of Concentration for Hydrologic Design and Analysis Using Geographic Information System Vector Objects,” *Journal of Hydroinformatics,* 4(2): 75-81. [79]

Noman, N.S., and Nelson, E.J. (2002). “ArcGIS Hydro Data Model – Chapter 5 River Channels,” *ESRI Press,* 31 pp.

Noman, N.S., Nelson, E.J., and Zundel, A.K. (Nov/Dec 2001). “A Review of Automated Flood Plain Delineation from Digital Terrain Models,” *ASCE Journal of Water Resources Planning and Management,* 127(6): 394-402.

Nelson, E.J., Miller, A.W., and Dixon, E. (2000). "Mud Canyon Fire: A Hydrologic Evaluation of Rainfall Runoff from a Burned Watershed," *International Journal of Wildland Fire,* 9(1).

Nelson, E.J., Jones, N.L., and Berrett, R.J. (1999). "Adaptive Tessellation Method for Creating TINs from GIS Data," *ASCE Journal of Hydrologic Engineering,* 4(1): 2-9.

Nelson, E.J., Moglen, G.E., and Arneson, L. (January 1999). "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,* 10(14). Published on CD ROM.

Nelson, E.J., and Jones, N.L. (1995). "Reducing Elevation Roundoff Errors in Digital Elevation Models," *Journal of Hydrology,* 169: 37-49.

Nelson, E.J., Jones, N.L., and Miller, A.W. (March 1994). "An algorithm for precise drainage basin delineation," *ASCE Journal of Hydraulic Engineering,* 120(3): 298-312.

Jones, N.L., Wright, S.G., and Maidment, D.R. (October 1990). "Watershed delineation with triangle based terrain models," *ASCE Journal of Hydraulic Engineering,* 116(10): 1232-1251.

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)

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

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. [80]

Garbrecht, J., and Martz, L.M. (March 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*, pp. 221-229.

Maidment, D. R., ed. (1993). "Handbook of Hydrology," (New York: McGraw-Hill), 1424 pp.

Martz, L.M., and Garbrecht, J. (March 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*, pp. 211-220.

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

Jenson, S.K. and Domingue, J.O. (1988). "Extracting topographic structure from digital elevation data for geographic system analysis", *Photogrammetric Engineering and Remote Sensing*, 54(11): 1593-1600.

Band, L.E., (1986). "Topographic partition of watersheds with digital elevation models", *Water Resources Research*, 22(1): 15-24.

Franke, R. (1982). "Scattered data interpolation: tests of some methods," *Mathematics of Computation*, 38(157): 181-200. [81]

Grayman, W. M., Males, R.M., and Harris, J.J. (July 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.

Watson, D.F. (1981). "Computing the n-dimensional Delauney tesselation with application to Voronoi polytopes," *The Computer Journal*, 8(2): 167-172. [82]

Lee, D.T., and Schacter, B.J. (1980). "Two algorithms for constructing a Delauney triangulation," *International Journal of Computer and Information Sciences*, 9(3): 219-242.

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

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

Shepard, D. (1968). "A two dimensional interpolation function for irregularly spaced data," *Proceedings of the 23rd National Conference ACM*, pp. 517-523. [83]

Clough, R.W., and Tocher, J.L. (1965). "Finite element stiffness matrices for analysis of plates in bending," Proceedings of the Conference on Matrix Methods in Structural Mechanics, AFFDL-TR-68-150, Wright-Patterson A.F.B., Ohio , *pp. 515-545. [84]*