InfoSewer™

Modeling Made Easy

Introduction to InfoSewer

Welcome to the InfoSewer Help File! One of the most powerful GIS Sewer modeling software in the world with many tools for Sewer Design, Water Quality, Runoff, RDII and Steady State/EPS Modeling! We'd like to take this opportunity to thank our customers for using InfoSewer , and show and describe how this software can better meet your modeling needs. You can learn more About InfoSewer by clicking on the Pop Up Links in this Help Topic Table.

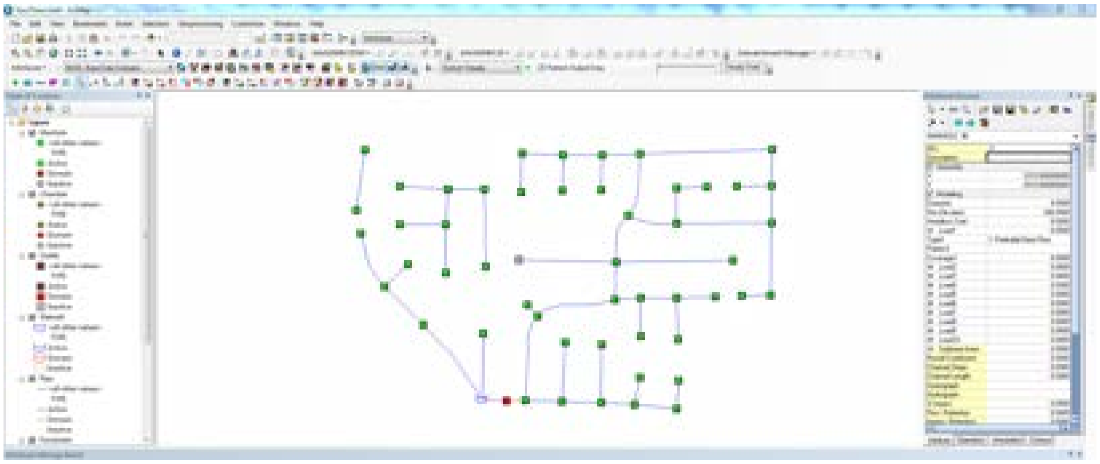
What's New	**User Interface**
Projects for InfoSewer	**Data Elements (Pipes, Manholes, Pumps, Wet Wells)**
Preferences and Utilities	**EPS/Dynamic Modeling**
Steady State Modeling	**Database Management**
Stormwater Modeling	**Curves and Patterns**
Scenarios and Sets	**GIS Overview**
Domains and Facilities	**Reports, Graphing and Output**
Running a Model	**User's Guide**
Design	**FAQ's and Blogs**
Printing and Plotting	**InfoSewer vs InfoSWMM**
Sewer Theory	**Innovyze St Venant Solutions**
Water Quality Theory	**How to Make Large Icons in Arc GIS or Arc Map**

InfoSewer bridges the gap between network modeling and Arc GIS based software to support many types of applications in sewer collection system analyses, including the following features:

Master Planning	**CMOM Program Planning**
Facility Sizing	**Wet-Well Design**
Facility Management	**Outlet Configuration**
Operations Study	**Pump Cycling**

Real-time Simulation	**Infrastructure Rehabilitation/Replacement**
Peak Flow Effects	**Flow Capacity Evaluation**
Overflow Management	**Infiltration/Inflow Modeling**
System Deficiency Identification	**System Improvements**
BOD Modeling	**Odor Control**
Source Tracing	**Pollutant Contribution Analysis**
Discharge Permit Compliance	**Corrosion Control**
Treatment Process Efficiency Evaluation	**Sewage Age Improvement**
Sediment Modeling and Impact Analysis	**Sensitivity Analysis**
Alternative Analyses	**Dynamic Loading Comparison**
Flow Monitoring Design	**Model Calibration**
Operator Training	**Capital Improvement Program Development**
Capital Budgeting	**Impact Fees Determination**
System Expansion	**New System Design**
System Layout Mapping	**Dynamic Profiling**
Project Management	**Stormwater Modeling**

Here is an example of how the InfoSewer Interface looks (the YourTown Example)



Comprehensive Sewer Collection Systems Analysis PDF File

[**Comprehensive Sewer Collection System Analysis**]

Click on the Link above to View the PDF File!

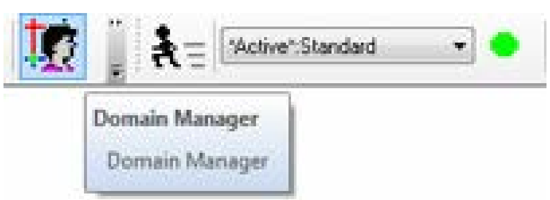
Welcome to the Innovyze Help Library

Welcome to the help library for the Innovyze software platform for engineering professionals for Water, Stormwater and Sewer modeling. The help file is organized in by topic sections sections to help you understand the GUI, engine and the many tools available to make the best and fastest model customized for your modeling needs. The Innovyze Help Libraries have the following sections:

• **Welcome to Innovyze Help for a Software Product**	• **Database Management**	• **Working with the Scenario Manager**	• **Reference Information**
• **What's New in the Product (Current Feature and Past History)**	• **Data Exchange**	• **Working With Projects**	• **Application Guides**
• **Press Releases for the Product (History)**	• **Utilities**	• **Working with the Run Manager**	• **User Guide**
• **Product Pages for**	• **Working with**	• **Working With**	• **Troubleshooting and Contact**

Innovyze Products similar to this Product	Element Data	Graphs and Reports	Information
• Getting Started for this Product for New and Returning Users	• Working with a Map	• Printing and Presentation	• FAQ's and Blogs
• User Interface for this Product	• Working with Domains and Facilities	• Theory	• Extensive Links to Innovyze Blogs, Forums, Support

Note by hovering the pointer over buttons or menu items in the system, you can access ToolTips for each menu tool. Some ToolTips provide access to a related topic in the help system, which are accessed by pressing F1.



Basic Files in InfoSewer

Each Project Contains:

- Network Map - Provides a visual schematic of the system and allows you to spatial reference your project.
- Feature Attributes - Element Information
- Modeling Parameters - Hydraulic Parameters Defining the Current Model Run
- Model Solution and Results - Static and Dynamic Results
- Model Scenarios - Contains all model scenarios

Project Storage Structure:

- Network Schematic - Stored as *.MXD File
- Database Tables and Modeling Files - Stored as *.IEDB Folder
- One directory containing model results - *.OUT Folder created after a successful Run
- Each Scenario has the INP, Text RPT files and OUT binary files

InfoSewer™

Modeling Made Easy

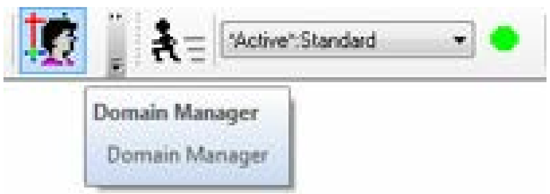
Welcome to the Innovyze Help Library

Welcome to the help library for the Innovyze software platform for engineering professionals for Water, Stormwater and Sewer modeling. The help file is organized in by topic sections sections to help you understand the GUI, engine and the many tools available to make the best and fastest model customized for your modeling needs. The Innovyze Help Libraries have the following sections:

Welcome to Innovyze Help for a Software Product	Database Management	Working with the Scenario Manager	Reference Information
What's New in the Product (Current Feature and Past History)	Data Exchange	Working With Projects	Application Guides
Press Releases for the Product (History)	Utilities	Working with the Run Manager	User Guide
Product Pages for Innovyze Products similar to this Product	Working with Element Data	Working With Graphs and Reports	Troubleshooting and Contact Information
Getting Started for this Product	Working with a Map	Printing and Presentation	FAQ's and Blogs

for New and Returning Users			
• **User Interface for this Product**	• **Working with Domains and Facilities**	• **Theory**	• **Extensive Links to Innovyze Blogs, Forums, Support**

Note by hovering the pointer over buttons or menu items in the system, you can access ToolTips for each menu tool. Some ToolTips provide access to a related topic in the help system, which are accessed by pressing F1.



What's New In InfoSewer

Innovyze is pleased to once again offer many great upgrades to our world-class Sewer modeling software. InfoSewer 7.6 SP1 brings you several advanced features and reiterates our dedication to bring you the best and most useful wet infrastructure modeling products in the world.

Click on the links below to see more information about each upgrade:

InfoSewer 7.6 SP1 Update 20

1. Several components were recompiled to prevent various database errors in the latest Windows Operating System. The recompiling uses Visual Studio 2010 and the changes are restricted to the GUI or UX.

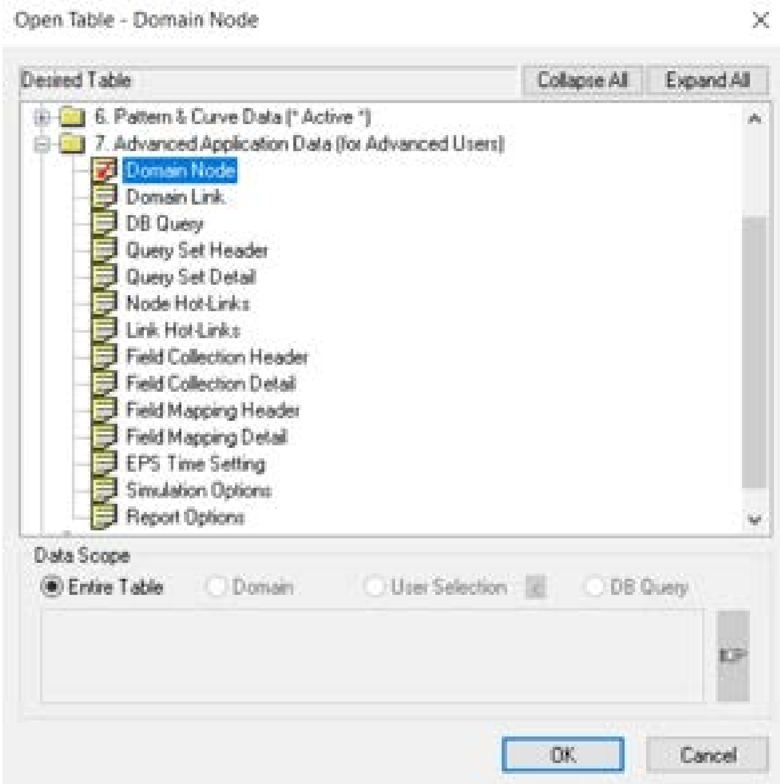
2. Fix to the Output Relate to align the columns (columns were offset by 2)

InfoSewer 7.6 SP1 Update 19

1. Bug fix to the Water Age Calculations
2. Updated Steady State and EPS Routing Help File
3. More Report File Debug Statements for the Advanced Force Main Solution
4. Added more information to the Warning Troubleshooting Section

InfoSewer 7.6 SP1 Update 18

1. Domain Nodes and Links were added to the DB Table
2. Installs now on ArcGIS 10 to 10.8
3. More Report File Debug Statements for the Advanced Force Main Solution



InfoSewer 7.6 SP1 Update 17

1. Installs now on ArcGIS 10 to 10.7

InfoSewer 7.6 SP1 Update 16

1. Improved the functioning of data associated with the Storm Tab between InfoSewer and InfoSewer Pro. InfoSewer without Pro now screens out non-applicable Storm based Pattern/Curve data to avoid porting PRO data into the Base version

2. Changed the engine so that 1. Rainfall curve and unit hydrograph pattern data are ignored if storm analysis flag is off. 2. Error message is given if a node is not associated with rainfall curve or unit hydrograph pattern AND no default is set. The results are the same for Pro Storm Analysis off and Non Pro.

3. Further improvements to the missing data issue for InfoSewer Pro Hydrographs and Hyetographs Error 1084: missing default unit hydrograph for the node: MH-307KK21 Error 1084: missing default unit hydrograph for the node: MH-913JJ21 Error 1085: missing default hyetograph for the node: MH-913JJ21

InfoSewer 7.6 SP1 Update 15

- A nice new engine update to allow less than 1 minute time steps for the report and simulation engine. You can now use 1 second time steps for the network.
- Enlarged import manager dialog size
- Fix to SI unit design of existing systems in InfoSewer with -9999 for link inverts
- Newly remade Help files sans H2OMap Sewer and based on the InfoSewer SA Ribbon Interface
- Change help file name to "Sewer.CHM".

Pre 2018 Changes

- Improvements to the Steady State Solution for the Advanced Force Main Solution
- New in Q1 2017 is the ability to export to a CSV file the Node and Link Reports so you can see the whole simulation reports at the same
- 32 Character Node and Link ID's - January 2016
- Greatly Expanded Help File from SWMM and H2OCal in January 2016
- Advanced Forcemain Network Support for more complicated upstream and downstream Force Main Connections (see Figure 1)
- New Help file format in 2014/2015
- Plan Profile Plotting of the Input Network
- Advanced Node and Link labeling for HGL Plots
- A complete list of node, link graphics for all Output Attribute Browser Variables
- Better memory allocation for long simulation and enhanced memory allocation for plot with many data points
- Expanded Output Manager Tabular Reports for EPS Simulations
- Expanded Warning and Error messages in the text report file
- Enhanced water quality routing through force mains, pumps and wet wells
- Enhanced export to H2OMAP SWMM
- Enhanced simulation of small hyetograph time steps for hydrographs
- Expanded output for the Design Feature of H2OMAP Sewer
- Improvements to the DB Editor for Import of GIS and OBDC data
- The ability to run longer simulations with shorter report time steps
- Enhancements the pump allocation routine for Steady State and EPS runs

Changes and Improvements for InfoSewer 7.6 SP 1 Update 14 (English)

Added an improved algorithm and provided two options for loop finder. https://en.wikipedia.org/wiki/Tarjan's_strongly_connected_components_algorithm (Updated file: HeXtn4Pkg.dll, Release date: 6/21/2017 4:30:33 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 13 (English)

Added support for ArcGIS 10.5

Changes and Improvements for InfoSewer 7.6 SP 1 Update 12 (English)

Fixed the missing Compare Report button for the tabular reports. (Updated file: HeUI2Pkg.dll, Release date: 4/7/2017 9:26:14 AM)

New name for Load Allocator so it is different than the SWMM DWF Allocators (Updated file: LoadAllocator.chm, Release date: 3/22/2017 6:48:03 PM)

Update for 3/21/2017 (Updated file: SEWER.CHM, Release date: 3/21/2017 3:49:53 AM)

Added a button for tabular report for exporting CSV files from the output report manager. Added a prompt message box for opening exported file by system default application (like Excel). (Updated file: HeUI2Pkg.dll, , Release date: 3/20/2017 4:13:39 PM)

1. A fix to the issue of extra manhole loading DB Table loading to an inactive node. The extra DB loading table was not well known to users and caused a few support cases when the model did not run. 2. A fix to the problem of users initializing older models with new versions of Sewer and older Output Relates. Users should not have to delete the OUT folder. (Updated file: HeDataPkg.dll, Release date: 3/6/2017 3:51:54 PM)

Fixed a memory leak issue in advanced force main analysis that can crash the engine. 1. Calculates the pump speed for inflow control pump in SS, with advanced FM analysis 2. Correctly sums coverage flow from 2 to many upstream gravity mains to a wet well system for the Steady State Solution. 3. Correctly calculates the flow with 2 or more Pumps between the same two Nodes for the Steady State Solution. The above changes do not apply to the EPS solution which was fine. (Updated file: HeEngPkg.dll, Release date: 1/21/2017 6:26:17 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 11 (English)

Pipe Profile Input HGL red icon to switch to the current simulation Run HGL now uses the current solution for EPS, Steady State or Design based on the InfoSewer output toolbar setting. (Updated file: HeUI2Pkg.dll, Release date: 9/27/2016 4:05:19 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 10

HeEngPkg.dll New Feature/Improvement Flow and Inflow control pump rules for normal and Advanced Force Main Analysis have been improved for Fixed Capacity pumps. The pump speed will always show 1.0 for fixed capacity pump. (Updated file: HeEngPkg.dll, Release date: 6/10/2016 5:31:17 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 9

Fixed an error from ArcGIS -- "The spatial index grid size is invalid". (Updated file: HeMapSrv10.dll,, Release date: 4/11/2016 3:25:29 PM)

Good Friday update for Arc GIS 10.4 along with more comparisons between Sewer, SWMM and ICM. (Updated file: InfoSewer.CHM, InfoLoadAllctr.chm, Release date: 3/25/2016 4:19:54 PM)

Leap Day update of all Help Files with further links to the Innovyze Blog Forum, updated equations, more about our Complex Hydrology Options, 2D and LID objects. They try to combine all learning styles which seem to manifold among our customers and perhaps impossible. The HTM files are ideal for making email notes, PDF additions or as starting points for blogs. Affected files include InfoSewer.CHM, InfoLoadAllctr.chm (Updated file: InfoSewer.CHM, InfoLoadAllctr.chm, Release date: 2/27/2016 5:04:42 PM)

Updated Help File so that it has the same level of Detail as SWMM and Water for Export Import Items. Updated Help File so that it has the same level of Detail as SWMM and Water for DB Tables due to User complaints. (Updated file: InfoSewer.CHM, Release date: 2/14/2016 6:48:59 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 8

32 Character ID Support was added to match the Engine. (Updated file: HeMapSrv10.dll, Release date: 2/5/2016 1:59:22 PM)

Help Files with further improvement Common tables, CSS across Water, Sewer and SWMM. (Updated file: InfoSewer.CHM, Release date: 2/5/2016 1:56:48 PM)

Support ID characters up to 32 for table reports. (Updated file: HeUI1Pkg.dll, Release date: 2/5/2016 11:23:53 AM)

Help Files with further integration to PB Press Releases, Sewer Icons for all Bullets, Integration on every page to www.innovyze.com and Sewer/SWMM MLK Day Help File Format along with further merging with the Water help files for common Innovyze Features. (Updated file: InfoSewer.CHM, Release date: 1/20/2016 12:25:06 PM)

Release 2016 splash image.. (Updated file: HeLoader.dll, Release date: 1/20/2016 11:51:12 AM)

Extended module copyright statement to cover 2016 (Updated file: HeUI2Pkg.dll, Release date: 1/14/2016 6:28:03 PM)

New help file based on H2omaap water design and more connections to ICM and InfoSWMM along with updated images and user guide and an intereconnected User Guides based on Nicklow, J.W., Boulos, P.F., and Muleta, M.K. (2004). Comprehensive Sewer Collection Systems Analysis Handbook for Engineers and Planners. MWH Soft. Pasadena, California, USA, 287 p. Nicklow, J.W., Boulos, P.F., and Muleta, M.K. (2006). Comprehensive Urban Hydrologic Modeling Handbook for Engineers and Planners. MWH Soft. Pasadena, California, USA. (Updated file: InfoSewer.chm, Release date: 12/11/2015 1:18:10 PM)

Increased the length of ID from 21 to 32 (Updated file: HeEngPkg.dll, Release date: 12/10/2015 3:33:58 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 7

Fix to bypass user import of the wrong Pipe Type. Sewer normally uses Type 1 and Type 0 or GM and FM's (Updated file: HeMapSrv10.dll, Release date: 11/4/2015 6:02:06 PM)

A fix to the Non Circular Conduit tool to allow the dialog to open in Arc GIS 10.3 (Updated file: HeMapSrv10.dll, Release date: 10/30/2015 5:02:36 PM)

Updated the combined Sewer help files with more connections to SWMM and Innovyze in general This is a combined Help file for InfoSewer and H2OMap Sewer as both program share the same engine, the same DB or Database File format, the same Attribute Browser, the same output reports and graphs and many of the same menu commands and tools. The obvious difference is that InfoSewer has many more spatial tools as it is an extension inside Arc GIS. (Updated file: InfoSewer.CHM, Release date: 10/19/2015 10:29:43 AM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 6

New Help file with User Guide, On Line Video's and FAQ Improvements (Updated file: InfoLoadAllctr.chm, Release date: 8/10/2015 9:15:57 PM)

New Help file with User Guide, On Line Video's and FAQ Improvements along with Innovyze St Venant Comparisons (Updated file: InfoSewer.chm, Release date: 8/10/2015 7:02:55 PM)

Updated the Innovyze Application Manager to eliminate the issues caused by ProgID changes from InfoWater. You can now switch smoothly from InfoSWWMM to and from InfoWater and InfoSewer (Updated file: HaAppMgr10.dll, Release date: 7/16/2015 10:47:52 AM)

Release number 50014. Extends advanced focemain analysis to steady state analysis. Default design manning coefficient in engine changed from 0.001 to 0.013. (Updated file: HeEngPkg.dll, Release date: 7/6/2015 2:09:37 PM)

Per Feng, ihe default pipe manning coefficient for Sewer design is too small as 0.001. It should be 0.013. (Updated file: HeUI2Pkg.dll, Release date: 7/6/2015 1:26:45 PM)

New Report Type for EPS Run : - introduced the "Mass Balance" report - requires updated HeEngPkg.dll and re-run of the model New Output Unit Manager Variable : - introduced the new "Flow Volume" unit for the above new report type - updated version of Output Unit Manager to cover this new output variable (Updated file: HeOutPkg.dll and HeUI3Pkg.dll, Release date: 6/19/2015 3:16:20 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 5

For the past couple years, we suffered mysterious behaviors that global application settings (windows positions, recent accessed projects, and so on) did not get properly saved and restored. The file access glitch was finally confirmed and remedied today. (Updated file: HeUtilPkg.dll, Release date: 5/7/2015 5:22:04 PM)

Re-ordered fields of pump information table. (Updated file: HeDBSrv.dll, Release date: 4/30/2015 4:12:56 PM)

1) applied ID size upgrade AGAIN 2) ensured common fields are consistent across all facility information tables 3) corrected the build-in definition for the number of the reserved fields. (Updated file: HeUtilPkg.dll, HeDBBPkg.dll, Release date: 4/30/2015 2:34:52 PM)

GIS Gateway: 1. New comparison function to compare data of InfoSWMM and GIS. 2. New logging function to allow users to track the data changes. 3. Allows users to save updated data to the selection sets. (Updated file: HeTools.dll, HeMapSrv10.dll, Release date: 3/18/2015 2:51:31 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 4

Improved sewer project loading error report (Updated file: HeUtilPkg.dll, HeUI2Pkg.dll, Release date: 1/21/2015 5:23:44 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 3

When auto-length is on, pipe length calculation will save to all scenarios. Features affected: 1. Insert Node to Pipe, 2. Redraw Pipe 3. Map2DB (Updated file: HeHxfSvr.dll, HeUI2Pkg.dll, Release date: 11/7/2014 12:36:04 AM)

Cross Product Help File Changes to have the User Guide as a CHM Topic, new FAQ""s, Tutorials, Direct Linkages to the Innovyze Forum, Blog, Twitter and www.innovyze.com as well as all new images, uniform fonts and expanded text sections - DWF Allocator (Updated file: InfoLoadAllctr.chm, Release date: 10/26/2014 7:51:01 AM)

New Splash Screen for InfoSewer. (Updated file: HeLoader.dll, Release date: 10/24/2014 1:45:38 PM)

File Geodatabase sources sometimes became disconnected. Now it has been fixed. (Updated file: HeMapSrv10.dll, Release date: 10/24/2014 12:13:41 PM)

GIS Gateway could not import file geodatabase properly. Now it has been fixed. (Updated file: HeMapSrv10.dll, Release date: 10/21/2014 12:48:26 PM)

Improvement to the HGL input and output HGL graphs. (Updated file: HeUI2Pkg.dll, Release date: 10/13/2014 4:31:32 PM)

A fix for EPS simulation so that consecutive adverse slope links correctly compute the flow. (Updated file: HeEngPkg.dll, Release date: 10/1/2014 4:21:29 AM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 2

A change to the Reserve Capacity Profile Graph so that instead of plotting mid link to mid link it graphs from Up Node to Down Node. (Updated file: HeOutPkg.dll, Release date: 8/20/2014 4:10:40 PM)

Forcemain and Pump's active status sometimes showed incorrectly in Activate Domain function. Now it has been fixed. (Updated file: HeMapSrv10.dll, Release date: 6/16/2014 3:17:31 PM)

Changes and Improvements for InfoSewer 7.6 SP 1 Update 1

A fix to the Design feature for New and Existing Networks. The program will now use the minimum of the link and manhole DS Invert Elevations for the case of bifurcating links. The previous version of InfoSewer did not set the Link US Invert to the Manhole Designed US Invert. (Updated file: HeEngPkg.dll, Release date: 6/9/2014 1:34:34 PM)

A change to the units of the variable overflow in closed gravity main links. The units are now flow units and overflow is listed as Maximum Flow minus the Full flow for the link. InfoSewer does not have overflow in closed links, this parameter shows how full the link is at any particular time based on the current flow and Qfull. This change also applies to the Gravity Main Range Report. (Updated file: HeOutPkg.dll, Release date: 5/29/2014 4:12:19 PM)

Manhole loading reports and graphs now have consistent parameter names for base flow. (Updated file: HeOutPkg.dll, Release date: 5/6/2014 2:17:51 PM)

Support updated copyright statement (Updated file: HeUI2Pkg.dll, SOFTWARE COPYRIGHT.rtf, Release date: 4/28/2014 6:39:16 PM)

InfoSewer was not showing the first output field in Map Display for Force Mains. It is now fixed. (Updated file: HeMapSrv10.dll, Release date: 3/31/2014 3:48:50 PM)

Fixed Thiessen polygon issue caused by thiessen polygon engine file name changes. (Updated file: HeAllocSrv10.dll, Release date: 3/31/2014 3:37:12 PM)

Changes and Improvements for InfoSewer 7.6 SP 1

Installation: All of the InfoSWMM configuration setting files in the C:\Program Files\InfoSWMM\Bin and C:\Documents and Settings\All Users\Application Data\MWH Soft have been moved to C:\Documents and Settings\All Users\Application Data\Innovyze directory (Updated file: All program files, Release date: 3/25/2014 5:16:51 PM)

There is now a case-insensitive comparison for Node IDs in the Sewer engine input file creation for those cases where the Node ID's had a lower case. (Updated file: HeEngPkg.dll, Release date: 3/21/2014 4:02:01 PM)

Changes and Improvements for InfoSewer 7.6 Update 10

A map error might occur after a very old InfoSewer project is upgraded. Not it has been fixed. (Updated file: HeMapSrv10.dll, Release date: 2/12/2014 7:55:32 PM)

Modify error message for invalid forcemain configuration. "invalid split force-mains without advanced forcemain network support" (Updated file: HeEngPkg.dll, Release date: 2/6/2014 12:38:06 PM)

1. Zoom to a node with % of full extent instead of a zoom factor of the geometry extent. 2. Append Nodes: fixed an issue to append nodes on pump elements. 3. Fill Link Connectivity: now able to fill connectivity for all of element types. (Updated file: HeMapSrv10.dll, Release date: 1/7/2014 11:47:51 AM)

Fixed a typo for Pump Usage in the EPS Graphs. (Updated file: HeOutPkg.dll, Release date: 12/30/2013 11:07:23 AM)

Changes and Improvements for InfoSewer 7.6 Update 9

Added Node Maximum Depth, Maximum Unfilled Depth and Maximum Surcharge Depth as used in the Loading Manhole Range Report as new Map Display Active Variables. (Updated file: HeOutPkg.dll, Release date: 12/13/2013 3:40:43 PM)

For large models exceeding 25000 nodes with RDII Storm Flow of more than a week, the temporary hydraulic file exceeded 2GG. It was changed to allow 64K integer file locations and now much larger Stormwater Models can be simulated for longer than normal EPS simulations. (Updated file: HeEngPkg.dll, Release date: 11/14/2013 3:13:06 PM)

Apply to Manhole in Domain was not working. Now it has been fixed. (Updated file: HeAllocSrv10.dll, Release date: 10/25/2013 5:10:21 PM)

Changes and Improvements for InfoSewer 7.6 Update 8

Added the intial gravity main storage to the Mass Balance report to have a better estimate of the overall continuity error for large models [Mass Balance Calculation] Total external inflow: 13314364.169 (cubic feet) Total storm inflow (part of external inflow): 0.000 (cubic feet) Total extern outflow: 13204576.715 (cubic feet) Total flooding out of open conduits: 0.000 (cubic feet) Initial Wet Well Storage: 6071.164 (cubic feet) Final Wet Well Storage: 6073.972 (cubic feet) Wet Well Flooding Volume: 0.000 (cubic feet) Init Gravity Main Storage: 68.916 (cubic feet) Final Gravity Main Storage: 134360.889 (cubic feet) Mass Balance Check: 0.184 (%) (Updated file: HeEngPkg.dll, Release date: 10/4/2013 11:32:17 AM)

A fix to the flag for EPS or Extended Period Simulation Based Design so that the flag is remembered from one Run Manager Session to another. (Updated file: HeUI2Pkg.dll, Release date: 9/25/2013 12:34:40 PM)

The maximum manhole unfilled depth and surcharge depth are now added to the range loading manhole reports so customers can more easily find the worst surcharge conditions during an EPS simulation. (Updated file: HeEngPkg.dll, HeOutPkg.dll, Release date: 9/17/2013 5:44:13 PM)

New Feature: local search for Closest Pipe method (Updated file: HeAllocSrv10.dll, Release date: 9/10/2013 3:12:44 PM)

Changes and Improvements for InfoSewer 7.6 Update 7

Pump control by wet well volume may cause a very small variable time step which slows the simulation, This has now been fixed. (Updated file: HeEngPkg.dll, Release date: 8/15/2013 10:00:42 AM)

Some elements' active flags might become incorrect after scenario switching. Now it has been fixed. (Updated file: HeMapSrv10.dll, Release date: 8/7/2013 3:46:22 PM)

Improved pollutant routing through pumps, force mains and chamber nodes for the case where the chamber node is a quality source. Pollutant is one water quality option in addition to Trace, Water Age, H2S, Corrosion and Sediment. (Updated file: HeEngPkg.dll, Release date: 8/6/2013 4:45:24 PM)

- support region-dependent real number format for DB tables (Updated file: HeUtilPkg.dll, Release date: 8/1/2013 4:46:17 PM)

Changes and Improvements for InfoSewer 7.6 Update 6

Sewer engine fix to prevent negative Wet Well volumes for large pump flows. Sewer requires a combination Gravity Main, Wet Well. Pump, Chamber Node and Force Main connected upstream and downstream to a pump. (Updated file: HeEngPkg.dll,, Release date: 7/5/2013 12:11:19 PM)

Changes and Improvements for InfoSewer 7.6 Update 5

Improvments to the way overflow is calculated from gravity mains and wet wells 1. The new FM solution can now be used for just one FM as well as multiple FM's with upstream and downstrem connections. 2. FM solver is modified to report wet well overflow by time steps. 3. Non FM solver codes are modified to calculate the wet well overflow consistently with the gravity main overflow. 4. Gravity Main Overflow Graphs now show overflow volume per report time step. (Updated file: HeEngPkg.dll, Release date: 6/19/2013 5:30:13 PM)

Two fixes and improvments to the engine of InfoSewer: 1. An error message that states that Manning"s coefficient is used instead of the HW Force Main coefficient for a FM is not repeated for all FM links, 2. The new and vastly improved FM solution for EPS simulation when accidently used for a Steady State Solution does not cause the simulation to freeze. The new FM solution is not applicable to the Steady State Solution. (Updated file: HeEngPkg.dll, Release date: 6/7/2013 9:38:11 AM)

Skips short pipes on pipe split candidate(s) searching. (Updated file: HeMapSrv10.dll, Release date: 4/15/2013 12:30:23 PM)

Changes and Improvements for InfoSewer 7.6 Update 4

HGL Advanced Labeling for Links now shows the Downstream Invert as well as the Upstream Invert Elevation (Updated file: HeOutPkg.dll, :Release date: 1/31/2013 5:00:08 PM)

Changes and Improvements for InfoSewer 7.6 Update 3

Population field could not be populated in Allocation Manager dialog. Now it has been fixed. (Updated file: HeAllocSr10.dll, Release date: 1/11/2013 12:03:42 PM)

Could not create new Output Relate properly. Now it has been fixed. (Updated file: HeMapSrv10.dll, Release date: 1/11/2013 12:00:32 PM)

Fixed incorrect icon reference on pipe and pump tabs in Scenario Comparison Report. (Updated file: HeUI2Pkg.dll, : Release date: 11/26/2012 6:24:05 PM)

Scenario comparison report was frozen due to incorrect reference to Simulation Time field. It is now fixed. (Updated file: HeUI2Pkg.dll, Release date: 11/23/2012 3:55:18 PM)

Now includes both “Manhole Sealing” and “Outlet Discharge HGL” tables with the Manhole data set. (Updated file: HeUtikPkg.dll, Release date: 11/16/2012 12:58:24 PM)

Changes and Improvements for InfoSewer 7.6 Update 2

Users could not draw a forceman from a chamber manhole. Now it has been fixed. (Updated file: HeGUI10.dll, Release date: 10/30/2012 3:00:45 PM)

Domain to ArcMap Selection command was not working properly on table joins. The issue was caused by the new map display engine. Now it has been fixed. (Updated file: HeMapSrv10.dll, Release date: 10/25/2012 4:45:17 PM)

InfoSewer Compiled Help File modified for deprecated features such as Customized Report. (Updated file: InfoSewer.chm, Release date: 10/16/2012 11:02:18 AM)

Changes and Improvements for InfoSewer 7.6 Update 1

Release revised splash image with 2012 copyright declaration. (Updated file: HeLoader.dll, Release date: 10/9/2012 5:49:32 PM)

Removed the inapplicable "customized report" pull-down menu from Report Manager. Customized Report was never implemented in Sewer products. The user can use Query Report with output relate(s) to achieve similar function. (Updated file: HeUI2Pkg.dll, Release date: 10/9/2012 12:16:49 PM)

Fixed Steady-State run Loading Manhole relate data field type mismatch problem. (Updated file: HeOutPkg.dll, Release date: 10/5/2012 11:34:36 AM)

A fix for the Mapped total flow for a link downstream of a flow splitter when the downstream link is an adverse sloped pipe. An adverse slope link flows full in InfoSewer and previously it was showing the incoming node flow and not the flow split flow. The range report, graphical display of flow and tabular report of flow for both EPS and Steady State simulations are now consistent. (Updated file: HeEngPkg.dll, Release date: 8/9/2012 5:56:17 PM)

ArcGIS version history

Version	Released
8.0	1999-12-27[4]
8.0.1	2000-01-13[5]
8.1	2001-05-01[6]
8.2	2002-05-10[7]
8.3	2003-02-10[8]
9.0	2004-05-11[9]
9.1	2005-05-25[10]
9.2	2006-11-14[11]
9.3	2008-06-25[12]
9.3.1	2009-04-28[13]
10.0	2010-06-29[14]
10.1	2012-06-11[15]
10.2	2013-07-30[16]
10.2.1	2014-01-07[17]
10.2.2	2014-04-15[18]
10.3	2014-12-10[19]
10.3.1	2015-05-13[20]
10.4	2016-02-18[21]
10.4.1	2016-05-31[22]
10.5	2016-12-15[23]

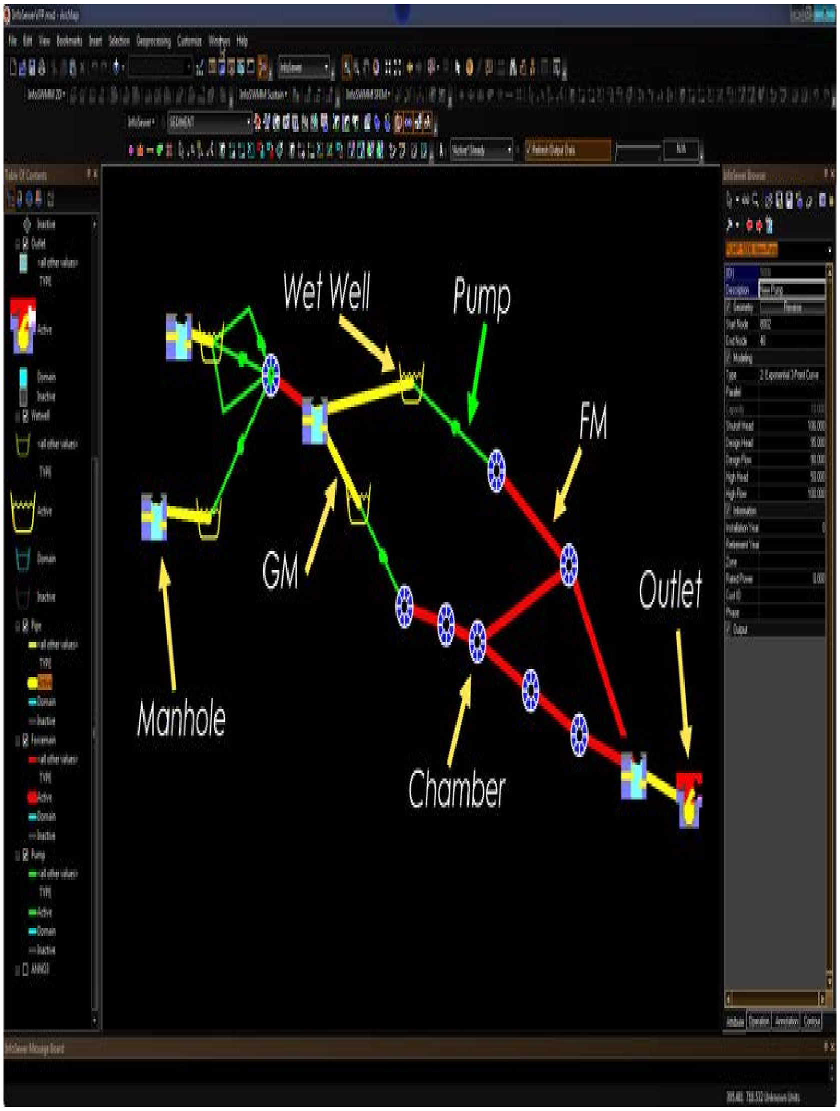


Figure 1. Rules for modeling force mains (FM), Gravity Mains (GM) and Pumps in InfoSewer and H2OMap Sewer

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer Menu

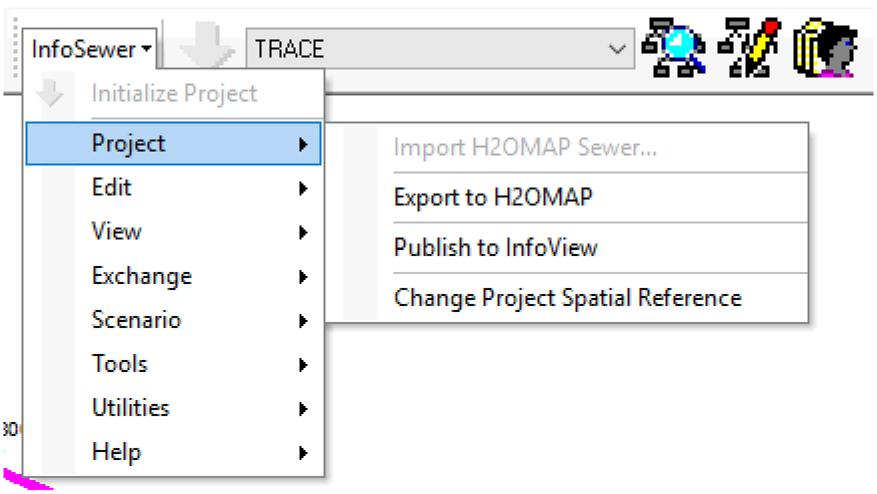
Descriptions of All Project Menu commands in InfoSewer.

Import H2OMap Sewer - H2OMap Sewer has been deprecated by Innovyze as of 2018.

Publish to InfoView - This command helps you to publish an InfoSewer Project to InfoView.

Export to H2OMap - Save to H2OMap Sewer (deprecated)

Change Project Spatial Reference - Change the Spatial Reference of the InfoSewer Map



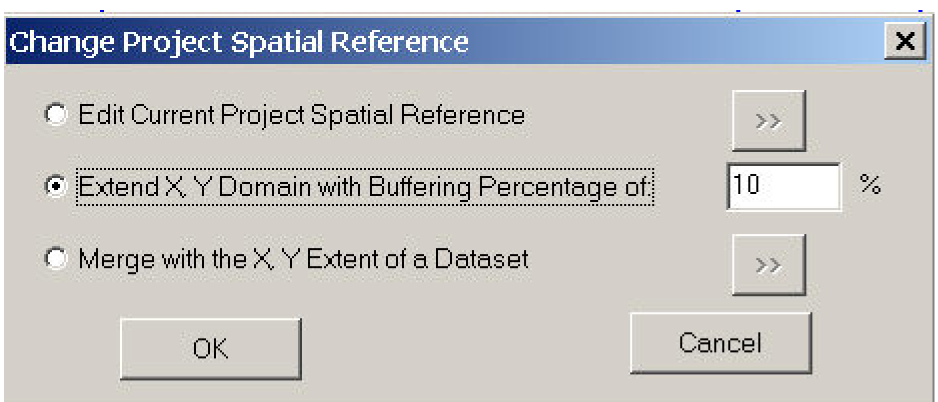
Change Project Spatial Reference

The spatial reference of a dataset in geodatabase can not be further modified once created in ArcGIS and new features can not be inserted if it's coordinates are beyond X, Y domain of the spatial reference. InfoSewer provides a very powerful tool to overcome this limitation of ArcGIS.

In InfoSewer the spatial reference of a project can be modified by either editing current project's spatial reference, extending the X, Y domain with buffering percentage, or merging with the X, Y extent of a dataset.

To launch the Change Project Spatial Reference dialog box, click on the InfoSewer Button 

, browse to the Project menu and select the Change Project Spatial Reference option.

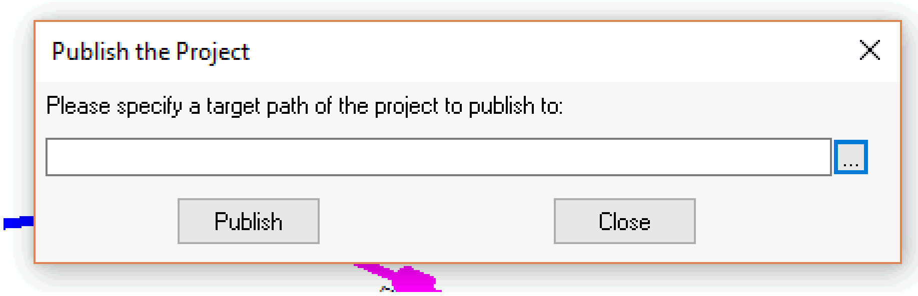


- Edit Current Project Spatial Reference: Use this to modify/edit the spatial reference of a current project.
- Extend X,Y Domain with Buffering Percentage: Use this to modify the spatial reference of a project by extending the X,Y domain with buffering percentage.
- Merge with the X,Y Extent of a Dataset: Use this to modify the spatial reference of a project by merging with the X,Y extent of a dataset.
- OK: Click on OK to accept changes to the spatial reference coordinates and close out of the dialog box.

- Cancel: Use this to cancel the selection and close out of the dialog box. Any changes and/or selections made will not be reflected in the project.

Publish to InfoView

This command helps you to publish an InfoSewer Project to InfoView.



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

User Interface

The InfoSewer user interface has been created in such a fashion as to facilitate model creation and review output results. To view each component of the user interface, please select from any of the following:

- Drawing Area
- ArcMap pull-down menus
- ArcMap Toolbars
- InfoSewer Toolbars & Icons
- InfoSewer Control Center
- InfoSewer Browser
- Attribute Browser

Element Attribute Browser

The Attribute Browser dialog box allows the user to view and edit the database information related to a selected link or node facility. Shown below, the dialog box contains all information related to links and nodes when selected by the user.

Unless "grayed" out, any of the records under the section headers (Geometry, Modeling, Information) are editable at any time. While using the Attribute Browser, click the red check box next to the title to expand and contract the section headers. The Output data section is available only after a successful model run. Additionally other features allow you to locate (search) elements, create default values and assign specific values through the Tools menu.

To learn more about the multitude of icons related to the Attribute Browser, click here or select any of the icons below.

The Link Element

The link tab allows the user to see the database information related to a graphically selected link.

Click on any of the drop down links below to learn more about the database information contained in the Attribute Browser.

[Pipe Data]

[Pipe ID]

- **ID** - The unique InfoSewer identification of the pipe.

- **Description** - A user defined description of the pipe.

[Geometry]

- **Start Node** - The upstream node for the pipe.

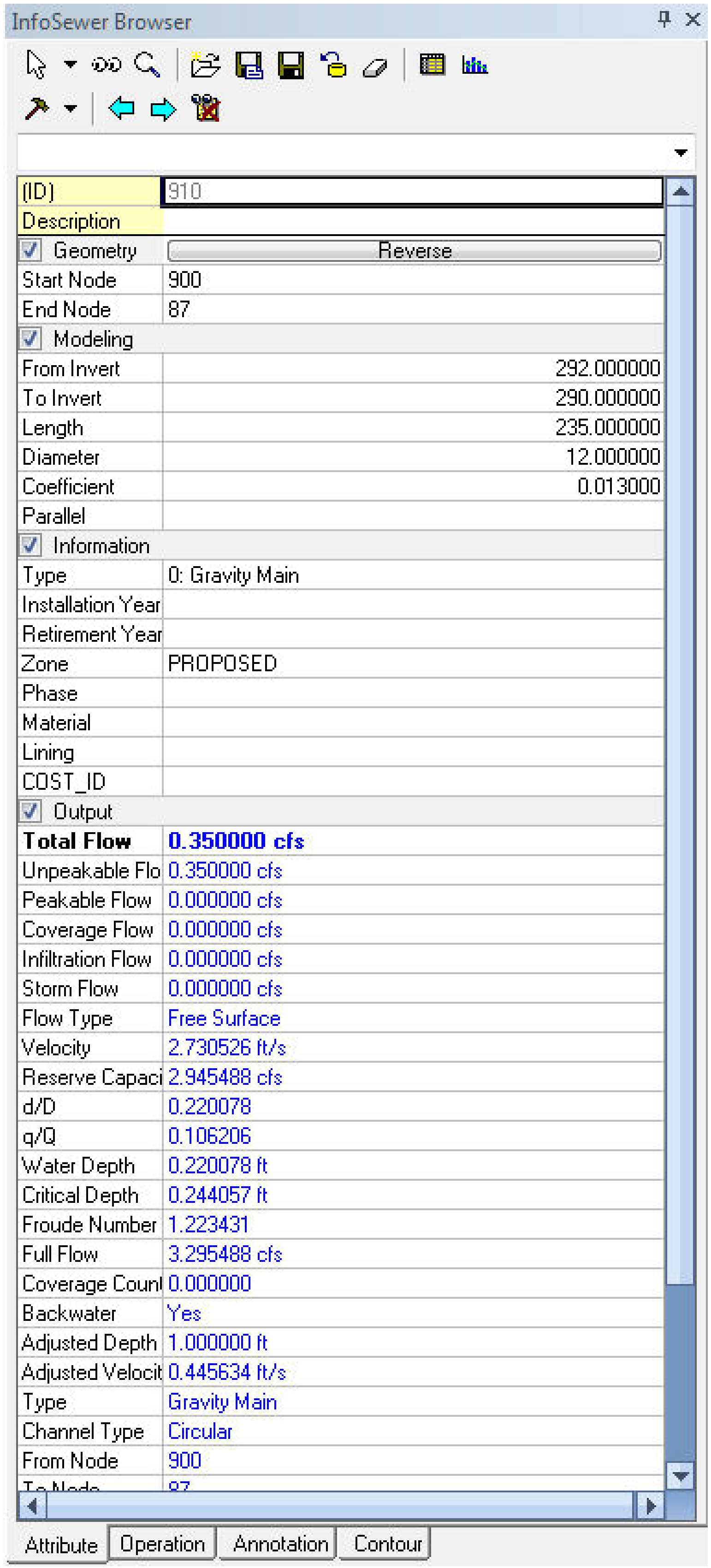
End Node - The downstream node for the pipe.

[Modeling]

- **From Invert** - The upstream invert elevation of the pipe. Units are feet or meters.

- **To Invert** - The downstream invert elevation of the pipe. Units are feet or meters.

- **Length** - The length of the pipe. Created automatically if Tools -> Preferences ->



Operation -> Auto Length Calculation is checked on. Units are feet or meters.

- **Diameter** - The diameter of the channel (for circular pipes) as specified by the user. For non-circular conduits, required input parameters may be one or more of channel depth, channel width, and side slopes. Units are inches or millimeters.

- **Coefficient** - The Manning (gravity flow) or Hazen-Williams (force main) roughness coefficient as assigned by the user.

- **Parallel** - The number of pipes in parallel with the existing pipe. Each parallel pipe is given the exact same parameters as the existing pipe. Unitless.

Output

- **Output** - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a pipe output report.

Pump Data

Pump ID

- **ID** - The unique InfoSewer identification of the pump.

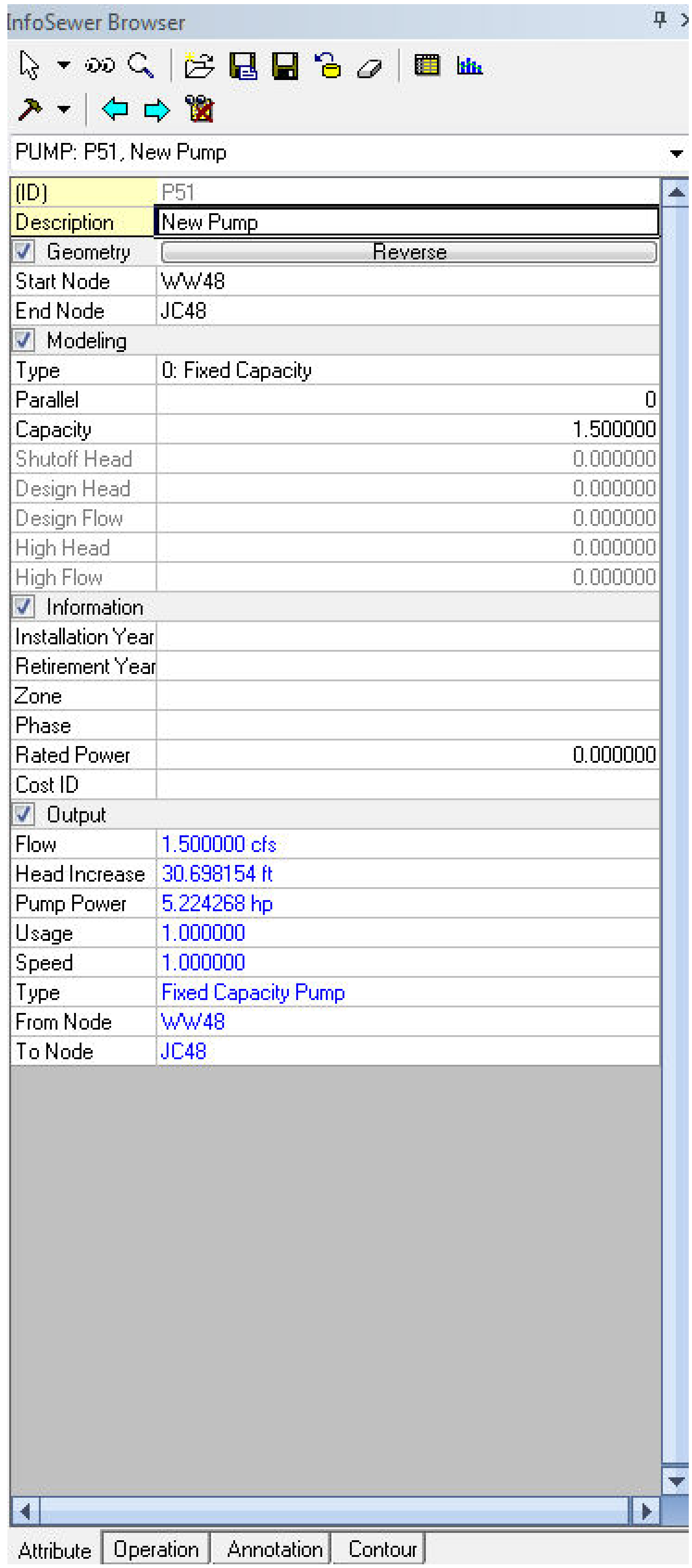
Description - A user defined description of the pump.

Geometry

- **Start Node** - The upstream node for the pump.
- **End Node** - The downstream node for the pump.

Modeling

- **Type** - The type of pump to be used by InfoSewer. To change the type, merely click in the drop down box



and select from the list. Click here to learn more about each type and the fields required.

- **Parallel** - The number of pumps in parallel to the existing pump. Parallel pumps are given the exact specifications of the existing pump. Unitless.

Click here to learn more about the types of pumps available in H2OMAP Sewer.

Information

- **Installation Year** - The year the pump was installed.

- **Retirement Year** - The year the pump is expected to be retired from service.

- **Zone** - The service area of the selected pump.

- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to modeling a simulation.

- **Rated Power** - The rated horsepower of the pump in the field.
- **Cost ID** - A database field used to assign differing cost identifiers to differing facilities.
- **(Other User Created Fields)** - Any fields added by the user will appear at the end of the Information tab. Click here to learn how to add a field.

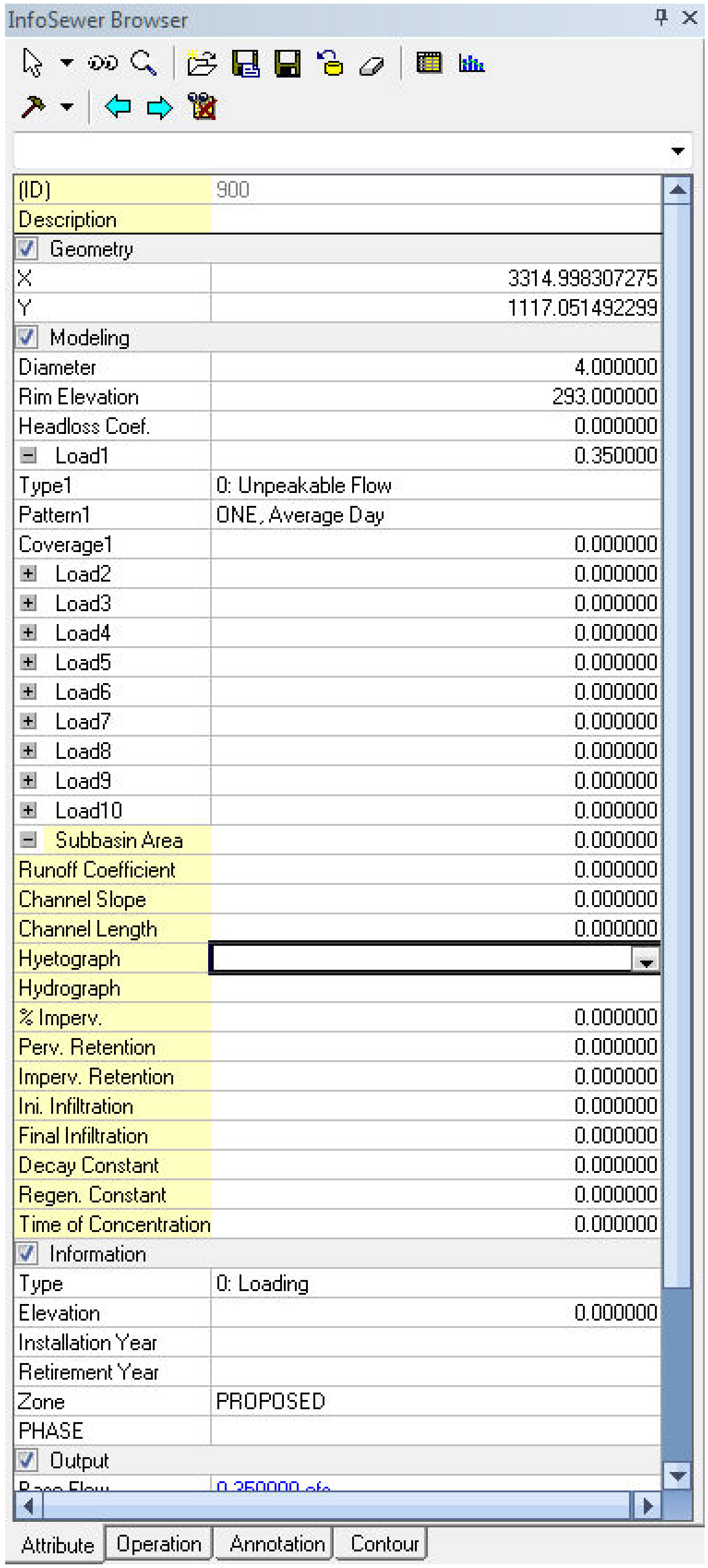
Output

- **Output** - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a pump output report.

The Node Element

The node tab allows the user to see the database information related to a graphically selected node.

[Manhole Data (Normal, Chamber, Outlet)]



[Node ID]

- **ID** - The unique H2OMAP Sewer identification of the manhole.

Description - A user defined description of the node.

[Geometry]

- **X** - The x-coordinate of the node.
- **Y** - The y-coordinate of the node.

[Modeling]

- **Diameter** - The internal diameter of the manhole. Feet or Meters.
- **Rim Elevation** - The elevation of the rim of the manhole as specified by the user (usually a value denoted above sea level). Units are feet or meters.
- **Headloss Coef** – Used to calculate headloss in the

manhole itself using the exit pipe velocity (see link for more details on Headloss Coefficients and manhole headloss)

- **Load1** - The base sewage loading demand at the manhole. Up to ten base loadings can be entered for a single manhole. The extra load option allows additional loadings to be added as needed. Units from the Run Manager.

- **Type1** - Specify whether the load is an unpeakable flow, an peakable base flow or a peakable coverage flow. Click here to learn more about each type.

- **Pattern1** - The user defined diurnal pattern for the manhole loading. Required for an EPS analysis.

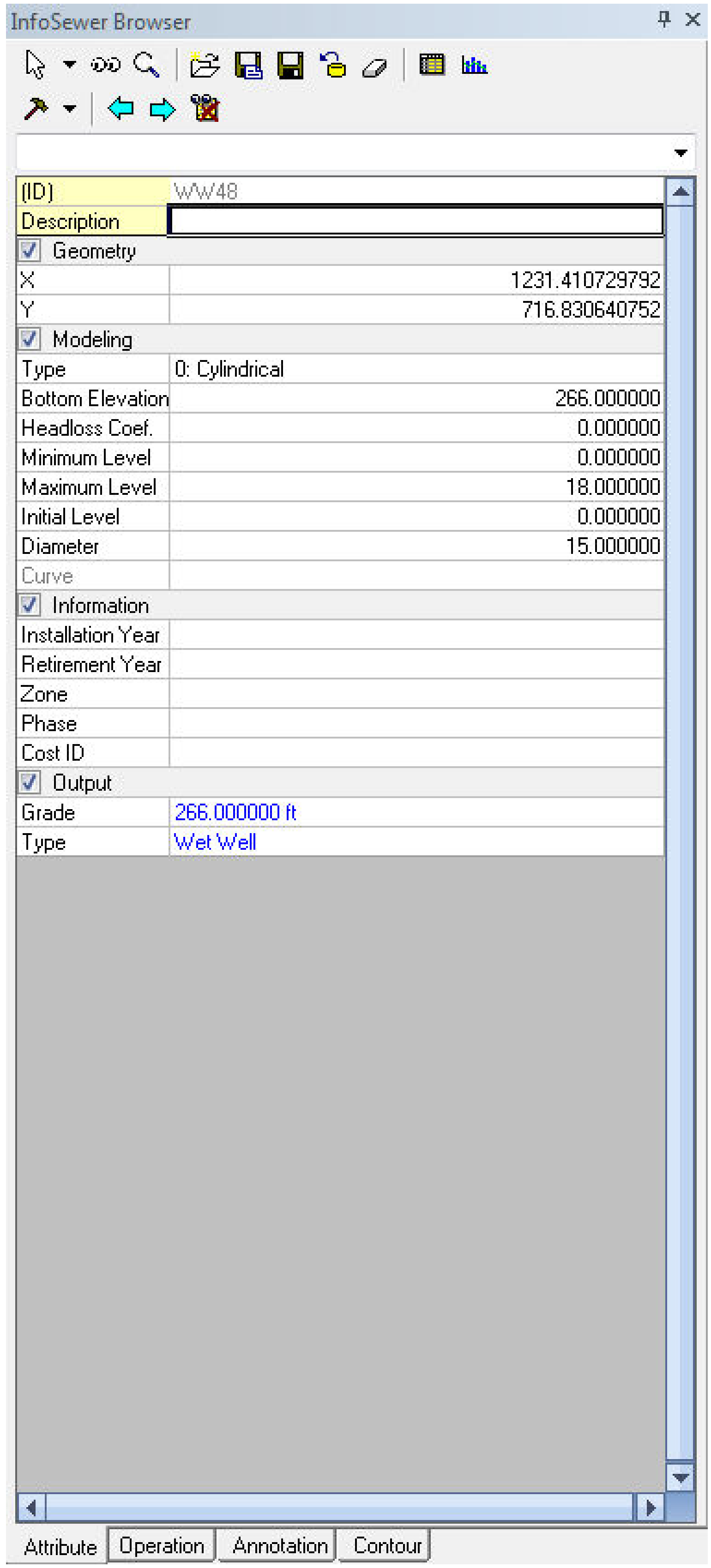
- **Coverage1** - The base population to be used during a peakable coverage flow determination.

- **Subbasin Area** - Area of the subwatershed that drains to the loading manhole. This variable is required for stormwater modeling.

- **Runoff Coefficient** - Ratio of storm runoff to rainfall volume. This variable is required during steady state simulation of stormwater.

- **Channel Slope** - Average slope of the longest flow channel for the subwatershed that drains to the loading manhole. This variable is required during steady state simulation of stormwater.

- **Channel Length** - Length of the longest flow path for the subbasin that drains to the manhole. This variable is required during steady state simulation of



stormwater. Units are feet or meters.

- **Hyetograph** - A plot of rainfall intensity versus time. This variable is required during dynamic simulation of stormwater.
- **Hydrograph** - Refers to a unit hydrograph pattern, and is required is required during dynamic simulation of stormwater.
- **% Impervious** : Percentage of the subwatershed that is covered with impervious land uses. This parameter is used to estimate depression storage losses.
- **Pervious Retention** : Refers to depression storage of the pervious portion of the subwatershed. The value should be given in feet for US Customary units, and in mm for SI units.

- **Impervious Retention** : Refers to depression storage of the impervious portion of the subwatershed. The value should be given in feet for US Customary units, and in mm for SI units.
- **Initial Infiltration** : maximum or initial infiltration rate (at t = 0). The parameter is accepted in inches/hour for US Customary, and in mm/hour for SI units. The default value is value is 3 inches/hour (76.2 mm/hour).
- **Final Infiltration** : final infiltration rate. The value should be provided in inches/hour for US Customary, and in mm/hour for SI units. The default value is 0.5 inches/hour.
- **Decay Constant** : decay coefficient for Horton's infiltration equation, in per

second. The default value is 0.001/second.

- **Regeneration Constant** : Decay coefficient for the exponential recovery curve, in per second. It is generally considered to be less than decay constant, implying a longer drying curve than wetting curve.

- **Time of Concentration:** Time of concentration for the subcatchment that is just starting to contribute to the node. Minutes.

Information

- **Type** - The type of manhole. Type 0 is a normal manhole, type 1 is a chamber manhole while type 2 is an outlet. Click here to learn more.

- **Elevation** - The ground elevation of the manhole. This value may differ than the rim elevation.

Units are feet or meters.

- **Installation Year** - The year the manhole was installed.
- **Retirement Year** - The year the manhole is expected to be retired from service.
- **Zone** - The service are of the selected manhole.
- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to modeling a simulation.
- **(Other User Created Fields)** - Any fields added by the user will appear at the end of the Information tab. Click here to learn how to add a field.

Output

Output - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a manhole node output report.

Wet Well Data

Wet Well ID

- **ID** - The unique InfoSewer identification of the wet well.

- **Description** - A user defined description of the wet well.

Geometry

- **X** - The x-coordinate of the wet well.

- **Y** - The y-coordinate of the wet well.

Modeling

Type - The type of wet well to be used by InfoSewer. To change the type, merely click in the drop down box and select from the list. Click here to learn more about each type and the fields required.

Information

- **Installation Year** - The year the wet well was installed.

- **Retirement Year** - The year the wet well is expected to be retired from service.

- **Zone** - The service area zone of the selected wet well.

- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to modeling a simulation.

- **Cost ID** - A database field used to assign differing cost identifiers to differing facilities.

- **(Other User Created Fields)** - Any fields added by the user will appear at the end of the Information tab. Click here to learn how to add a field.

Output

Output - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a wet well output report.

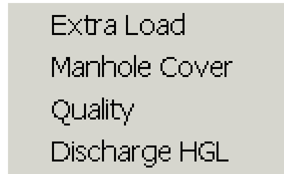
Other Related Topics - Tools menu

Tools Icon

Use this command to access certain advanced tools provided by InfoSewer to manipulate data related to the currently active element. Click on the Down Arrow icon  to launch the **Tools** menu. The tools menu changes depending on the type of element presently selected as shown below.

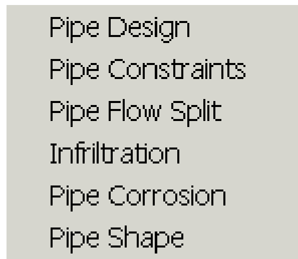
Click on any portion below to learn more:

Manhole

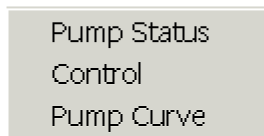


Wetwell

Pipe



Pump

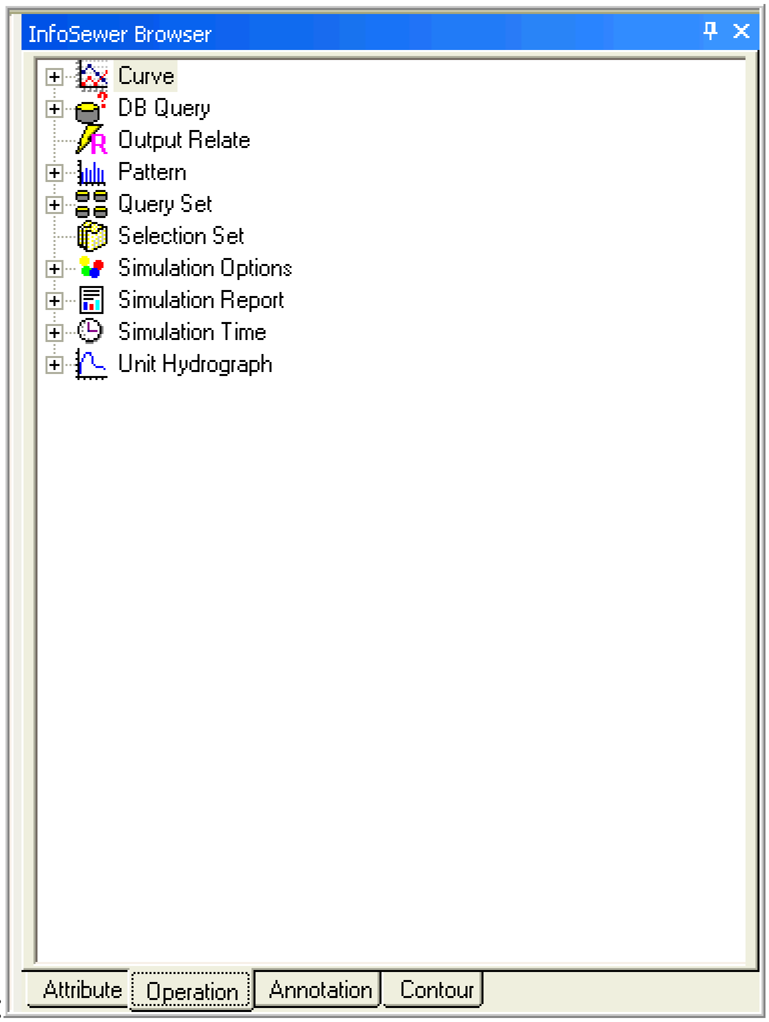


Other Related Topics - Attribute Browser

Operation Data

The Operation Tab section of the InfoSewer **Browser** provides you with tools to create, edit and operate on the different operational features of InfoSewer.

Click on any section below to learn more



Other Related Topics - Annotation Dialog Box, Annotation Methodology, Contour-Labelling, Contour-Level, Contour Dialog Box, Contour Methodology, Contour Options, InfoSewer Browser

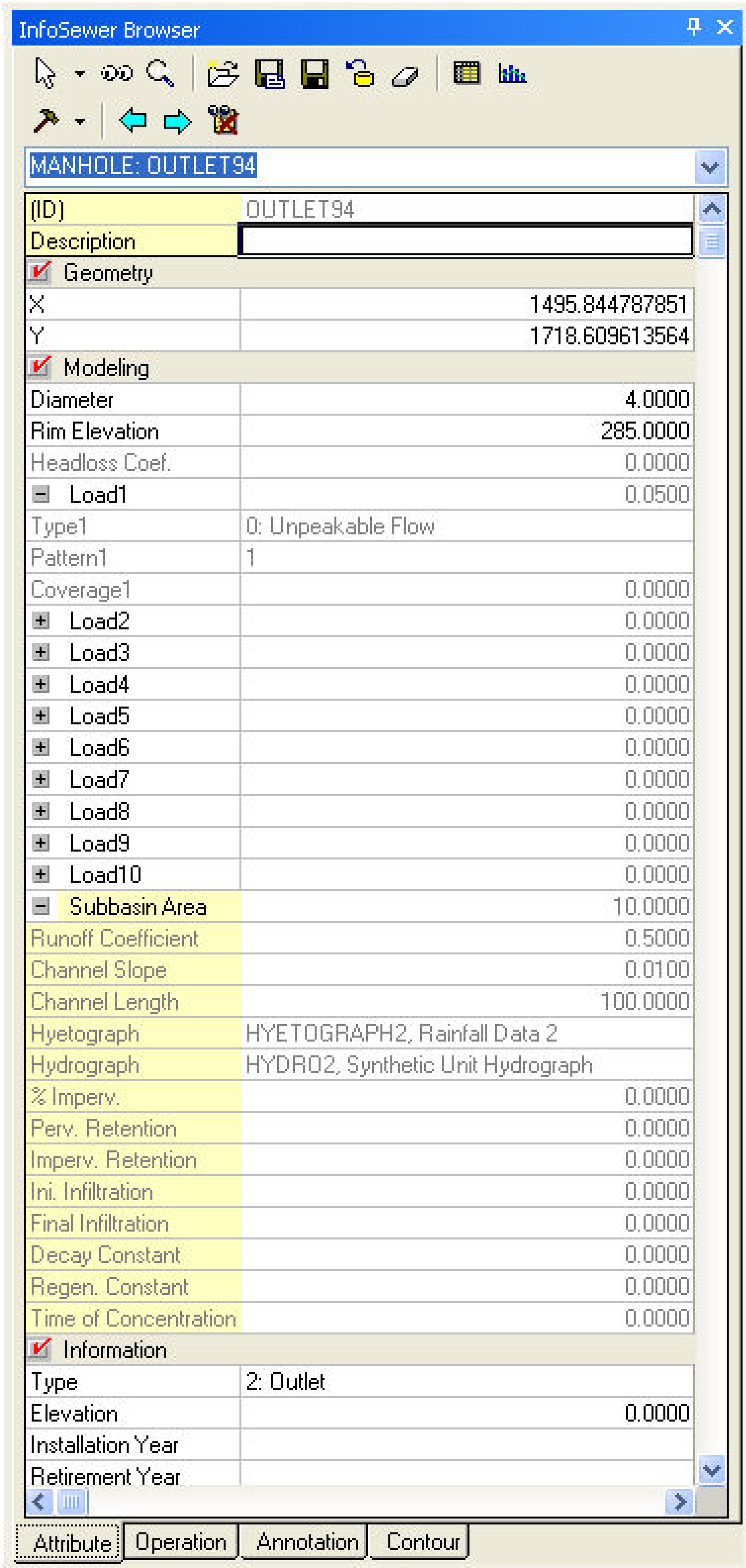
Browser Overview

The InfoSewer Browser provides a centralized location to access the various tools to create, edit, operate and analyze InfoSewer models. The Browser dialog box can be invoked by using the **Attribute Browser** command from the **InfoSewer> View** menu.

The InfoSewer Browser provides four major capabilities for the modeling and analysis of water distribution systems:

- **Attribute Browser:** The Attribute tab displays all of the element attribute tables for the selected data element. See Attribute Browser
- **Contour:** Use the contour tab to plot nodal input and output contours. To learn more about the different contouring options provided by InfoSewer click here.
- **Operation Data:** The Operation Data tab contains all of the operational features of InfoSewer. Click here to learn more about the Operation Data feature of InfoSewer.
- **Annotation:** The Annotation tab is used to color code pipes or nodes based on input and/or output results. Click here to learn more about the InfoSewer Annotation Manager.

Click on any section below to learn more:


InfoSewer Browser
MANHOLE: OUTLET94
(ID) OUTLET94
Description
Geometry
X 1495.844787851
Y 1718.609613564
Modeling
Diameter 4.0000
Rim Elevation 285.0000
Headloss Coef. 0.0000
Load1 0.0500
Type1 0: Unpeakable Flow
Pattern1 1
Coverage1 0.0000
Load2 0.0000
Load3 0.0000
Load4 0.0000
Load5 0.0000
Load6 0.0000
Load7 0.0000
Load8 0.0000
Load9 0.0000
Load10 0.0000
Subbasin Area 10.0000
Runoff Coefficient 0.5000
Channel Slope 0.0100
Channel Length 100.0000
Hyetograph HYETOGRAPH2, Rainfall Data 2
Hydrograph HYDRO2, Synthetic Unit Hydrograph
% Imperv. 0.0000
Perv. Retention 0.0000
Imperv. Retention 0.0000
Ini. Infiltration 0.0000
Final Infiltration 0.0000
Decay Constant 0.0000
Regen. Constant 0.0000
Time of Concentration 0.0000
Information
Type 2: Outlet
Elevation 0.0000
Installation Year
Retirement Year
Attribute
Operation
Annotation
Contour

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

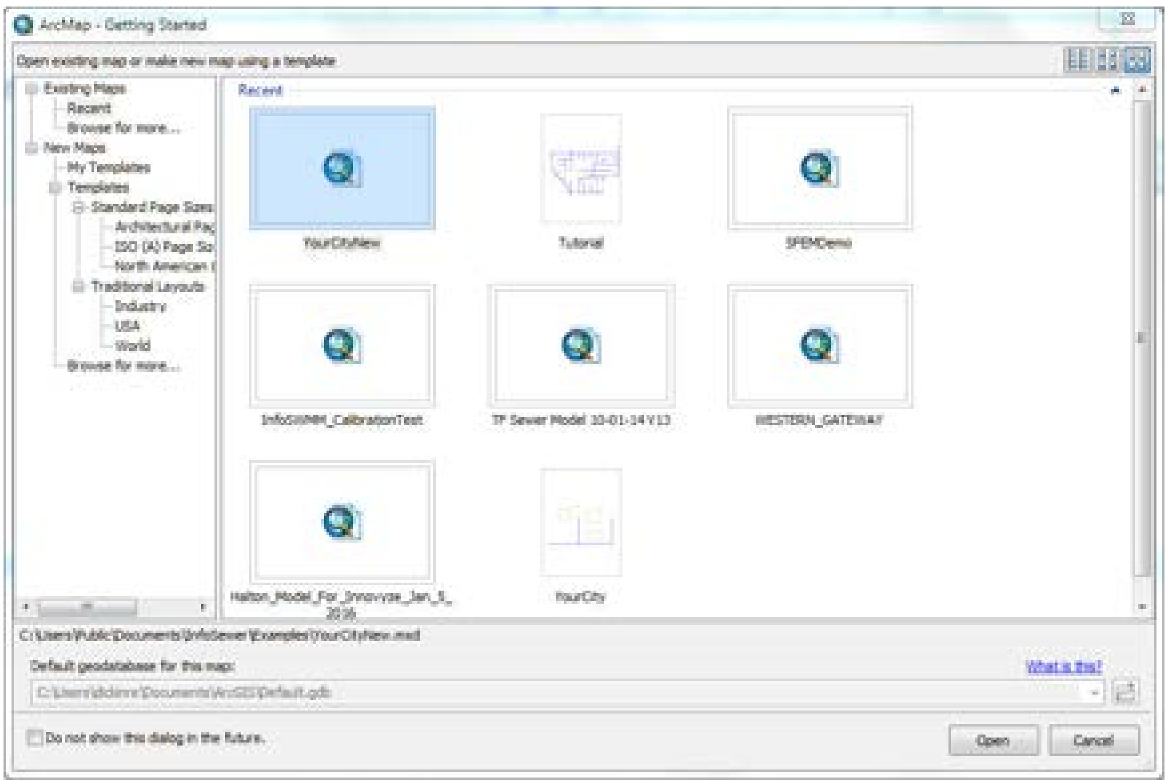
Innovyze Help File Updated March 31, 2021

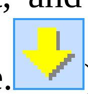
More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

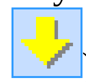
Opening an Existing Project in Arc Map 10.8, 10.7, 10.6, 10.5, 10.4, 10.3

Existing InfoSewer projects could be opened using several ways:

- From Windows Start Menu:
- From the Start Menu, point to Programs and then to InfoSewer, and double click on the InfoSewer logo. You could also double click on InfoSewer short cut you might have. This will open an ArcMap, and the following dialog box will pop up.



- From the list of InfoSewer projects, double click on the project you want to open.
- This will open the project, and you will need to initialize the project by clicking on the arrow (i.e. ) from the  button from the InfoSewer Control Center Toolbar. This will open the InfoSewer project.
- From Window's Explorer

- Point to the directory where the project is located (e.g. C:\Program Files\InfoSewer\Examples).
- Double click on the .mxd file. This will open the project and you will need to initialize the project by clicking on the arrow (i.e. ) from the 

button from the InfoSewer Control Center Toolbar. This will open the InfoSewer project.

Working With Projects

Essentially, InfoSewer Project refers to the entirety of the contents (i.e., the data, the graphical representation, and the output) of the modeling exercise conducted using InfoSewer. Every InfoSewer project can have up to three major files/file folders. As an example, an InfoSewer project named "Sample" will have a file folder named **Sample.ISDB** that contains databases of the project; a file named **Sample.mxd** that contains the graphical representation of the drainage network; and an output file folder named **Sample.OUT**. The output file folder is created only after successful InfoSewer simulation is done.

This section of the help file describes the following features and functionality of InfoSewer that the modeler may has to perform or may has to utilize while working with an InfoSewer project.

- Opening InfoSewer Project
- Setting Preferences
- The Scenario Manager
- The Domain Manager
- The Facility Manager
- The Elevation Extractor
- Running a Model
- Output Data Analysis

InfoSewer™

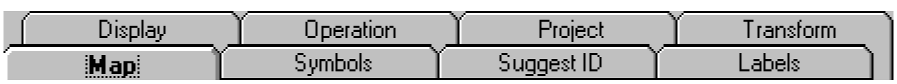
Modeling Made Easy

Preferences

The Preferences command is used to control the InfoSewer system settings. All changes made on the Preferences dialog box must be applied prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

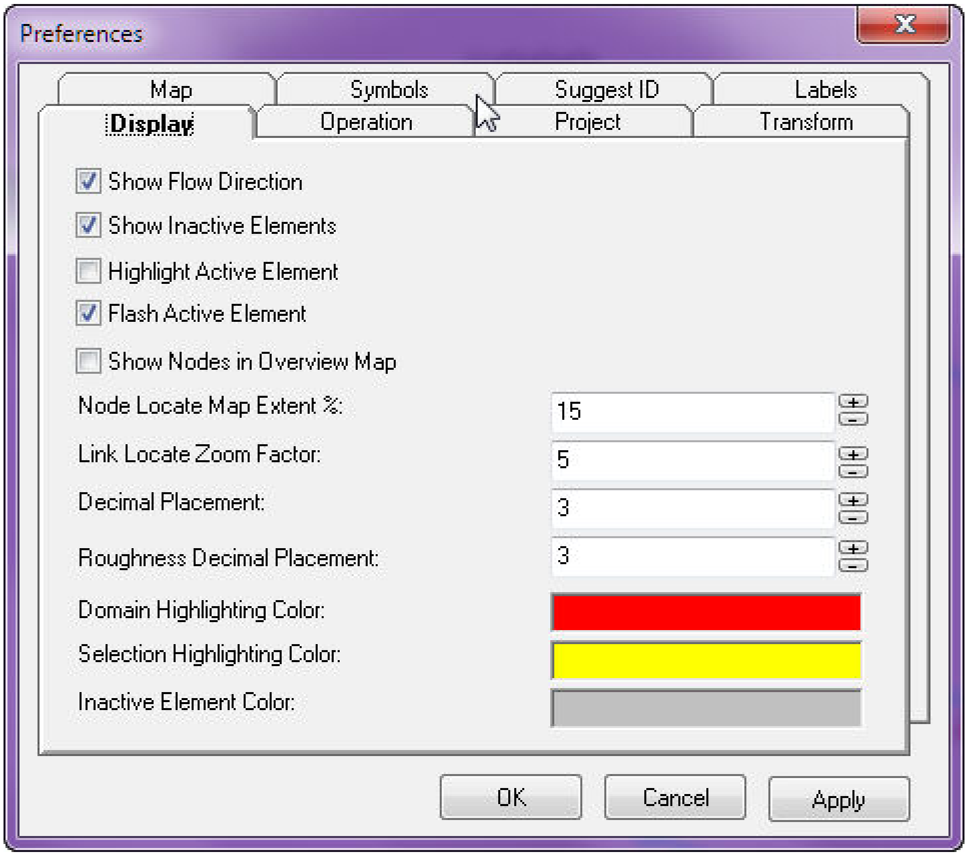
- Operation Settings - Specify the different operation settings here.
- Display Settings - Use this to specify the display properties.
- ID and Description - ID and Descriptions may be set here.
- Labeling - Specify your Map label preferences here.
- Selection Settings - Specify the project selection settings.
- Default Symbol- Specify the default symbol sizes for all the InfoSewer data elements here.

To display project preferences, from the **Tools** menu, select **Preferences**. Click on any tab to learn more about the user options related to the subject. A description of all options is shown below each dialog box.



Display

The Display tab controls visual effects and features within InfoSewer.

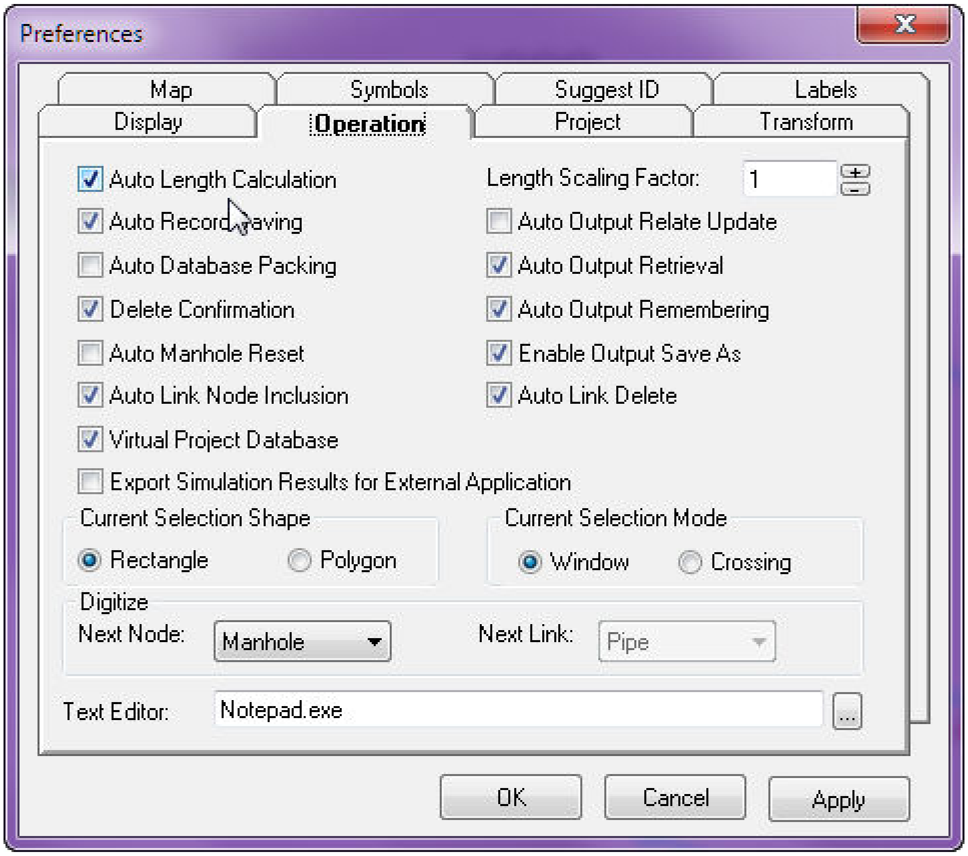


- **Show Flow Direction** - Enables whether or not flow arrows will be shown in the InfoSewerproject. When OFF, the user will not be able to place flow arrows in the project, regardless if the "Show Flow Direction" check box under the Control Center -> Map Display -> Display Settings box is checked.
- **Show Inactive Elements** - Controls whether or not inactive elements in the project will be displayed or not. When ON (checked), elements not in the current *active* facility set will be displayed with the color as shown in the Inactive Element Color box.
- **Highlight Active Element** - When ON (checked), any element selected in InfoSewer with the Select icon will be highlighted with a selection box. When OFF, data elements that are selected in InfoSewer will not be highlighted.
- **Show Nodes in Overview Map** - When ON (checked), all nodes will be displayed in the Map Overview box.

- **Locate Node Map Extent %** - The value must be greater than zero and is used with the current map (full) extent to calculate the size of the graphical extents of a zoom. For example, the current map full extent: 10.0 X 16.0; node locate map extent %: 50.0 -> zoom in box size: 5.0 X 8.0.
- **Link Locate Zoom Factor** - The value must be greater than zero and is used with the extent box of the link to be located to calculate the size of the zoom in box. For example, link locate zoom factor: 2.0; extent size of the link to be located: 4.0 X 2.0 -> zoom in box size: 8.0 X 4.0.
- **Decimal Placement** - Controls the level of decimal accuracy when model results are placed as annotation (text) in the project.
- **Domain Highlighting Color** - Sets the color for elements as they are added to a domain. Red is the default.
- **Selection Highlighting Color** - Sets the color for elements as they are added to a selection set. Yellow is the default.
- **Inactive Element Color** - Sets the color for elements when they are not part of the *active* facility set. When the Show Inactive Elements box is checked, inactive elements are shown in the InfoSewer display as this color. Grey is the default.

Operation

The Operation tab allows the user to control project settings options relating to the way data is controlled in InfoSewer.



- **Auto Length Calculation** - When ON (checked), the length of digitized pipe segments will be automatically calculated and input into the InfoSewer database. When (OFF) unchecked, pipe lengths will not be automatically created from the graphical distances.
- **Auto Record Saving** - Controls if modifications to records on various dialog boxes (such as the Edit Patterns, Edit Curves, Edit Data Set, etc.) are immediately saved when you pick another record for editing on the same dialog box. When ON (checked), InfoSewer automatically saves edits to one database record when you change to another record on the same dialog box. When OFF, you must choose the Save button before moving to another record.
- **Auto Database Packing** - When ON (checked), the Auto Database Packing preference automatically deletes database records related to a deleted element. When OFF (unchecked), records are only marked for deletion, allowing records to be recalled. Use the Recall command to restore data elements deleted from the InfoSewer project. When a database is "packed", this means that all records marked for deletion

are permanently deleted from the database. Do not pack a database if you plan to recall data elements.

- **Delete Confirmation** - Controls whether or not you are prompted to verify delete operations. When this option is not checked, components will be immediately deleted without user verification.
- **Auto Manhole Set** - Controls how manhole sets will be loaded (activated) in the current project. When ON (checked), sewer loadings for all active manholes are set to null (zero) and then, for those manholes with a matching record in the current loading set, loads will over-write the current loading set. When OFF, loadings will be re-loaded from the loading set for those manholes with matching records in the loading set. For those manholes with no matching records in the newly loaded set, the value of the current loadings are preserved.
- **Auto Link Node Inclusion** - Controls whether or not nodes directly connected to activated links will also be activated for a hydraulic simulation. When ON (checked), all nodes directly connected to activated links will also be activated. When OFF, nodes directly connected to activated links are not automatically activated. Using the "Add Link Nodes" button from the Facility Manager will ensure that end nodes will be included in the activated set. A hydraulic simulation will fail if end nodes are not activated.
- **Length Scaling Factor** - Applies a global scaling factor to be applied to all pipes added after setting this preference. The pipe length (imported or calculated) is multiplied by the entered scaling factor. Do not use if your GIS base map is to actual scale units.
- **Auto Output Relate Update** - Controls whether or not output relates are automatically updated each time you run a simulation. When OFF (unchecked), results from the most recent simulation run are *not* written to the relate table.
- **Auto Output Retrieval** - Controls whether or notInfoSewer makes simulation results from a custom scenario available when activating that scenario. When ON (checked), InfoSewer determines if the scenario being loaded has a valid output source containing results from a previous simulation run and then loads that output source into

InfoSewer as the *active* output. When OFF, InfoSewer disregards any previously generated simulation results when activating a custom scenario. If OFF, the user must re-run the desired scenario before reviewing simulation results for the loaded scenario.

- **Auto Output Remembering** - Controls whether or not H2OMAP Sewer preserves simulation results for the current scenario when de-activating that scenario and switching to another custom scenario. When ON (checked), simulation results associated with the scenario being de-activated remain available in the current project as an output source. When OFF, only the *active* output source is available for reports, graphing, etc.

- **Enable Output Save As** - Controls whether or not simulation results are saved with an InfoSewer project when that project is saved as a new file name. When ON (checked), H2OMAP Sewer copies simulation results for the current project to the new project. When OFF (unchecked), only model inputs are saved to the new project. If you choose to save simulation results to the new project, the [project_name].OUT directory will be copied in addition to the [project_name].DB folder containing model input data.

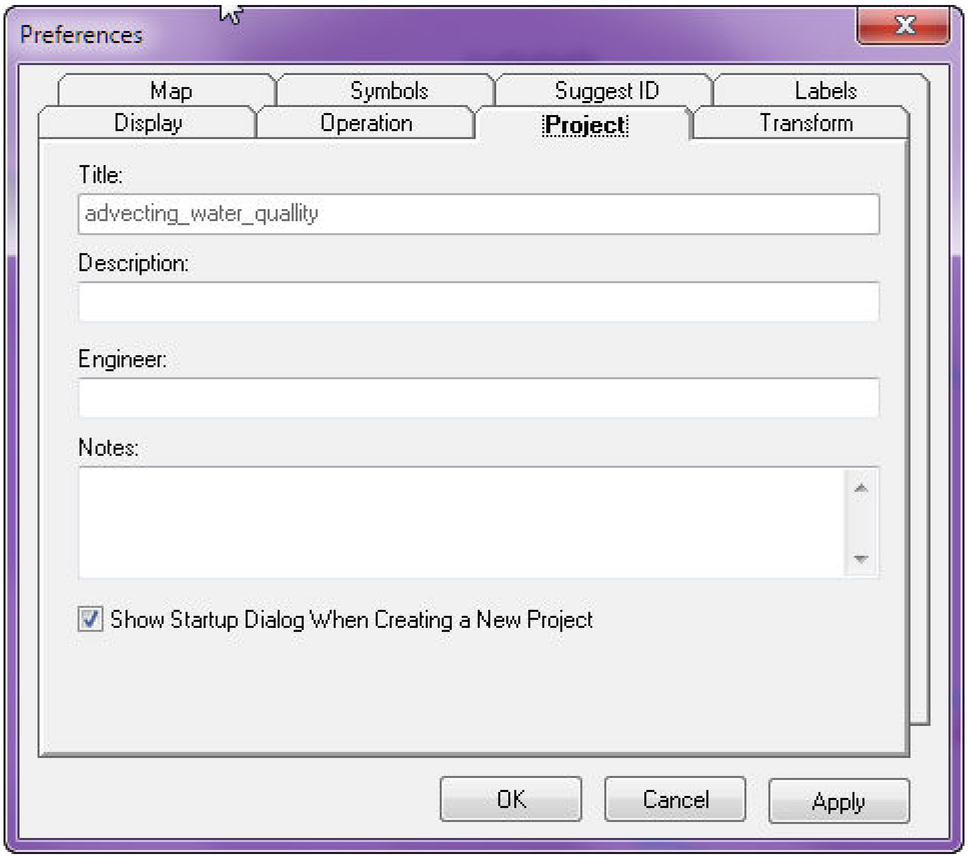
- **Export Simulation Result for External Application** - If turned on this feature generates modal outputs in dbf format, which could be opened from spreadsheets and flexibly manipulated by the users. The dbf files containing model results will be saved to the "*.OUT" folder of the InfoSewer Project, where * is the file name of the project. The exact location of the dbf file will be …*.OUT/Scenario/scenario name/RunType/. Where scenario name indicates name of the modeling scenario you might have defined and run. If no custom scenario is created and/or if you are interested in the base scenario results, the name will be "Base". Run Type could be "Dynamic", "Static", or "Design" depending on the type of simulation you executed (i.e., dynamic simulation, steady state analysis, or design simulation), respectively.

- **Auto Link Delete** - Controls whether or not the link(s) connected to a node selected for deletion will remain or be deleted. When ON, the link connected to a node selected for deletion will be deleted as well. Use the Recall command to recall a deleted node or link.

- **Current Selection Shapes** - Controls whether a graphical selection in the InfoSewer project is either by a window (rectangle) or a polygon selection. Choose the option that works best for you.
- **Current Selection Mode** - Controls whether a graphical selection in the InfoSewer project selects objects contained only within a "window" selection or to select those that "cross" the selection box or polygon.
- **Digitize (Next Node/Next Link)** - Select a different data element from the dropdown box to have InfoSewer create a different element type when the digitize network icon is initialized. For example, if the user is digitizing a pipe network, use "pipe" as the next link option. When the digitize network command is initialized, the network will create pipes as the link entities.
- **Text Editor** - Allows the user to specify an ASCII text editor for editing output reports generated via the Run Manager.

Project

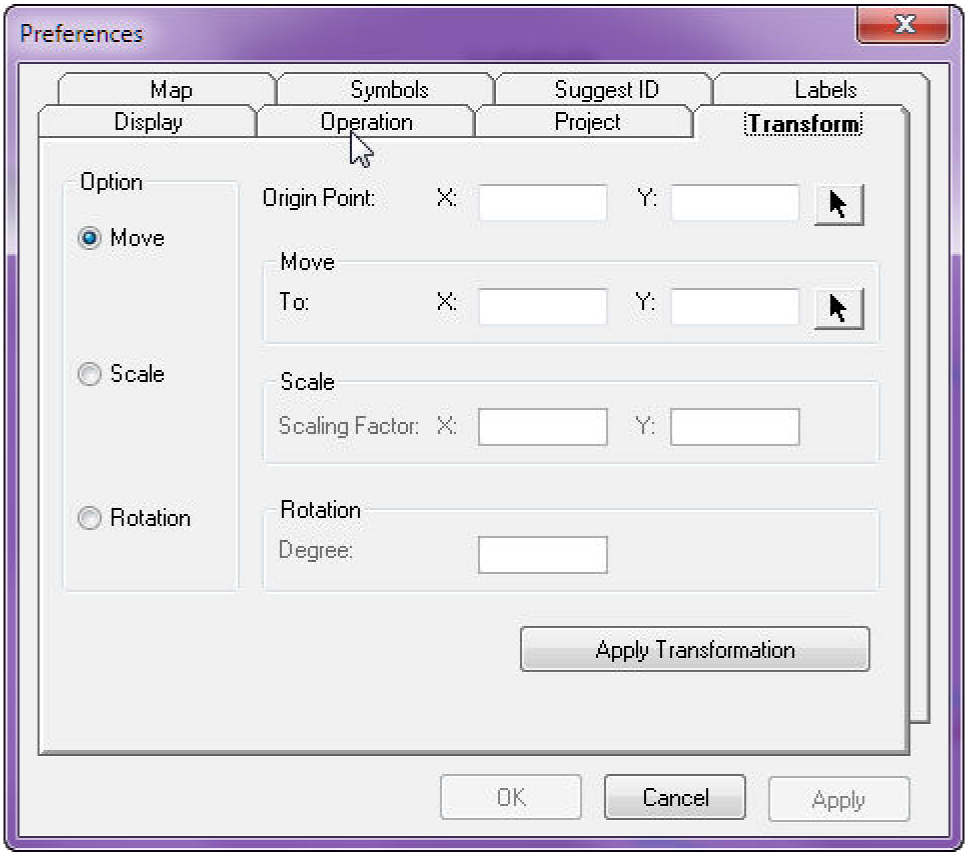
The Project tab allows the user to place information in InfoSewer that is project specific.



- **Title** - The file name of the InfoSewer project.
- **Description** - A description for the InfoSewer project.
- **Engineer** - The engineer responsible for the hydraulic model.
- **Notes** - Notes as desired.

Transform

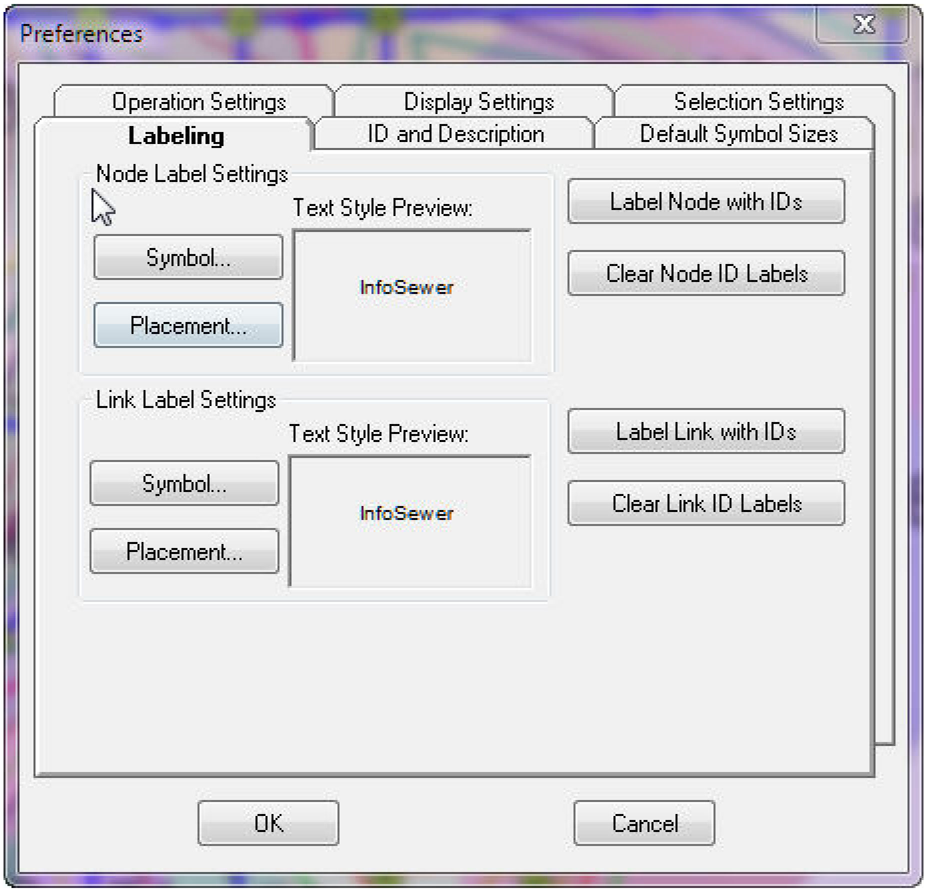
The Transform tab allows the user to move, scale or rotate the InfoSewer project from one coordinate to another.



- **Option** - Select the radial button for the type of translation to be performed. Move the InfoSewer project from one coordinate to another. Scale the project according to an X and Y scaling factor. Rotate the project around an origin point according to a decimal degree angle.
- **Origin** - The origin point for which the move, scale or rotate option is to occur.
- **Move** - Specify the To X,Y coordinate to move the InfoSewer project.
- **Scale** - Specify an X and/or Y scaling factor to scale the InfoSewer project. Specify the same scale factor in the X and Y box to ensure a uniform scale. If you wish to distort the scaling procedure (where one direction is larger than the other), specify differing scale factors for X and Y scales.
- **Rotate** - Specify an decimal degree angle (0-360) to rotate the InfoSewer project around the origin point.

Labels

The Labels tab allows the user to control the appearance of ID labels during an InfoSewer project.

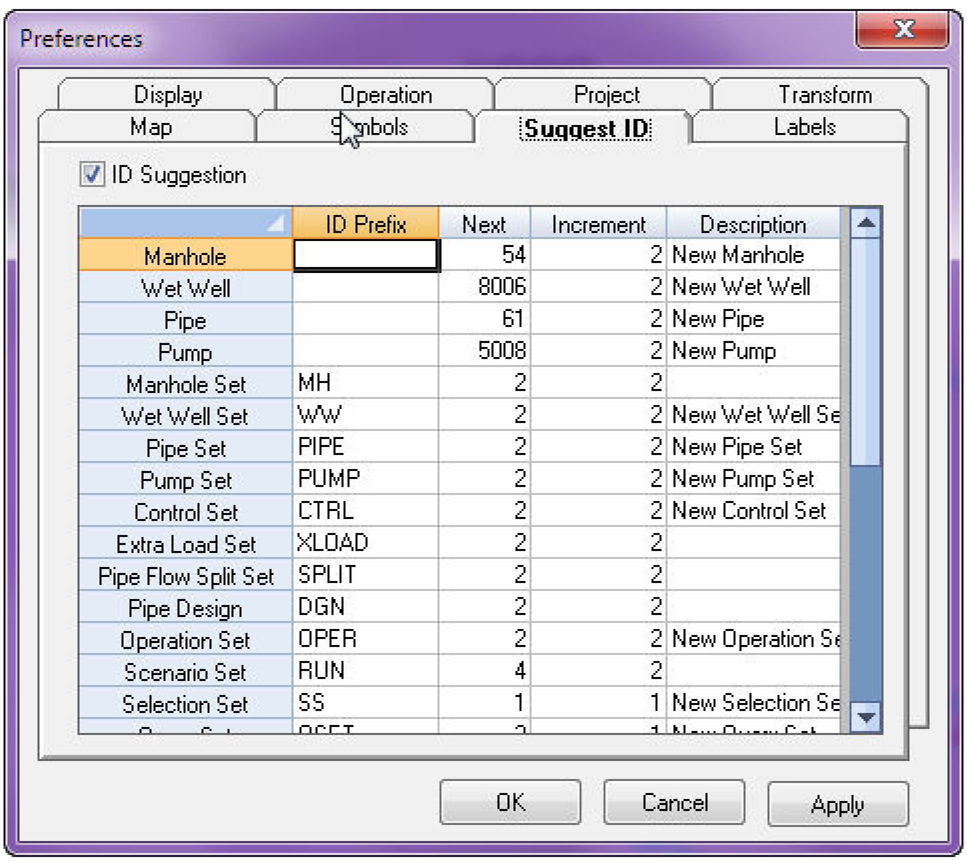


- **Scale Text** - When ON (checked), InfoSewer will keep node and pipe labels at a constant map scale as specified with the text size box. When OFF (unchecked), the size of the text font will maintain a size in relative proportion to the zoom scale.
- **Text Size** - The size of label text in map units. For example, for a 100 scale plot with a 1/8 inch text height for labels, set the text size to 12.5 (0.125 x 100).
- **Font & Color** - Click on the Font and Color buttons to change the appearance of the ID labels.

- **Show Node ID Label** - When ON (checked), all nodes in the InfoSewer project will be labeled with the ID field.
- **Node ID Label Horizontal Offset** - Number of X units away from the center of the node the insertion point of the ID label will be placed.
- **Node ID Label Vertical Offset** - Number of Y units away from the center of the node the insertion point of the ID label will be placed.
- **Show Link ID Label** - When this box is checked, all links in the InfoSewer project will be labeled with the ID field text.
- **Link ID Label Offset** - Distance in map units from which to place ID label text for links. A positive value equates to placing the text above the line, a negative value below the line.

Suggest ID

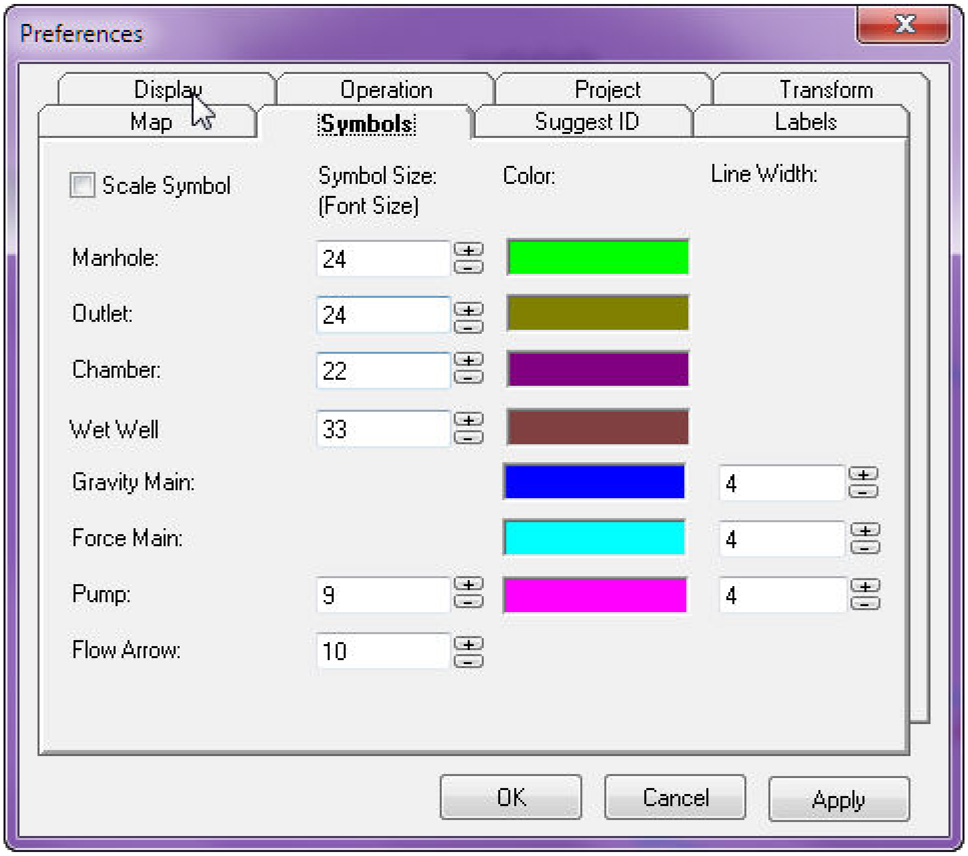
The Suggest ID tab allows the user to change InfoSewerdefaults for data element labeling, data sets, curves, patterns, etc.



- **ID Suggestion** - When checked on, InfoSewer uses the information as shown in this dialog box when creating new data elements, data sets, etc. When OFF (unchecked), the user will have to supply the ID and description for each new element or set, etc.
- **ID Prefix** - For data elements, the prefix represents the alpha-numeric value (input by the user) that will be inserted ahead of the unique ID in the InfoSewer element database. For example, if the ID prefix for a Wet Well is changed to "SA12-" then the ID in the database for the next tank created in InfoSewer will be "SA12-[ID value]". The ID prefix tab also allows how the next data set, curve, pattern, etc. will be named.
- **Next** - The Next column represents what the "Next" data element, curve, pattern, etc. created will be labeled. Of course, the prefix comes first, then this next alpha-numeric value comes behind the prefix as a unique ID in the InfoSewerdatabase.
- **Increment** - The labeling interval to be used in the InfoSewer project.
- **Description** - The default description that will be applied when an element or set is created. For example, if the user is digitizing a series of nodes that are to be placed on Elm Street, then changing the description to Elm Street will ensure that every node digitized will have that as its description.

Symbols

The Symbols tab is used to adjust the size (graphical appearance) of the InfoSewer data elements.



- **Scale Symbol** - When this box is checked, InfoSewer will fix the size of the data elements as described in the symbol size box. This means that when the user zooms in and out of the InfoSewer project, the size of the data elements will remain constant.
- **Symbol Size** - Used to adjust the size of the date element in the InfoSewer display. The numbers represent scale ratios in relation to the extents of the InfoSewer display.
- **Color** - Click on the color box to change the default colors for the InfoSewer data elements.
- **Line Width** - In map units, specify the width of the line to be used for InfoSewer data elements.

Map for InfoSewer

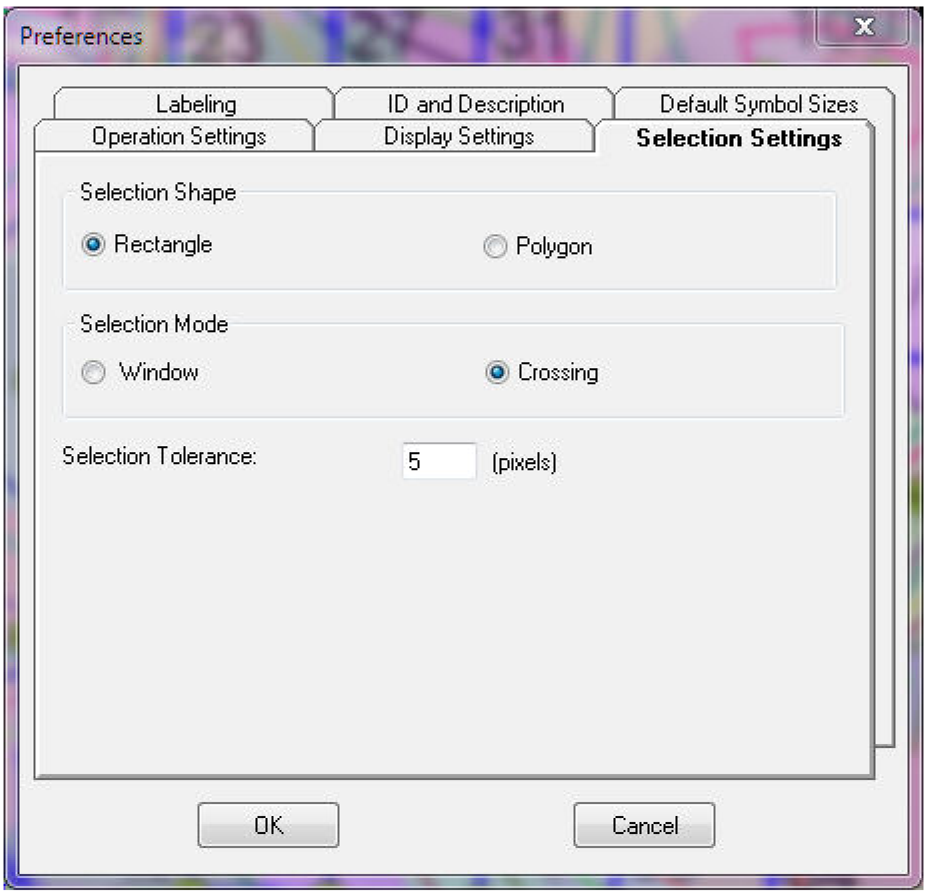
The Map Properties tab is used to control the appearance of the project.

- **Scrollbars** - When OFF (unchecked), the horizontal and vertical scrollbars along the map display will be disabled.
- **3D Appearance** - When ON (checked), the border for the display will become beveled, giving the border a 3D appearance.
- **Border Style** - Select from having the border for the display shown or not shown.
- **Background** - Clicking in the color box allows the user to change the color of the background in the display. When a color is selected, click OK and then Apply.
- **Snap to Grid Point** - When checked, entities created or moved in will be snapped to the nearest grid point.
- **Show Grid Point** - When ON (checked), will display a grid at a spacing as provided in the Grid Spacing box. If the grid spacing is too small, will not display the grid, but the grid will still be present.
- **Grid Spacing** - The mapping units of space between each portion of the grid.
- **Origin** - The point at which the grid begins. For example, if an origin of 1.8,1.8 is specified with a grid spacing of 0.5, then the grid ticks will fall on every .8 and .3 map units.
- **Pick Point (Pick Node)** -To specify a node (or a point in space) as an origin instead of entering a coordinate.

Preferences - Selection Settings for InfoSewer

The Preferences - Selection Settings command is used to control the InfoSewer selection settings. All changes made on the Preferences dialog box must be saved (by clicking the OK button) prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

To display Selection Settings, from the InfoSewer Control Center -> InfoSewer button -> Tools sub-menu, select Project Preferences -> Selection Settings.



- Selection Shape - This option specifies the default selection type as rectangle or a polygon.

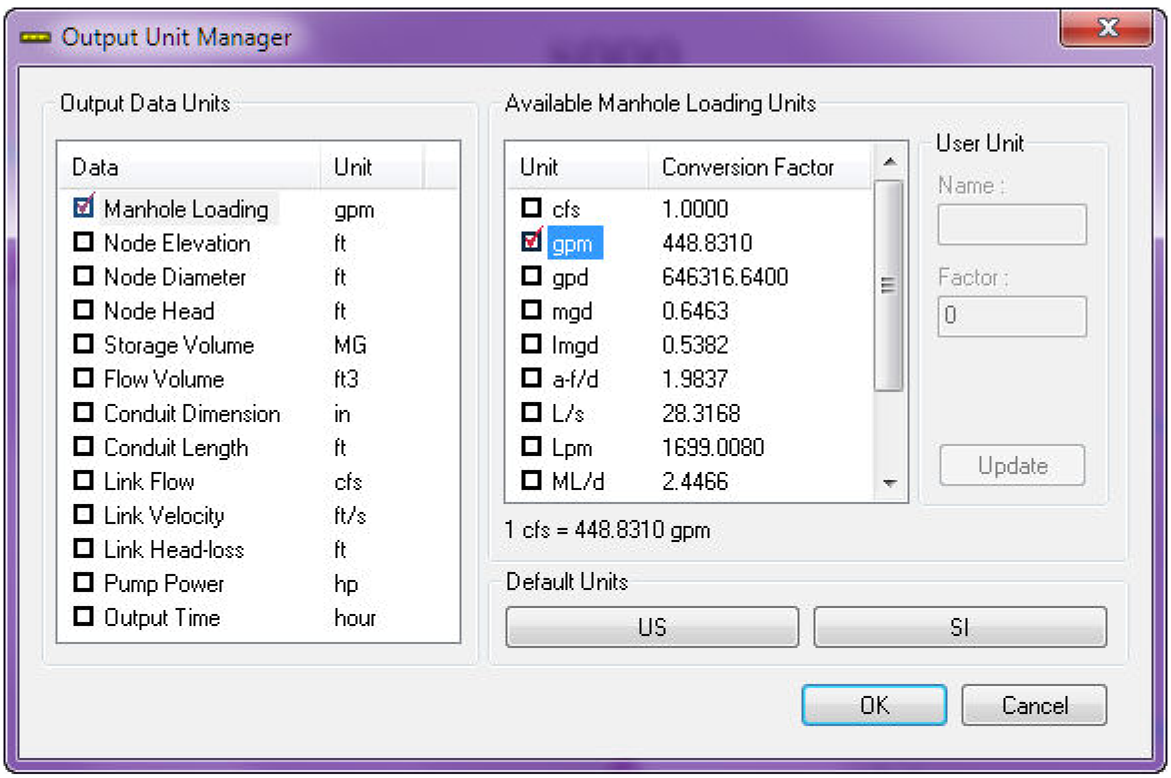
- Selection Mode - This option specifies the default selection type as a window selection (completely surround the elements) or a crossing (merely touching the desired elements).
- Selection Tolerance - In this box specify the range of tolerance for selection settings.
- OK - Click on OK to save and exit from your InfoSewer Preferences dialog box.
- Cancel - Use this to exit without saving from your InfoSewer Preferences dialog box.

Output Unit Manager

The Output Unit Manager is used to specify the units of measurement for simulation result variables to units other than the defaults. You may choose from any pre-defined units or may define user-specified units by applying the appropriate conversion factor.

In changing units from the Output Unit Manager, the user is able to create and compare both reports and graphs from the same model run without having to re-run the model.

To view the Output Unit Manager, from the **Tools** menu, select **Output Unit Manager**.



- **Output Data Units** - All **Sewer** units dependent values that can be modeled.
- **Available Units** - All units of measurement available from **Sewer** that output data can be changed to (red check box indicates currently selected model unit).

Units selected with the Output Unit Manager affect all **Sewer** commands and functions that reference simulation results including, but not limited to, the Animation Viewer, Annotation, Map Display, Map Legend, Custom Report Manager, Query Reports, and the Output Report Manager (graphs and text reports).

(Note: Be sure to refresh graphs and reports on the Output Report Manager to reflect units chosen on the Output Unit Manager dialog box. Select each graph and report in turn and then use the Refresh button to update graphs and reports.)

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer Project Menu

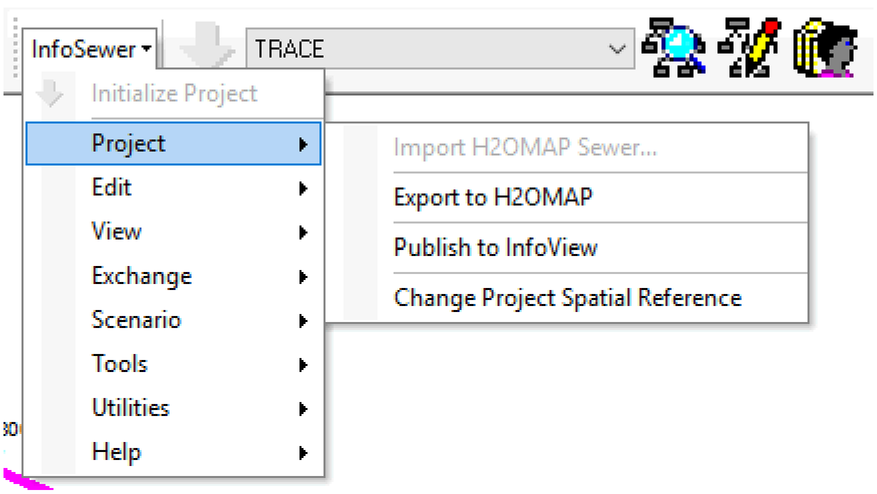
Descriptions of All Project Menu commands in InfoSewer.

Import H2OMap Sewer - H2OMap Sewer has been deprecated by Innovyze as of 2018.

Publish to InfoView - This command helps you to publish an InfoSewer Project to InfoView.

Export to H2OMap - Save to H2OMap Sewer (deprecated)

Change Project Spatial Reference - Change the Spatial Reference of the InfoSewer Map



Create New InfoSewer Project

When creating a new InfoSewer project you can define the project's coordinate system here. InfoSewer provides the following tools to specify a project's coordinate system:

ArcGIS Default - Use the InfoSewer default coordinate system. This default spatial reference system follows ArcGIS's default settings. It's extent (or XY domain) is:

Minimum X: -10000

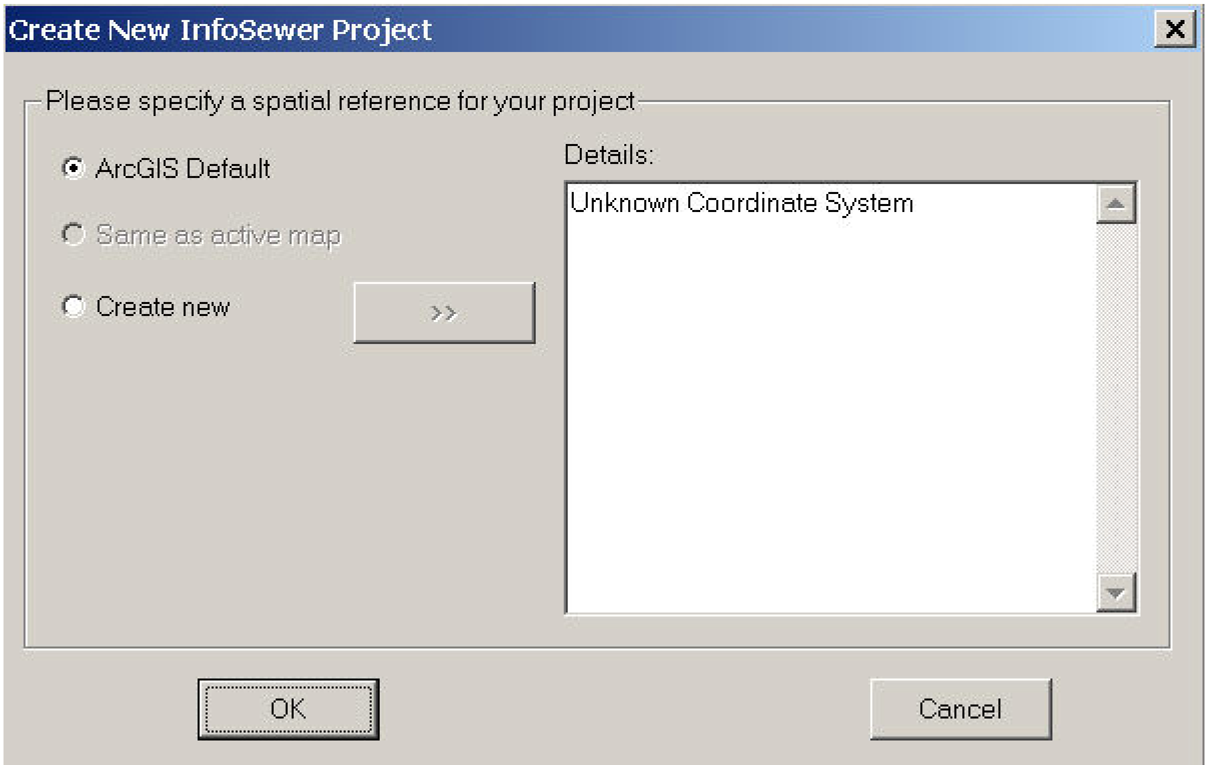
Maximum X: 11474.83645

Minimum Y: -10000

Maximum Y: 11474.83645

The Same as the Active Map - Use this to assign the presently active map's background layer's coordinate system. This option is enabled only if you have a background layer associated with your project.

Create New - Use this to either define a new custom coordinate system or use an existing available coordinate system from the ArcGIS library. Click here to learn more.



Other Related Topics - Change Project Spatial Reference, Degree of Resolution, New Spatial Reference Dialog box, Spatial System Coordinate Range

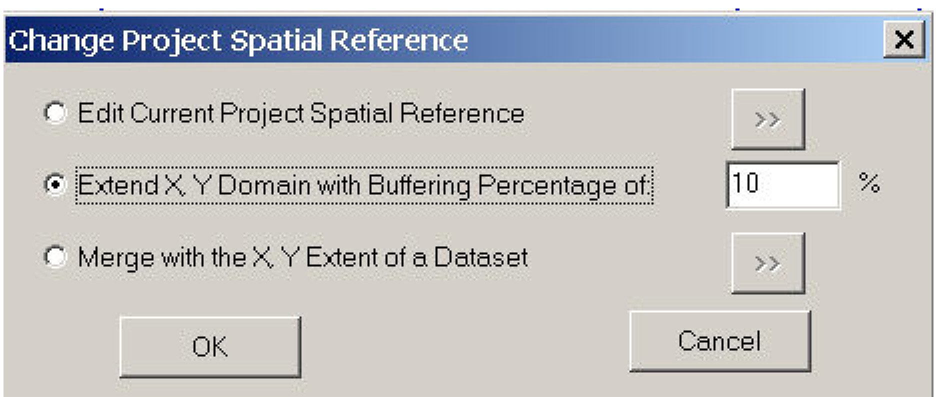
Change Project Spatial Reference

The spatial reference of a dataset in geodatabase can not be further modified once created in ArcGIS and new features can not be inserted if it's coordinates are beyond X, Y domain of the spatial reference. InfoSewer provides a very powerful tool to overcome this limitation of ArcGIS.

In InfoSewer the spatial reference of a project can be modified by either editing current project's spatial reference, extending the X, Y domain with buffering percentage, or merging with the X, Y extent of a dataset.

To launch the Change Project Spatial Reference dialog box, click on the InfoSewer Button 

, browse to the Project menu and select the Change Project Spatial Reference option.



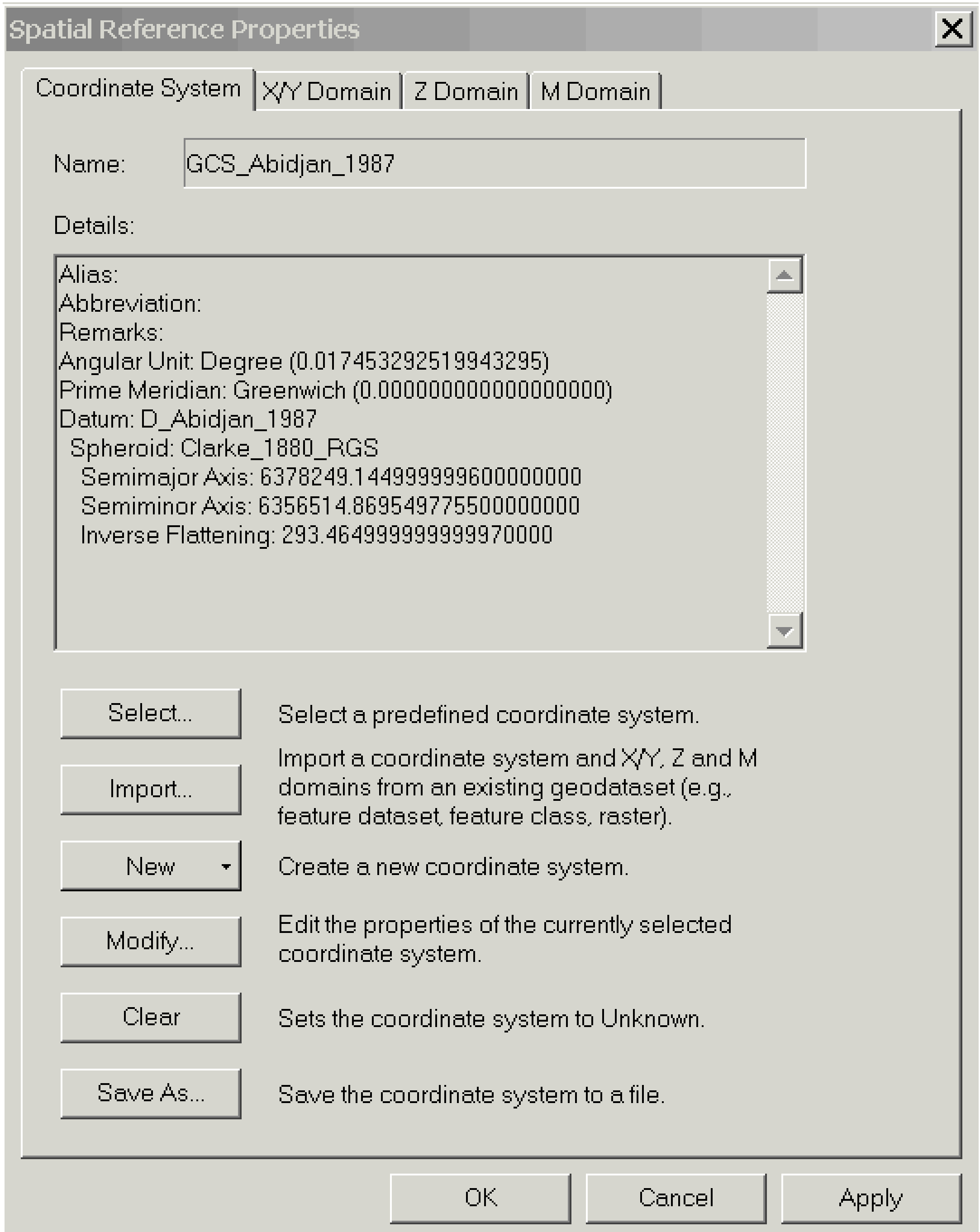
- Edit Current Project Spatial Reference: Use this to modify/edit the spatial reference of a current project.
- Extend X,Y Domain with Buffering Percentage: Use this to modify the spatial reference of a project by extending the X,Y domain with buffering percentage.
- Merge with the X,Y Extent of a Dataset: Use this to modify the spatial reference of a project by merging with the X,Y extent of a dataset.
- OK: Click on OK to accept changes to the spatial reference coordinates and close out of the dialog box.

- Cancel: Use this to cancel the selection and close out of the dialog box. Any changes and/or selections made will not be reflected in the project.

New Spatial Reference Dialog box

Use this to either define a new custom coordinate system or use an existing available coordinate system from the ArcGIS library.

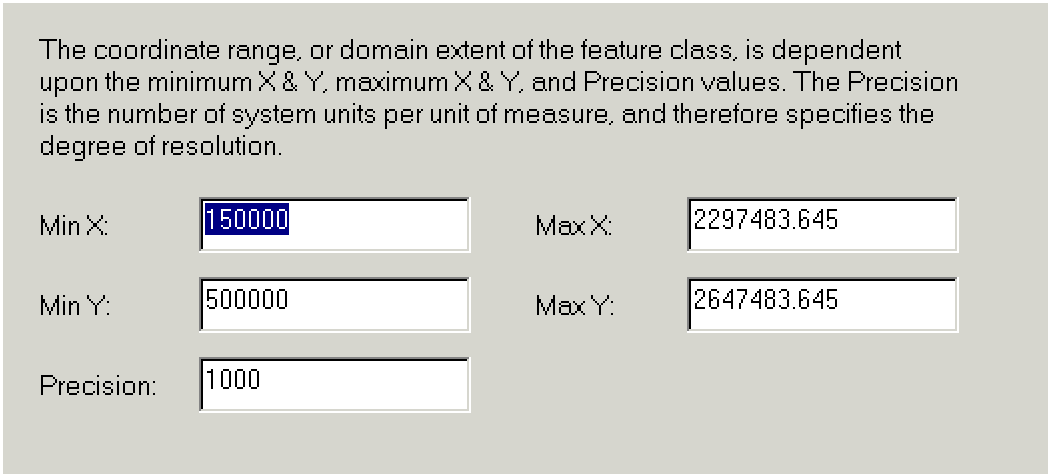
Click on any section below to learn more. Click on Next to launch the Spatial System Coordinate Range dialog box.



Other Related Topics - Change Project Spatial Reference, Create New InfoSewer Project, Degree of Resolution, Spatial System Coordinate Range

Spatial Coordinate Range

Click on **Next** on the New Spatial Reference Dialog box to launch this dialog box. Specify your coordinate range here. Click on Next to launch the Degree of Resolution dialog box.

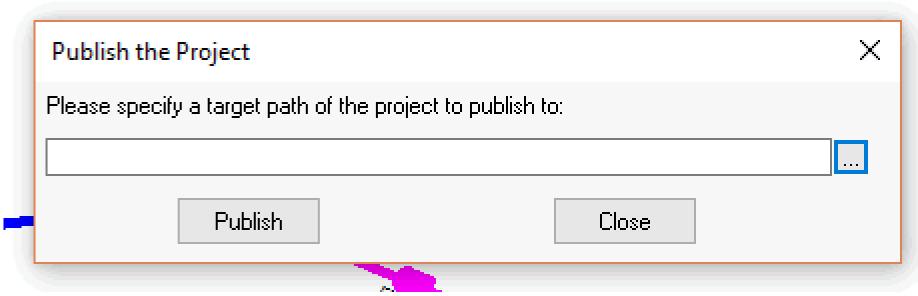


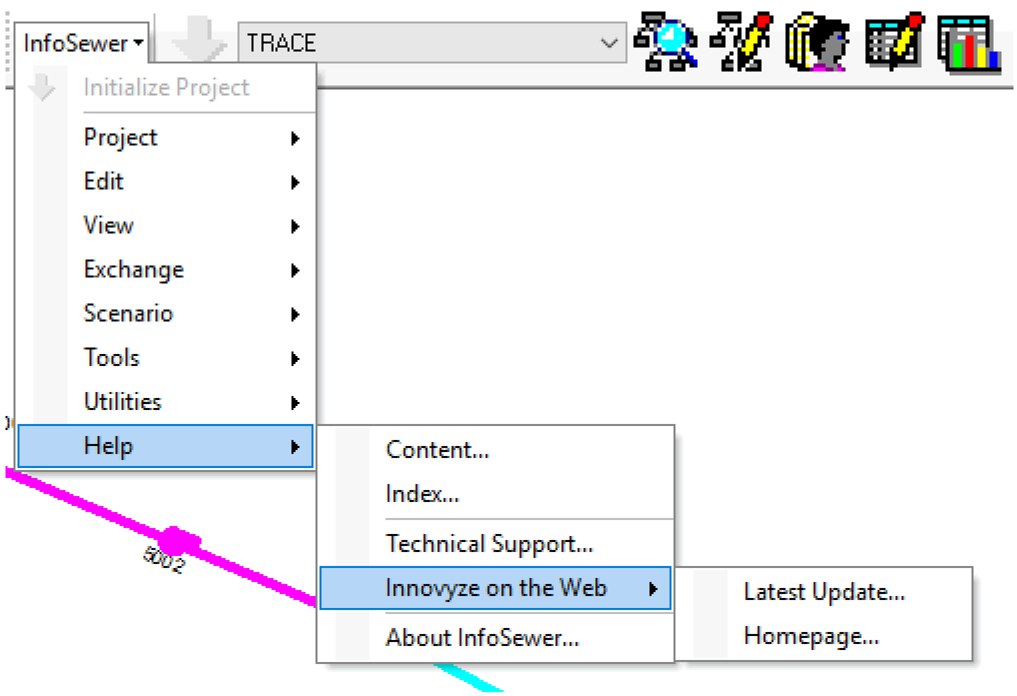
Other Related Topics - Change Project Spatial Reference, Create New InfoSewer Project, Degree of Resolution, New Spatial Reference Dialog box

---

Publish to InfoView

This command helps you to publish an InfoSewer Project to InfoView.





InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Edit Menu

Descriptions of All Edit Menu commands in InfoSewer.

The **Database Editor (DB)** allows the user to open any InfoSewer database and edit user input fields (except ID). To access the DB Editor, from the **Edit** menu, select **DB Tables** to see the Open Table dialog box below.

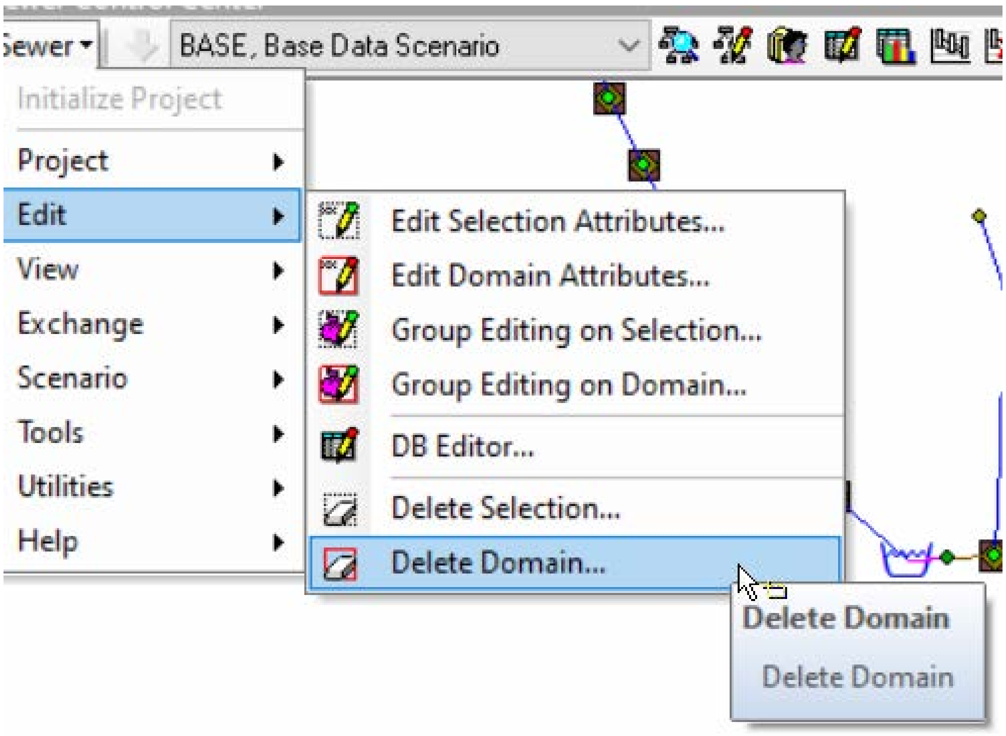
The **Edit Attribute** dialog box allows the user to edit any field from the selected (graphical or Domain) InfoSewer data elements. This feature is helpful by allowing quick edits of database data to occur without having to use to DBEditor command.

The **Group Editing** command allows you to specify modeling properties of multiple network components in a single operation.

> To run the command, from the **Edit** menu, select **Group Editing on Selection** or **Group Editing on Domain**. If you have created a domain, the group editing dialog box will appear once the Group Editing on Domain command is initiated. The group editing on selection option allows for a graphical selection to be made. Once initiated, a "?" will appear next to the mouse cursor. Drag a selection window across the screen to select the data elements for group editing.

Delete Selection - deletes selected Elements.

Delete Domain - deletes the Domain.


Sewer
BASE, Base Data Scenario
Initialize Project
Project
Edit
View
Exchange
Scenario
Tools
Utilities
Help
Edit Selection Attributes...
Edit Domain Attributes...
Group Editing on Selection...
Group Editing on Domain...
DB Editor...
Delete Selection...
Delete Domain...
Delete Domain
Delete Domain

Data Elements

InfoSewer data can be broadly grouped as physical and non-physical objects. Physical objects refer to those network elements that can be visually displayed on a map in the InfoSewer workspace. A data element is any representative facility of a sewer system and, in InfoSewer a data element is either a pipe, pump, manhole, or wet well. These facilities are further classified as being either a link or a node. Click any of the links below to learn more.

Links - Any data element that begins and ends at a node element. Links convey water from one node (point) to another.

- **Pipes**
- **Pumps**

Nodes - A point in the network where links join together. A stationary facility identified by a single point (X,Y).

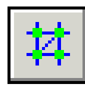
- **Manholes**
- **Wet Wells**

Note: All data elements have a unique 32 character alpha-numeric identification field.

Digitize Network

The Digitize Network command is most helpful when digitizing links and nodes in bulk. The order of the link and node creation does not matter when using the Digitize Network command. This method is most useful when creating a network "on the fly". Data corresponding to the different elements may be entered once the network is in place through the database tables.

Methodology

- Launch the **InfoSewer Edit Network** toolbar from the **View** menu **->** **Toolbars** command
- Choose the **Digitize Network**  icon from the InfoSewer **Edit Network** toolbar and click on it.
- Select an upstream node if one exists else left click on the part of the network where you wish to create one.
- Select intermediate vertices to identify link curvature by left clicking at various points.
- Finally double click to indicate the position of the end node. Continue the process till your entire network has been digitized or till you are satisfied with the digitization process.
- Once the digitization process has been completed, enter data into the element database tables by using the **DB Editor** command from the InfoSewer Control center -> InfoSewer button -> Edit pull-down menu.

Note: You may change the **Next Node Type** under the **Digitize** section of your **InfoSewer Control center -> InfoSewer** button **-> Tools** pull-down menu **-> Project Preferences** command -> **Operation Settings** tab.

Manholes

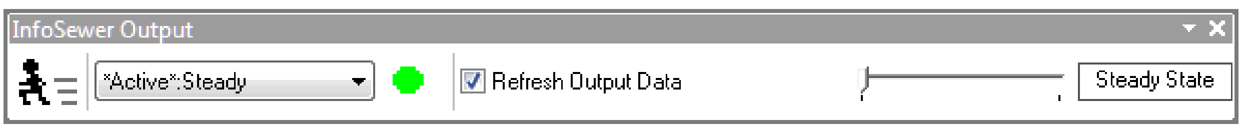
Manholes are junctions placed at the intersection of two or more links. Three types of manholes exist in a sewer collection system.

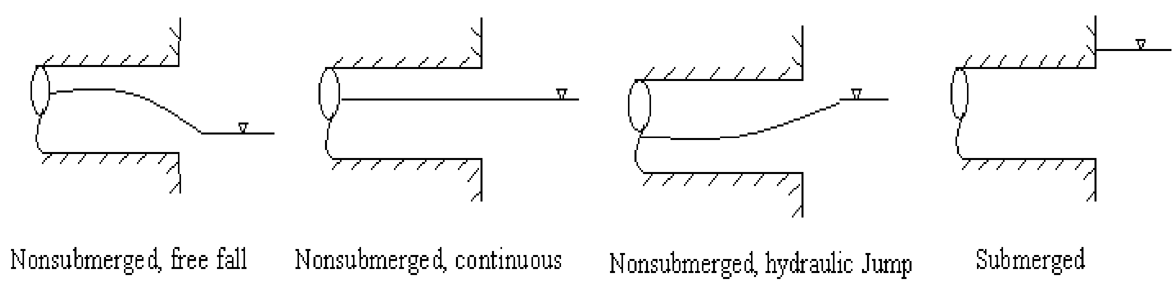
1. **Normal:** Where the physical location of a manhole exists in the collection system or a physical break is needed for a change in pipe attributes (e.g. diameter, material).
2. **Chamber:** A quasi-node placed to represent a break between a pump and a force main. No loads can be defined for a chamber.
3. **Outlet:** The facility where sewage flows exit the collection system. These nodes define the discharge end or the most downstream element of a sewer network. The exit condition can be grouped into four cases as shown below.

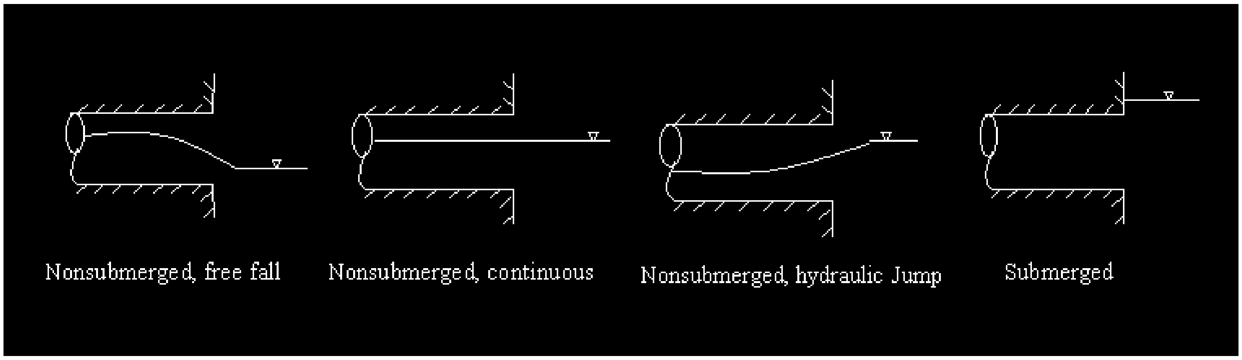
Nonsubmerged, free fall

Nonsubmerged, continuous

Nonsubmerged, hydraulic jump, Submerged







Manholes must also have their elevation above datum (e.g., sea level) specified in order for HGL profiles to be generated. Click here to view the fields required for a manhole.

How do I...

- **Create a manhole?**
- **Edit a manhole?**
- **Edit a group of manholes?**
- **Delete a manhole?**
- **Recall a deleted manhole?**

Create a Manhole

To create a manhole, click on the create node icon or from the **Create** menu, click **Add/Insert Manhole.**

Once the command is initiated, a crosshair cursor will appear. Select the location of the new node by left mousing clicking anywhere on the map display.

With the manhole placed, a dialog box will appear requesting the ID for the manhole, a description and the type of manhole being placed (Normal, Chamber, or Outlet). Once an ID, description and type are specified, press OK to end the command.

(Note: You can also use the **Digitize Network** command from the **Create** menu to create a pipe and node network at the same time. Once digitized, the user is able to add relevant modeling data in the Attribute Browser dialog box.)

Edit a Manhole

Graphic Edit - To edit the manhole graphically, use the Move Node icon to select the node. Once the node is selected, left mouse click on top of the node and hold the button and drag the mouse at the same time. A red rubber band will appear showing the user where the node will be located. Once the node is moved to its new location, release the mouse button. The node is now relocated. (Note: If the user has not enabled "Auto Length Calculation" under the **Tools** -> **Preferences** menu, all pipes connected to that node will not have their lengths automatically changed.)

Data Edit - To edit the data related to a node, first select the node using the Select icon. Once selected, the user is able to edit the database data related to the node under the Node tab of the Attribute Browser dialog box.

Edit the Data for a Group of Manholes

Group edits can be made by either creating a domain first and then selecting the Edit Domain Attributes icon, or the user can use the Edit Selection Attributes icon. Both icons are found on the Edit Network toolbar or can also be found under the **Edit** Menu.

Once the Edit Selection Attributes command is initialized, the user makes a selection and presses the <Enter> key. At this point, the Edit Attribute dialog box is displayed allowing the user to edit any field for any of the 4 data elements that may have been selected.

Delete a Manhole

To delete a manhole, from the **Edit** menu, select **Delete Node** or select the Delete Node icon. H2OMAP Sewer will prompt the user to confirm deletion. (Note: All links attached to the node being deleted will also be deleted.) The node and associated links are deleted graphically but can still be recalled from the database using the recall command. However, if Auto Database

Packing is enabled (Tools -> Preferences), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Manhole

To recall a deleted node, go to **Utilities** menu, point to **Recall**, then select **Node**. In order to perform the recall, the user must know the ID of the node that was deleted. If the ID is unknown, select the Show Deleted Nodes command from the Recall menu to see a list of ID's that have been deleted.

Other Related Topics - Data Elements, Manhole, Pipes, Pumps, Wet Well

---

Wet Wells

Wet wells are used by InfoSewer to model the storage of sewage water within a lift station. The capacity of a wet well is only evaluated during an EPS/Dynamic model run. The only facility that can leave a wet well is a pump.

How do I...

- Create a wet well?
- Edit a wet well?
- Delete a wet well?
- Types of Wet Wells
- Recall a deleted wet well?

Create a Wet Well

To create a wet well, click on the add wet well icon or from the **Create** menu, click **Add Wet Well.**

Once the command is initiated, a crosshair cursor will appear. Select the location of the new wet well by left mousing clicking anywhere on the map display.

Once the wet well has been located, a dialog box will become present requesting the ID number of the wet well and description for the wet well being created. Once these date are entered, press OK to end the command.

Types of Wet Wells

There are 2 different types of wet wells the user is able to input into InfoSewer.

Cylindrical -A cylindrical wet well has a constant diameter in relation to its high and low water levels. Via the Modeling option under the Attribute Browser, enter the following:

Bottom Elevation -The elevation (above sea level) of the bottom of the wet well, ft (m).
Initial Water Level - The height of water above the bottom of the wet well (ex. 12 ft.) when the simulation is to begin, ft (m).
Diameter - The diameter of the wet well, ft (m).
Maximum Level - The highest allowable water level (ex. 22 ft.), ft (m).
Minimum Level - The lowest allowable water level (ex. 3 ft.), ft (m)

Variable Area -A wet well where the volume changes with respect to height of sewage in the wet well. The only way to model a variable area wet well is to create a curve that represents the relationship between height and volume. Via the Modeling option under the Attribute Browser, enter the following:

Curve - The curve assigned to the wet well for an EPS. Click here to see an example curve.

There are 2 different types of wet wells the user is able to input into InfoSewer. Click on any wet well below to learn more.

0. *Cylindrical*
1. *Variable Area*

Edit a Wet Well

Graphic Edit - To edit the wet well graphically, use the Move Node icon to select the wet well. Once selected, left mouse click on top of the node and hold the button and drag the mouse at the same time. A red rubber band will appear showing the user where the node will be located. Once the node is moved to its new location, release the mouse button. The node is now relocated. (Note: If the user has not enabled "Auto Length Calculation" under the **Tools** -> **Preferences** menu, all pipes connected to that node will not have their lengths automatically changed.)

Data Edit - To edit the data related to a wet well, first select the node using the select tool. Once selected, the user is able to edit the database data related to the storage node under the Node tab of the Attribute Browser dialog box.

Delete a Wet Well

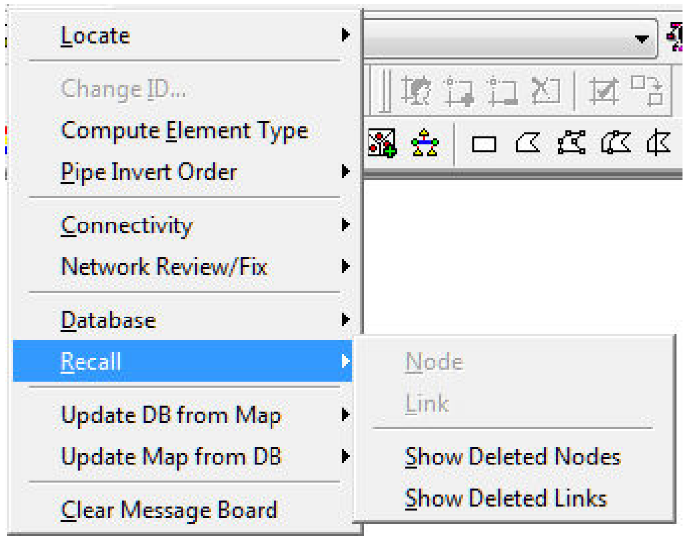
To delete a wet well, from the **Edit** menu, select **Delete Node** or select the Delete Node icon. InfoSewer will prompt the user to confirm deletion. (Note: All links attached to the node being deleted will also be deleted.) The node and associated links are deleted graphically but can still be recalled from the database using the recall command. However, if Auto Database Packing is enabled (Tools -> Preferences), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Wet Well

To recall a deleted wet well, go to **Utilities** menu, point to **Recall**, then select **Node**. In order to perform the recall, the user must know the ID of the node that was deleted. If the ID is unknown, select the Show Deleted Nodes command from the Recall menu to see a list of ID's that have been deleted.

The Recall command is used to restore/or undelete network components that have been deleted. This command will not work if the user has enabled Auto Database Packing option (Project Preferences). Recall is found under

Utilities / Recall. Recalled components will be redrawn with their graphical properties (node size, pipe connectivity and shape) at the time of deletion. All database values assigned to those components will be restored.



Other Related Topics - Data Elements, Manhole, Pipes, Pumps, Wet Well

---

Pipes

A pipe is a link that conveys water from one node to another. Click here to view the fields required for populating a pipe.

How do I...

- **Create a pipe?**
- **Edit a pipe?**
- **Edit the data for a group of pipes?**
- **Delete a pipe?**
- **Recall a deleted pipe?**
- **Redraw a pipe?**

Create a Pipe

To create a pipe, click on the create link icon or from the **Create** menu, click **Add Pipe**.

Once the command is initiated, a pencil cursor will appear. Select the *From Node* for which the pipe is to be placed and begin digitizing the location of the pipe. Intermediate shape-defining vertices can be placed by clicking the mouse in any location. Configure the pipe accordingly and double-click the mouse on top of the *To Node* to end the digitization process.

Once the pipe has been located, a dialog box will become present requesting the ID number of the pipe and a description. Once an ID and description are entered, press OK to end the command.

(Note: The pipe being added must be snapped to an existing node. Use the **Digitize Network** command from the **Create** menu if the nodes to which the pipe will be connected do not yet exist. Once digitized, the user is able to add relevant modeling data in the Attribute Browser dialog box.)

Edit a Pipe

Graphic Edit - To edit the pipe graphically, select the Edit Link Vertex icon and click on the pipe or from the **Edit** menu, select the **Edit Link Vertex** command. To add a vertex, click on the pipe two times - one to select the pipe, the other to add a vertex. Once a vertex is added the selected vertex will turn blue. By clicking on the blue box and holding down the left mouse button, the user can drag the vertex to its new location. If the user has enabled "Auto Length Calculation" under the **Tools** -> **Preferences** menu, then any graphical changes will be recorded as having new lengths.

Data Edit - To edit the data related to a pipe, first select the pipe using the select tool. Once selected, the user is able to edit the database data related to the pipe under the Link tab of the Attribute Browser dialog box.

Edit the Data for a Group of Pipes

Group edits can be made by either creating a domain first and then selecting the Edit Domain Attributes icon, or the user can use the Edit Selection Attributes icon. Both icons are found on the Edit Network toolbar or can also be found under the Edit Menu.

Once the Edit Selection Attributes command is initialized, the user makes a selection and presses the <Enter> key. At this point, the Edit Attribute dialog box is displayed allowing the user to edit any field for any of the 4 data elements that may have been selected.

Delete a Pipe

To delete a pipe, from the **Edit** menu, select **Delete Link**. From here the user selects the desired pipe to be deleted and is asked by H2OMAP Sewer to confirm the deletion of this pipe. This pipe is deleted graphically but can still be recalled from the database using the recall command. However, if Auto Database Packing is enabled (Tools -> Preferences), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Pipe

To recall a deleted pipe, go to **Utilities** menu, point to **Recall**, then select **Link**. In order to perform the recall, the user must know the ID of the pipe that was deleted. If the ID is unknown, select the Show Deleted Links command from the Recall menu to see a list of ID's that have been deleted.

Redraw a Pipe

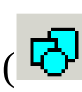
Redrawing a pipe is the same as modifying its geometry. To redraw a pipe, select the Redraw Link icon or from the **Edit** menu, select the **Redraw Link** command. Once initialized, the mouse pointer will change to a crosshair. At this point the user is able to redigitize the new location of the pipe currently highlighted. This command is dependent upon highlighting the pipe in question first, then changing its geometry via this command.

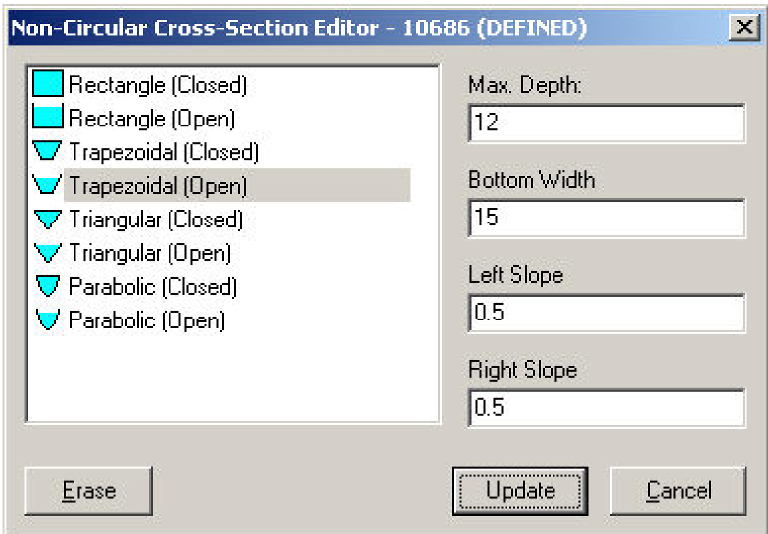
Specify Channel Shape

InfoSewer supports modeling of wide varieties of channel shape types including circular, trapezoidal, rectangular, triangular, square, parabolic, etc. The conduits could be closed type (pipes), or could be open to atmosphere. **When conduits are created, the model considers the shape as circular**.

If the desired shape is non-circular type, the user should specify the shape and the corresponding input parameters (e.g., maximum depth, top/bottom width, and/or side slopes). Shape and size specification of non-circular conduits could be performed using either of the following two methods.

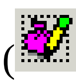
For one channel at a time

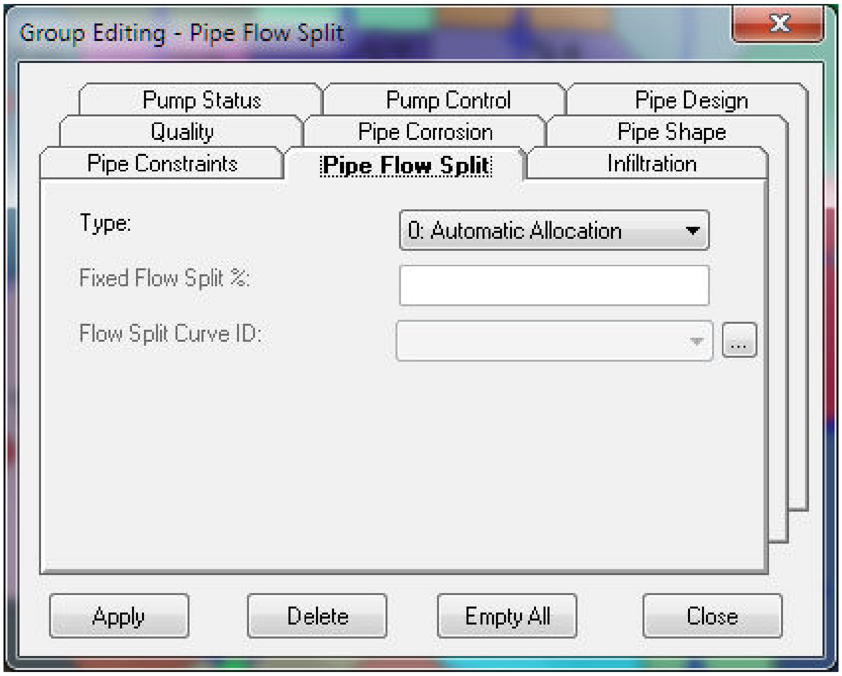
- Select the desired channel and click on the channel shape icon () located at the top right corner of the attribute browser.
- The above action initiates the non-circular cross-section editor dialog box shown below. Here the desired shape could be selected, and the corresponding input parameters could be edited.



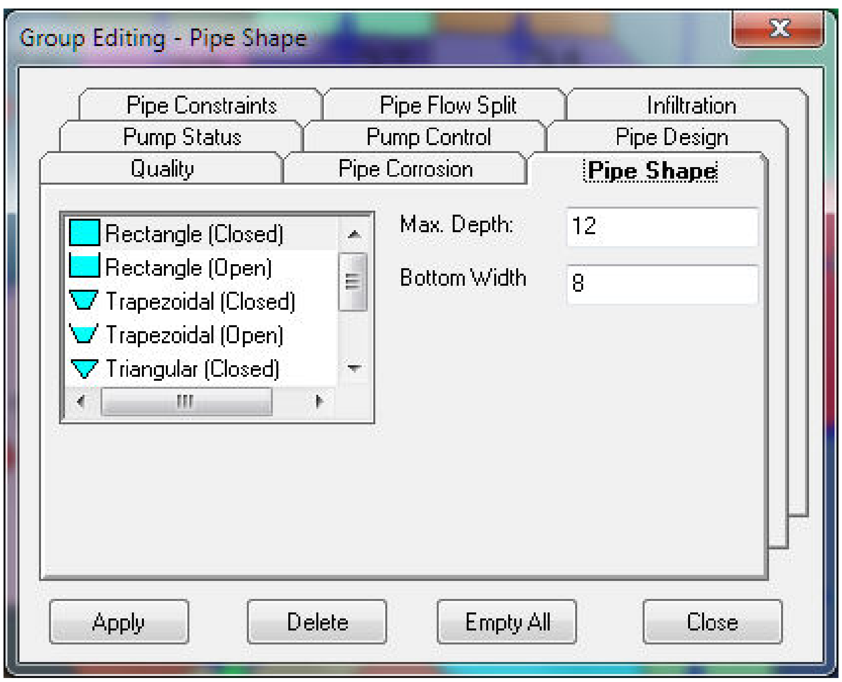
- Repeat the above two steps for all non-circular conduits in the collection system.

Group editing

- Using this option, one can simultaneously supply the desired data for multiple conduits of same shape and size. This could be done using either of the Group Edit on Selection () or the Group Edit on Domain () features. Once the desired conduits are selected and  is clicked, or the conduits are grouped into a domain and  is clicked, the following dialog box would appear.



- One here, one may click on "Pipe Shape" button which would automatically activate the following dialog box.



- Select the desired shape and specify the corresponding inputs. Note that by doing the above three steps, you are implying that all the conduits selected (in the domain) have the same shape and have the same sizes (i.e., maximum depth, top/bottom width, and side slopes).

Other Related Topics - Data Elements, Manhole, Pipes, Pumps, Wet Well

---

Pumps

A pump is a link that imparts energy to a fluid thereby raising its hydraulic head. The relationship describing the head imparted to a fluid as a function of its flow rate through the pump is termed the pump characteristic curve.

In a sewer collection system, pumps are only allowed in conjunction with wet wells. Therefore, a wet well and chamber (manhole) must first be digitized prior to creating a pump.

How do I...

- **Create a pump (types of pumps)?**
- **Edit a pump?**
- **Delete a pump?**
- **Recall a deleted pump?**
- **Redraw a pump?**
- **Create Pumps in Parallel?**

Create a Pump

To create a pump, click on the add pump link icon or from the **Create** menu, click **Add Pump**.

Once the command is initiated, a pencil cursor will appear. Select the *Wet Well* for which the pump is to be placed and begin digitizing the location of the pump. Intermediate shape-defining vertices can be placed by clicking the mouse in any location. Configure the pump accordingly and double-click the mouse on top of the *Chamber* to end the digitization process.

Once the pump has been located, a dialog box will become present requesting the ID number of the pump, a description and the type of pump being created. Once these data are entered, press OK to end the command.

Types of Pumps

There are 3 different types of pumps the user is able to input into H2OMAP Sewer. Click on any pump to learn more.

0. ***Fixed Capacity***
1. ***Design Point Curve***
2. ***Exponential 3-Point Curve***

Edit a Pump

Graphic Edit - To edit the pump graphically, select the Edit Link Vertex icon and click on the pump or from the **Edit** menu, select the **Edit Link Vertex** command. To add a vertex, click on the pump two times - one to select the pump, the other to add a vertex. Once a vertex is added the selected vertex will turn blue. By clicking on the blue box and holding down the left mouse button, the user can drag the vertex to its new location.

Data Edit - To edit the data related to a pump, first select the pump using the select tool. Once selected, the user is able to edit the database data related to the pump under the Link tab of the Attribute Browser dialog box.

Delete a Pump

To delete a pump, from the **Edit** menu, select **Delete Link**. From here the user selects the desired pump to be deleted and is asked by H2OMAP Sewer to confirm the deletion of this pump. This pump is deleted graphically but can still be recalled from the database using the recall command. However, if Auto Database Packing is enabled (Tools -> Preferences), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Pump

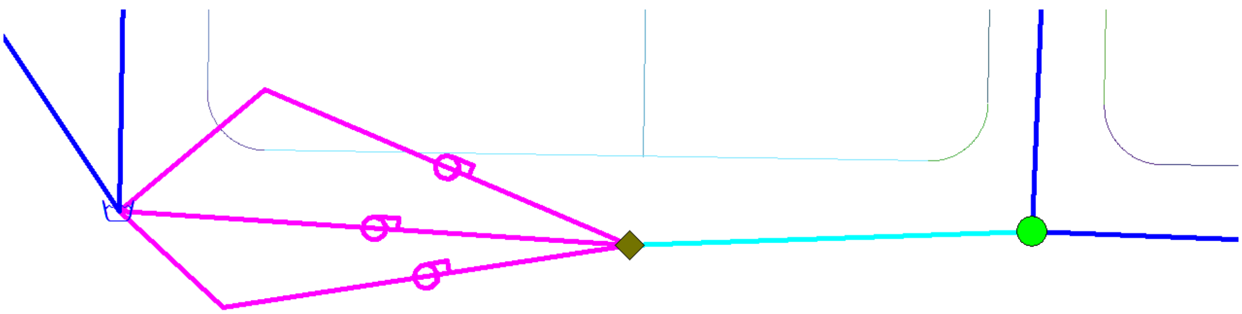
To recall a deleted pump, go to **Utilities** menu, point to **Recall**, then select **Link**. In order to perform the recall, the user must know the ID of the pump that was deleted. If the ID is unknown, select the Show Deleted Links command from the Recall menu to see a list of ID's that have been deleted.

Redraw a Pump

Redrawing a pump is the same as modifying its geometry. To redraw a pump, select the Redraw Link icon or from the **Edit** menu, select the **Redraw Link** command. Once initialized, the mouse pointer will change to a crosshair. At this point the user is able to redigitize the new location of the pump currently highlighted. This command is dependent upon highlighting the pump in question first, then changing its geometry via this command.

Create Pumps in Parallel

H2OMAP Sewer allows you to model an unlimited number of pumps in parallel as shown in the figure below. Each pump may have unique properties.



Pumps in parallel of the same type (i.e. all fixed capacity pumps or pumps described by characteristic curves) can be represented with one pump by specifying the number of parallel pumps in the *PARALLEL* field of the attribute browser. Pumps described by characteristic curves can be of exponential or design point type.

For Extended Period and Dynamic Simulations (EPS/Dynamic), InfoSewer allows you to control the on-off status of each pump based on the upstream wet-well levels or volumes.

Other Related Topics - Data Elements, Manhole, Pipes, Pumps, Wet Well

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

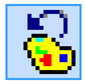
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

View Menu

Descriptions of All View Menu commands in InfoSewer.

The InfoSewer **Map Display** feature may be used to annotate your map. To learn more about color coding and/or changing symbol sizes click here.

Choose the **Reset Display** icon  to clear your Map.

Refresh Map resets your or others changes to the Map.

ID Labeling for Nodes and Links

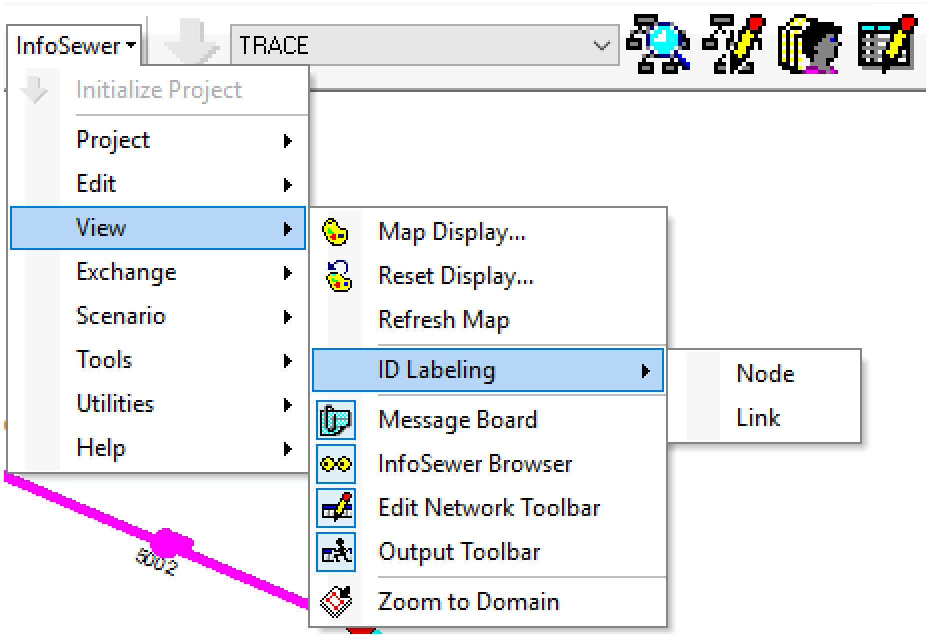
Message Board shows or does not show the **Message Board**.

InfoSewer Browser toggles the showing of the **InfoSewer Attribute Browser**.

Edit Network Toolbar toggles the showing of the **Edit Network Toolbar**.

Output Toolbar toggles the showing of the **Output Toolbar**.

Zoom to Domain Zooms your View of the Map to the Extents of the Current Domain.



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

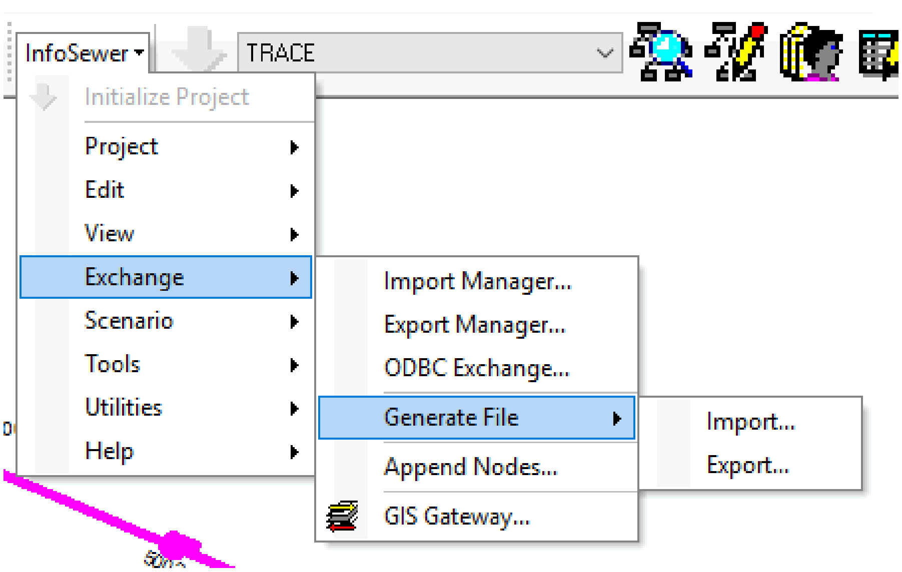
More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Exchange Menu

Descriptions of All *Exchange Menu commands in InfoSewer.*

InfoSewer supports the following data exchange functionality.

- Import/Export Managers - Background mapping, external models or InfoSewerdata elements.
- Import/Export Generate Files - Import generate files in to InfoSewer or export InfoSewerin to generate file.
- Append Nodes - Allows the user to create a conduit and node network from third party software application like AutoCAD, Microstation, or ESRI shapefiles.
- ODBC Exchange - Download and upload data to and from InfoSewer by creating data links.
- GIS Gateway - Allows exchange of Geodatabase information to and from the model.
- Windows Clipboards - Export and view model results outside of InfoSewer.



Data Exchange

InfoSewer has an extensive data exchange functionality. Using InfoSewer data exchange features one can import collection system maps and the associated data created by H2OMap Sewer; export collection system maps and the associated data created by InfoSewer into H2OMAP Sewer; import/export input data and simulation outputs from/to data formats that are compatible with GIS, CAD, CSV; export/import collection system geometry to/from ArcInfo generate files, and exchange model outputs with other applications including presentation applications, spreadsheets, and word processors.

.

InfoSewer supports the following data exchange functionality.

- Import/Export Managers - Background mapping, external models or InfoSewerdata elements.
- Import/Export Generate Files - Import generate files in to InfoSewer or export InfoSewerin to generate file.
- Append Nodes - Allows the user to create a conduit and node network from third party software application like AutoCAD, Microstation, or ESRI shapefiles.
- ODBC Exchange - Download and upload data to and from InfoSewerby creating data links.
- GIS Gateway - Allows exchange of Geodatabase information to and from the model.
- Windows Clipboards - Export and view model results outside of InfoSewer.
- Import H2OMap Sewer - Import H2OMap Sewer Model into InfoSewer.
- Export to H2OMAP Sewer - Export to H2OMAP Sewer

In addition to the data exchange features supported by InfoSewer, the following database connection and exchange features are available from

ArcGIS.

- Spatial Database Connection
- OLE Database Connection

Issues to Consider When Importing GIS data into InfoSewer

If you import data from a GIS you should ensure the following before importing into InfoSewer:

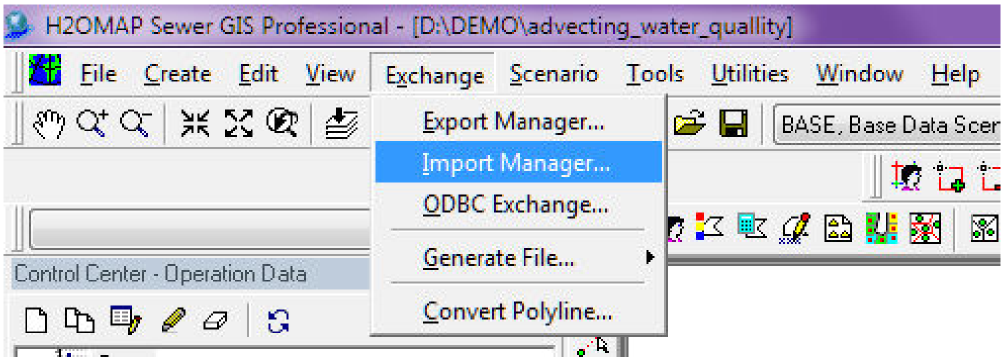
- All conduits (arcs, lines) in the GIS are connected to two unique nodes.
- All conduits in the GIS are snapped together where those conduits represent connected conduits in your sewer collection system. In other words, be sure that where two or more conduits should be connected, those conduits should share the same node in the GIS or other external database.
- There are no disconnected nodes in the GIS database. Each node should be connected to at least one conduit in the GIS.
- The graphical representation of the conduits in the GIS corresponds to the values in your database that represent the from- and to-node identifiers. In other words, if a conduit in your GIS database has a from-node value of "1" and a to-node value of "2", then the graphical representation of that conduit on the map display shows that conduit starting at node "1" and ending at node "2".

Note:

- If you do not meet the above-listed criteria, you may experience one or more errors when running a simulation in InfoSewer. The two most common situations that may occur if you do not meet the above-listed criteria before importing are the following:
  - Isolated Node – One or more nodes are disconnected from the network.
  - Coincident Nodes – Two or more nodes share the same location, appearing to a viewer as a single node.
- InfoSewer has tools that will help you correct these problems in the event that your GIS is too burdensome. From the **InfoSewer Control**

Center -> InfoSewer button **-> Utilities** menu, point to **Connectivity** and select **Orphan Nodes** or **Orphan conduits**. By doing this, InfoSewer will identify which nodes are not connected to conduits and which conduits do not have both an upstream and downstream node.

The GIS Gateway is an integral part of InfoSewer



Import Manager

All other data elements are imported with the Import Manager. Here the user can import from either an ESRI Shapefile, MapInfo MIF/MID or an ASCII delimited text file.

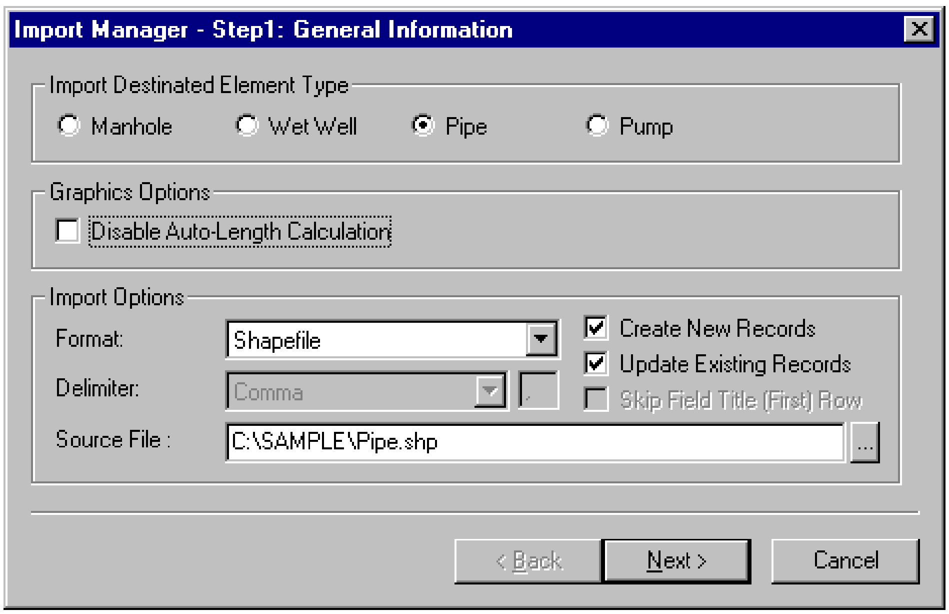
To run the Import Manager, from the **Exchange** menu, select **Import Manager**.

The Import Manager itself is a two-step wizard that guides the user through the import process.

Step 1 – General Information

The first step of an import is to identify which InfoSewer element and import options are desired.

Click on any portion of the dialog box below to learn more.



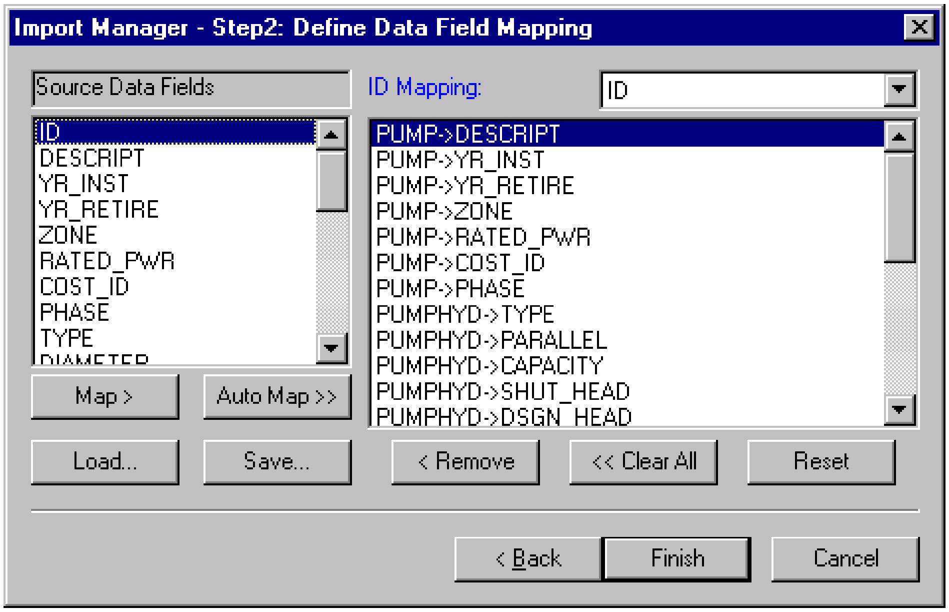
Choose the desired options and then choose the Next button to continue to step 2.

Note: The checkbox for "No Geometry Update" is enabled when you select "Shapefile" or "MIF/MID" format because these two formats have geometry information, but "Delimited Text" does not.

Step 2 – Define Data Field Mapping

The second step is to define which fields will be mapped to their respective InfoSewer database fields.

Click on any portion of the dialog box below to learn more.



Once all of the database fields have been mapped, select the Finish button to complete the import process.

(Note: Be sure to correctly map the ID field, which contains the unique identifier for each H2OMAP Sewer component. This ID field must also be present in the file being imported, since it is the only unique identifier in the database.)

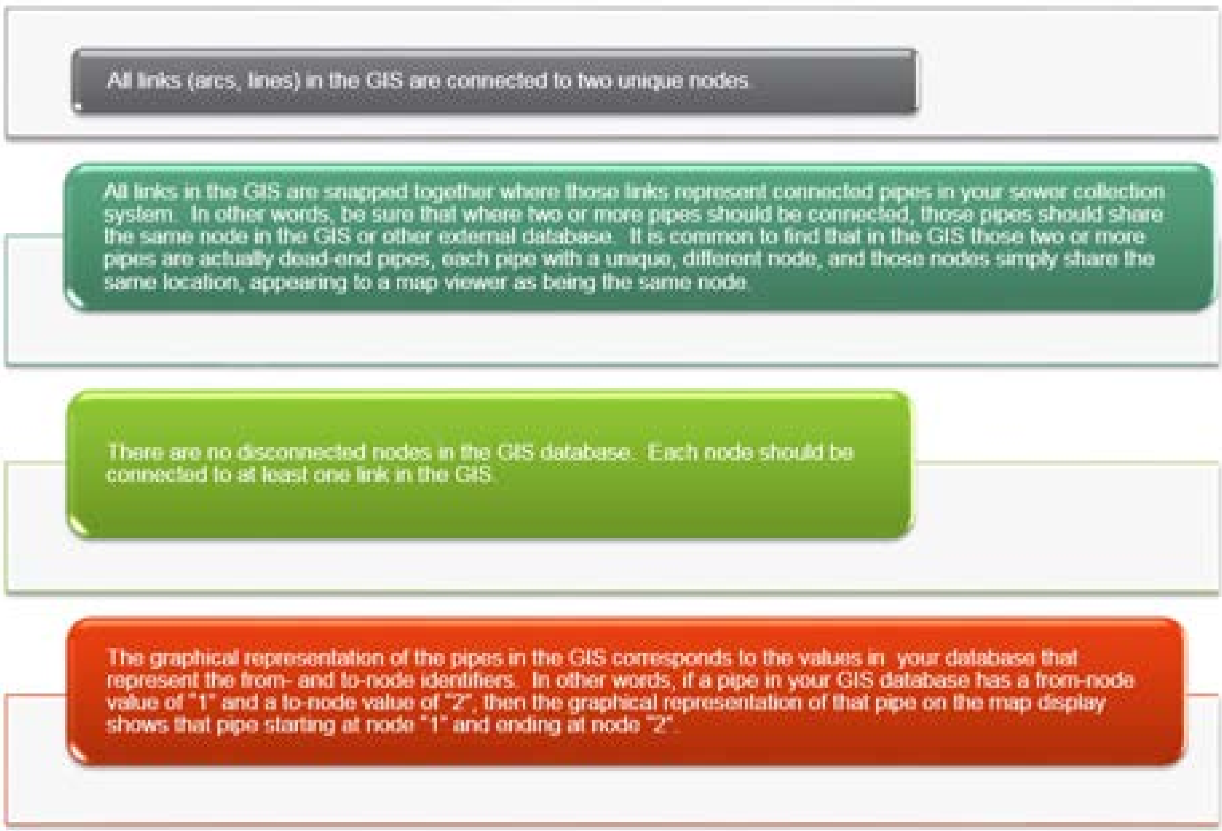
Issues to Consider When Importing GIS data into H2OMAP Sewer

If you import data from a GIS you should ensure the following before importing into H2OMAP Sewer:

1. All links (arcs, lines) in the GIS are connected to two unique nodes.
2. All links in the GIS are snapped together where those links represent connected pipes in your sewer collection system. In other words, be sure that where two or more pipes should be connected, those pipes should share the same node in the GIS or other external database. It is common to find that in the GIS those two or more pipes are actually dead-end pipes, each pipe with a unique, different node, and those

nodes simply share the same location, appearing to a map viewer as being the same node.

3. There are no disconnected nodes in the GIS database. Each node should be connected to at least one link in the GIS.
4. The graphical representation of the pipes in the GIS corresponds to the values in your database that represent the from- and to-node identifiers. In other words, if a pipe in your GIS database has a from-node value of “1” and a to-node value of “2”, then the graphical representation of that pipe on the map display shows that pipe starting at node “1” and ending at node “2”.



(Note: If you do not meet the above-listed criteria, you may experience one or more errors when running a simulation in H2OMAP Sewer.)

The two most common situations that may occur if you do not meet the above-listed criteria before importing are the following:

1. Isolated Node – One or more nodes are disconnected from the network.

2. Coincident Nodes – Two or more nodes share the same location, appearing to a viewer as a single node.

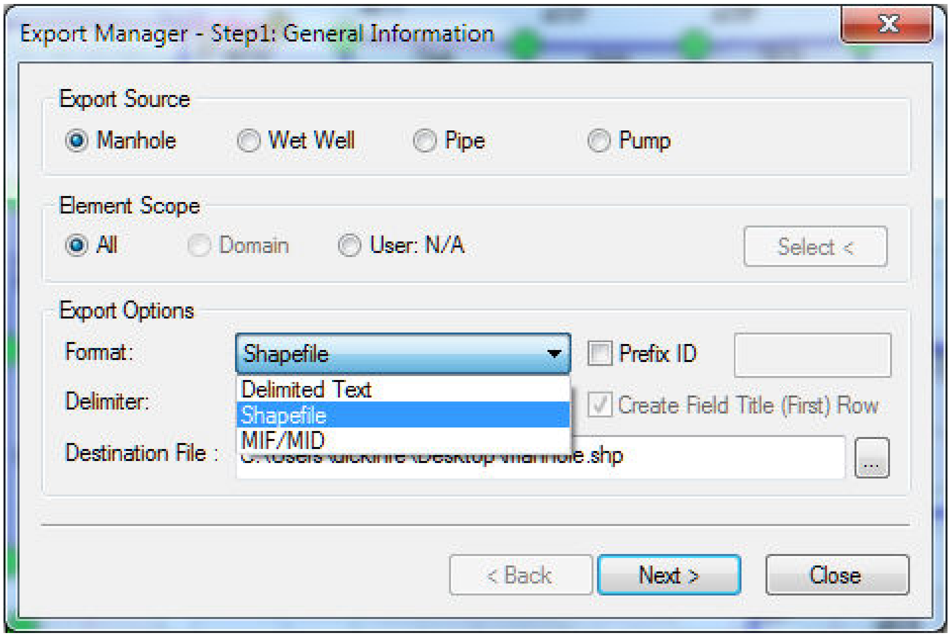
InfoSewer has tools that will help you correct these problems in the event that your GIS is too burdensome. From the **Utilities** menu, point to **Connectivity** and select **Orphan Nodes** or **Orphan Links**. By doing this, the program will identify which nodes are not connected to links and which links do not have both an upstream and downstream node. Once these facilities are identified, you can then correct the errors by moving nodes or redrawing the pipes.

At any point, the user can also update the graphics from the database data and vice versa. Click here to learn more about the utilities within InfoSewer.

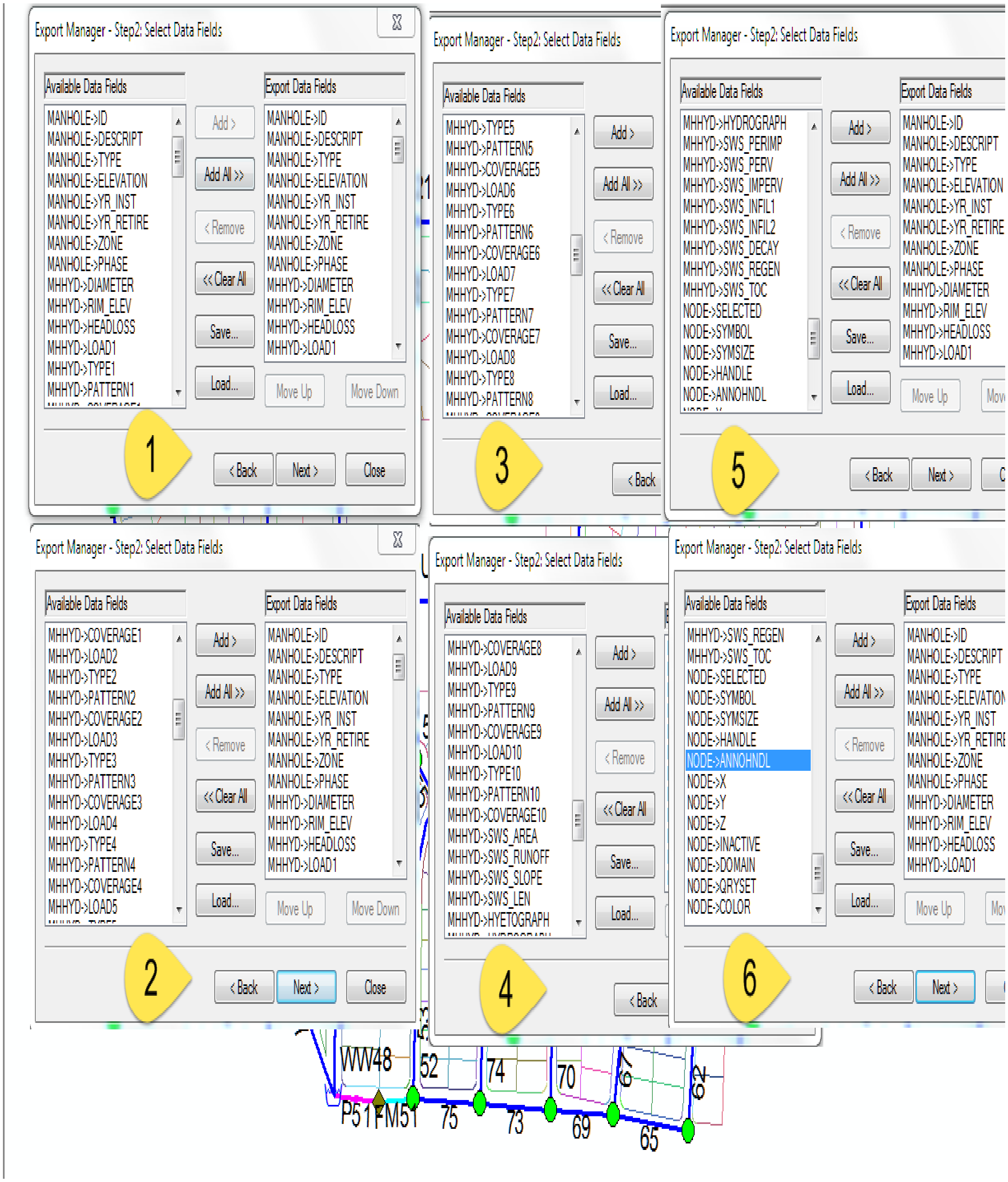
This will prove useful for sharing your modeling efforts with your organization via enterprise or web (Intranet or Internet) applications.

Export Options are:

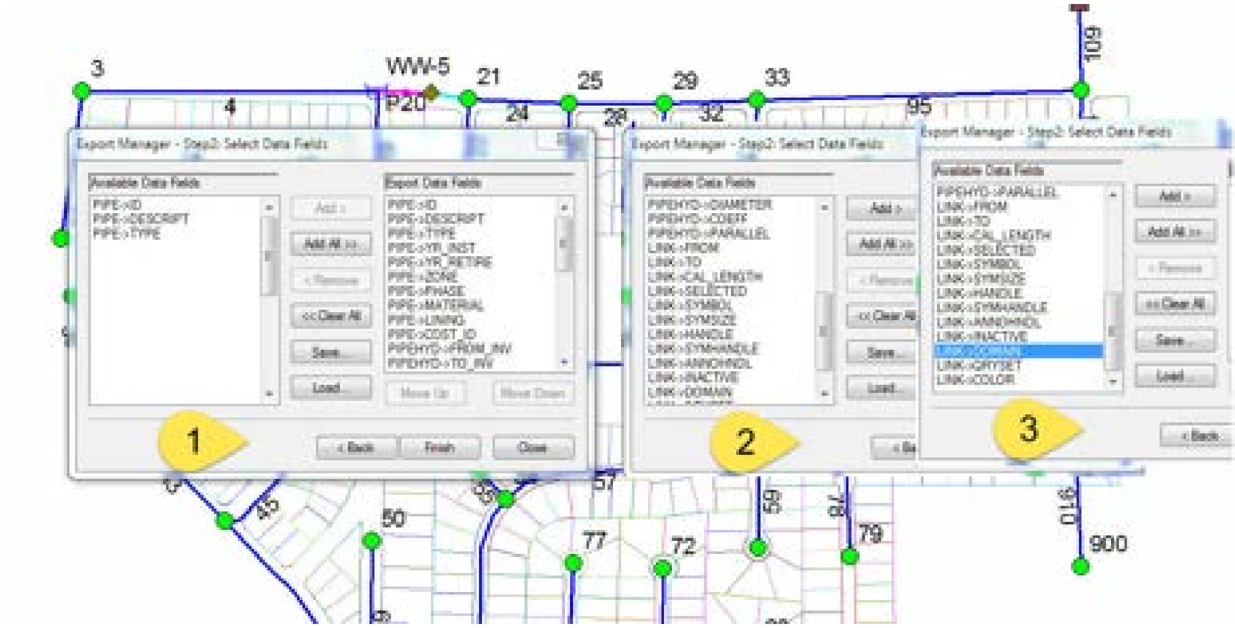
- CSV Files The checkbox for “No Geometry Update” is enabled when you select “Shapefile” or “MIF/MID” format because these two formats have geometry information, but “Delimited Text” does not.
- Shapefiles The checkbox for “No Geometry Update” is enabled when you select “Shapefile” or “MIF/MID” format because these two formats have geometry information, but “Delimited Text” does not.
- MIIF/MID The checkbox for “No Geometry Update” is enabled when you select “Shapefile” or “MIF/MID” format because these two formats have geometry information, but “Delimited Text” does not.
- for Pumps, PIpes, Wet Wells and Manholes



Manhole Export and Import

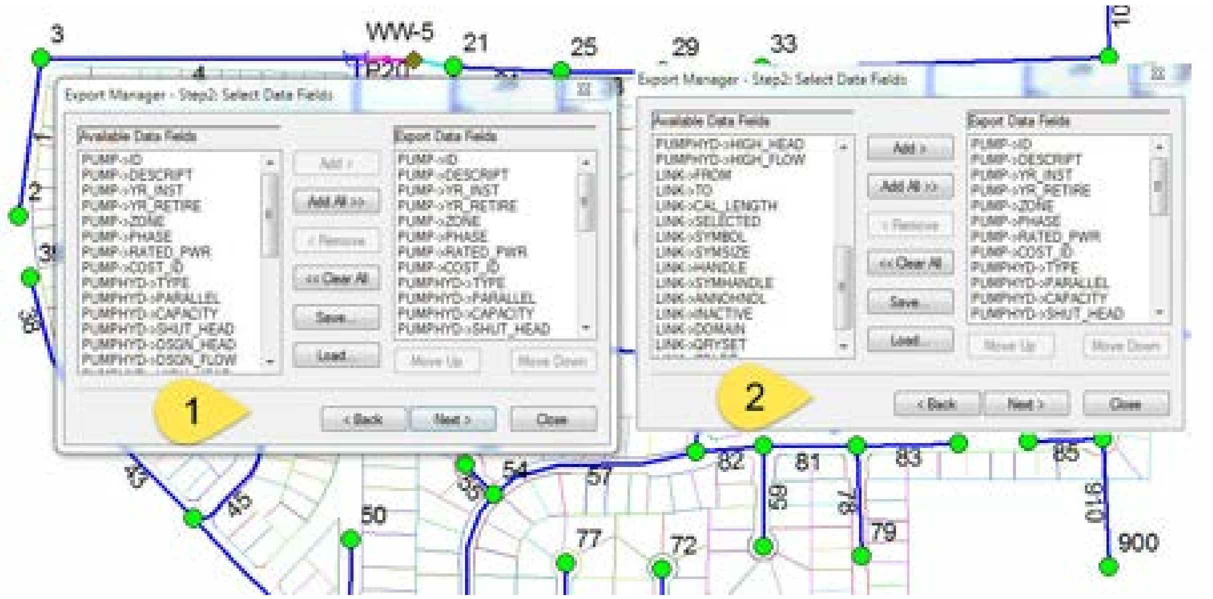


Pipe Export and Import



[Wet Well Export and Import]()

[Pump Export and Import]()


Export Manager - Step2: Select Data Fields
Available Data Fields
PUMP->ID
PUMP->DESCRIPT
PUMP->YR_INST
PUMP->YR_RETIRE
PUMP->ZONE
PUMP->PHASE
PUMP->RATED_PWR
PUMP->COST_ID
PUMPHYD->TYPE
PUMPHYD->PARALLEL
PUMPHYD->CAPACITY
PUMPHYD->SHUT_HEAD
PUMPHYD->DSGN_HEAD
PUMPHYD->DSGN_FLOW
Add >
Add All >>
< Remove
<< Clear All
Save...
Load...
Export Data Fields
Move Up
Move Down
< Back
Next >
Close
1
Export Manager - Step2: Select Data Fields
Available Data Fields
PUMPHYD->HIGH_HEAD
PUMPHYD->HIGH_FLOW
LINK->FROM
LINK->TO
LINK->CAL_LENGTH
LINK->SELECTED
LINK->SYMBOL
LINK->SYMSIZE
LINK->HANDLE
LINK->SYMHANDLE
LINK->ANNOHNDL
LINK->INACTIVE
LINK->DOMAIN
LINK->QRYSET
2

Export Manager

InfoSewer is able to export any data element or model result at any time.

Click on any hyperlink below to learn more.

- Export Data Elements (Pipes, Manholes, etc.)
- Export Manager
- Export Model Results

Export Data Elements

There are several ways to export data elements, depending on the type of file that is to be exported.

ArcInfo Generate File

If the export file is an ArcInfo Generate file, then from the **Exchange** menu, point to **Generate** then select **Export**. At this point the user will be prompted to select the elements to be mapped (links and/or nodes) and the location to write the export files. It is important to note that when generate files are created, they do not contain the database information from H2OMAP Sewer. To generate database information, the user must also export a delimited text file (CSV) for import into ArcInfo.

Export Manager

All other data elements are exported through the Export Manager. Here the user can export to either an ESRI Shapefile, MapInfo MIF/MID or an ASCII delimited text file.

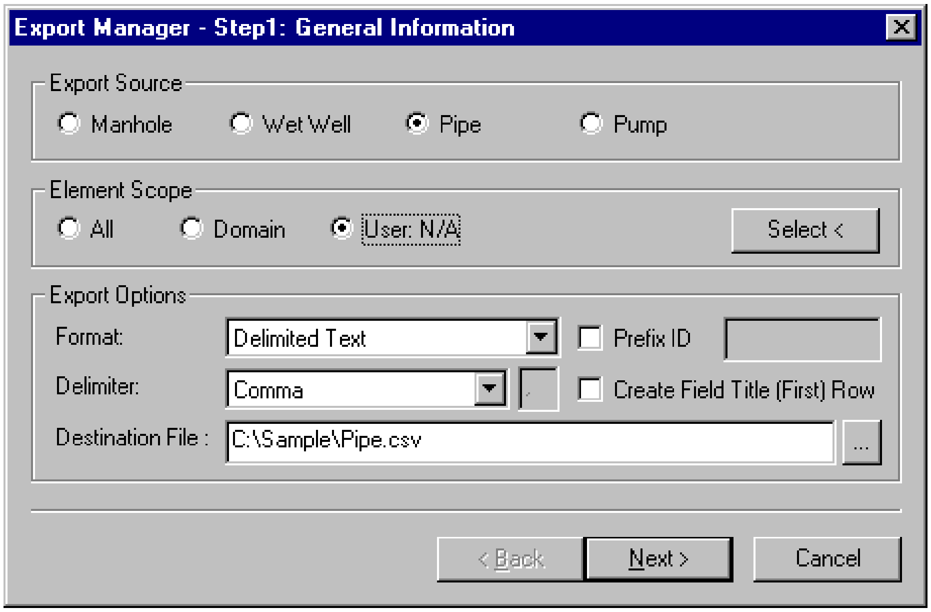
To run the Export Manager, from the **Exchange** menu, select **Export Manager**.

The Export Manager itself is a three-step wizard that guides the user through the export process.

Step 1 – General Information

The first step of an export is to identify whichIn InfoSewer element, scope, output format and destination file are desired.

Click on any portion of the dialog box below to learn more.

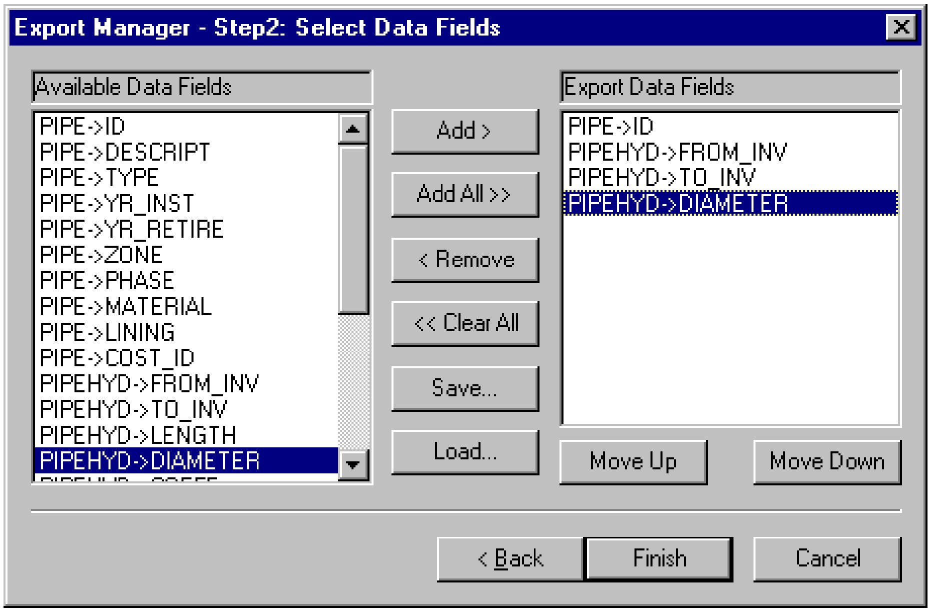


Choose the desired options and then choose the Next button to continue to step 2.

Step 2 – Define Display Fields

The second step is to define which fields will be included in the export file and the order in which they will be created. Field prefixes (ex. MANHOLE:) reflect the component types. The user also has the option of saving a custom configuration for later use.

Click on any portion of the dialog box below to learn more.

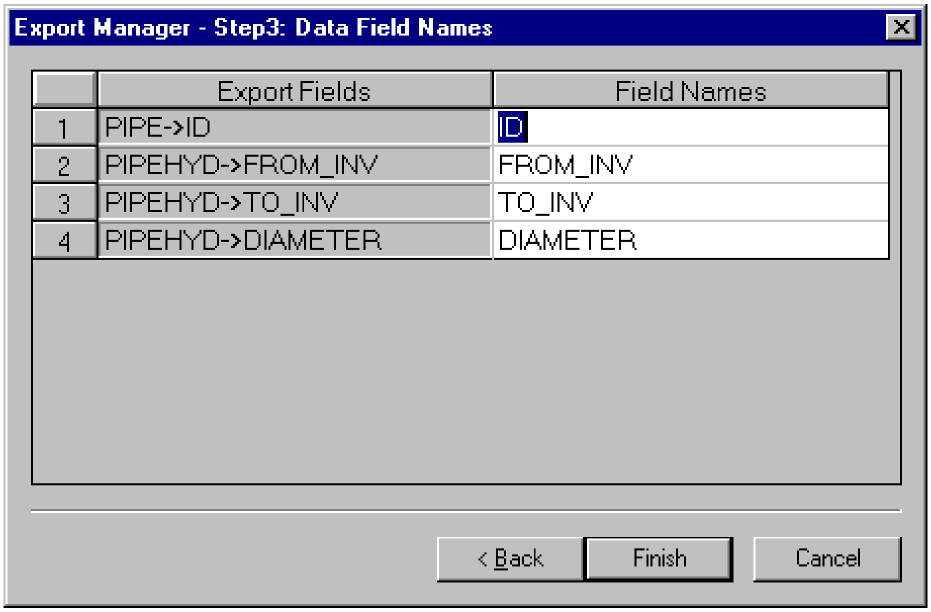


Choose the desired fields and then choose the Next button to continue to step 3.

Step 3 – Define Display Scope (ESRI and MapInfo Only)

The third step is only applicable to ESRI Shapefile and MapInfo MIF/MID export files. Here the user is able to customize the database field names created in the export file by edit their names.

The Export Fields shows the user which database fields were chosen to be exported while the Field Names represents the database field header assigned to the new export file. The user is able to click on any field name and change the name prior to the export file being created.



Once the finish button is clicked, an export file is created. The export file can now be edited by third party software programs.

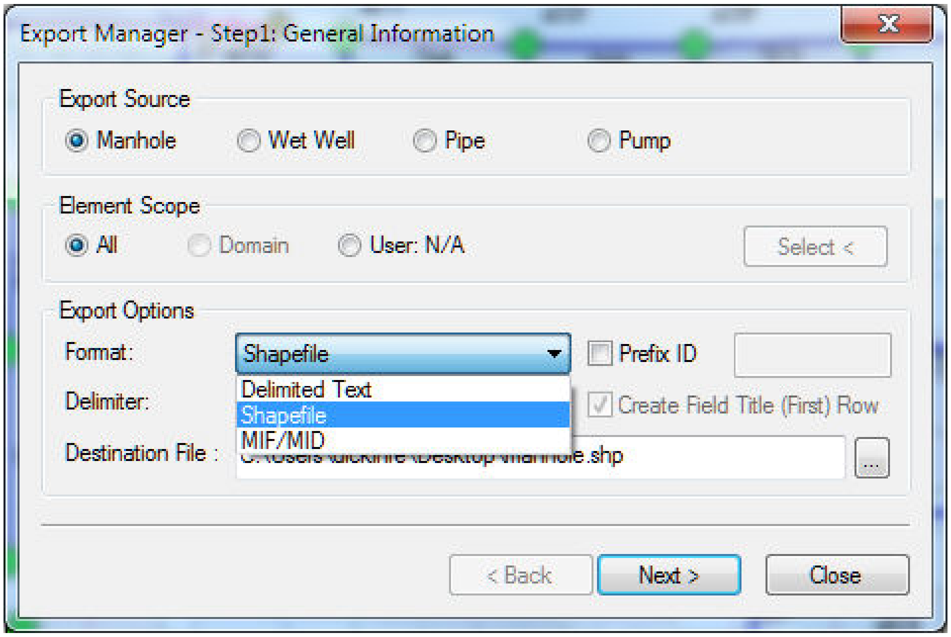
Exporting Model Results

Because InfoSewer is Windows based, any simulation report or database table that resides in a project is capable of being copied at any time. Just open the desired output report or database table and highlight the data you wish to export. Then, right-mouse click over the highlighted area and select Copy. Open any third party package like Microsoft Excel and select Paste from the Edit menu.

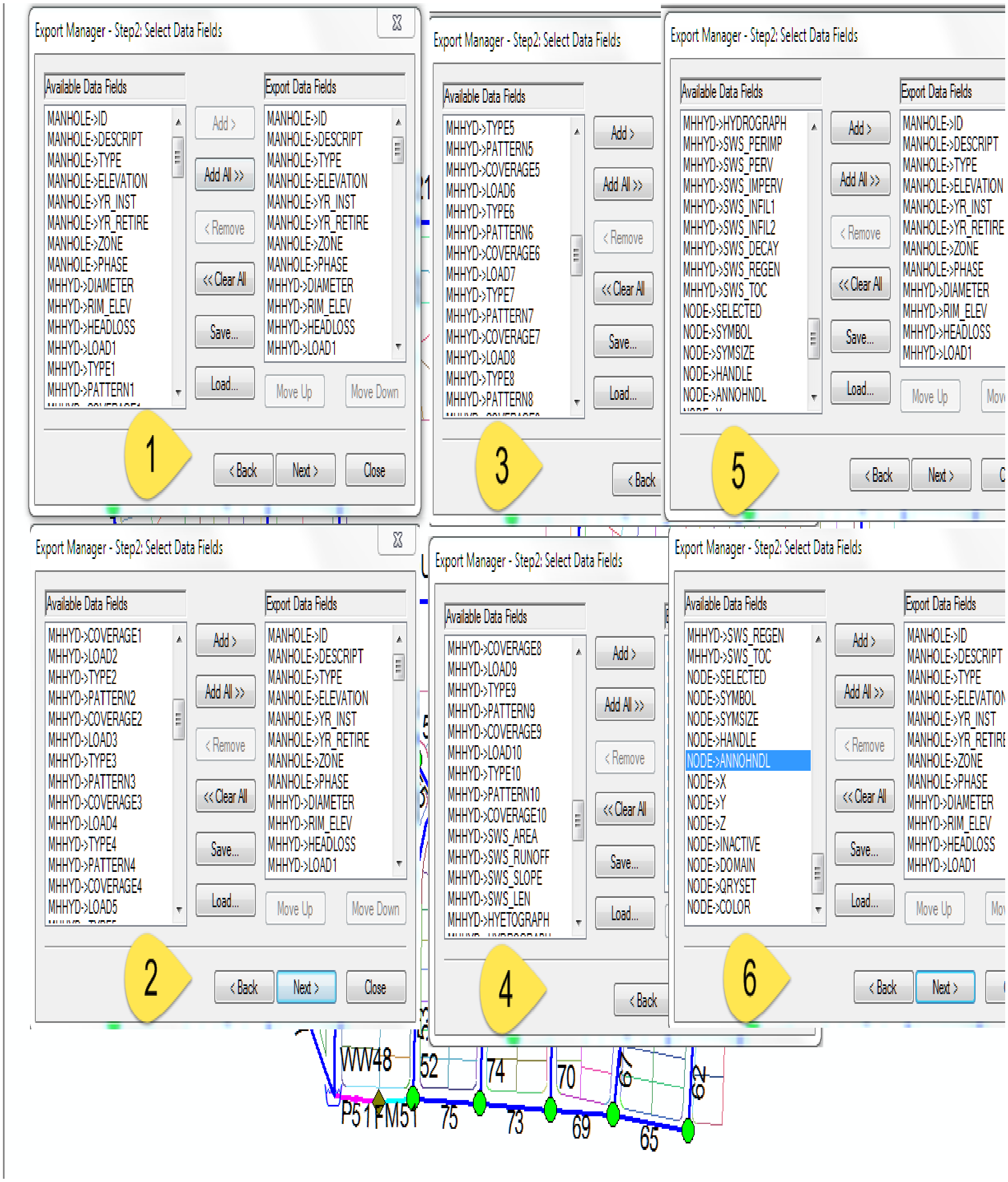
This will prove useful for sharing your modeling efforts with your organization via enterprise or web (Intranet or Internet) applications.

Export Options are:

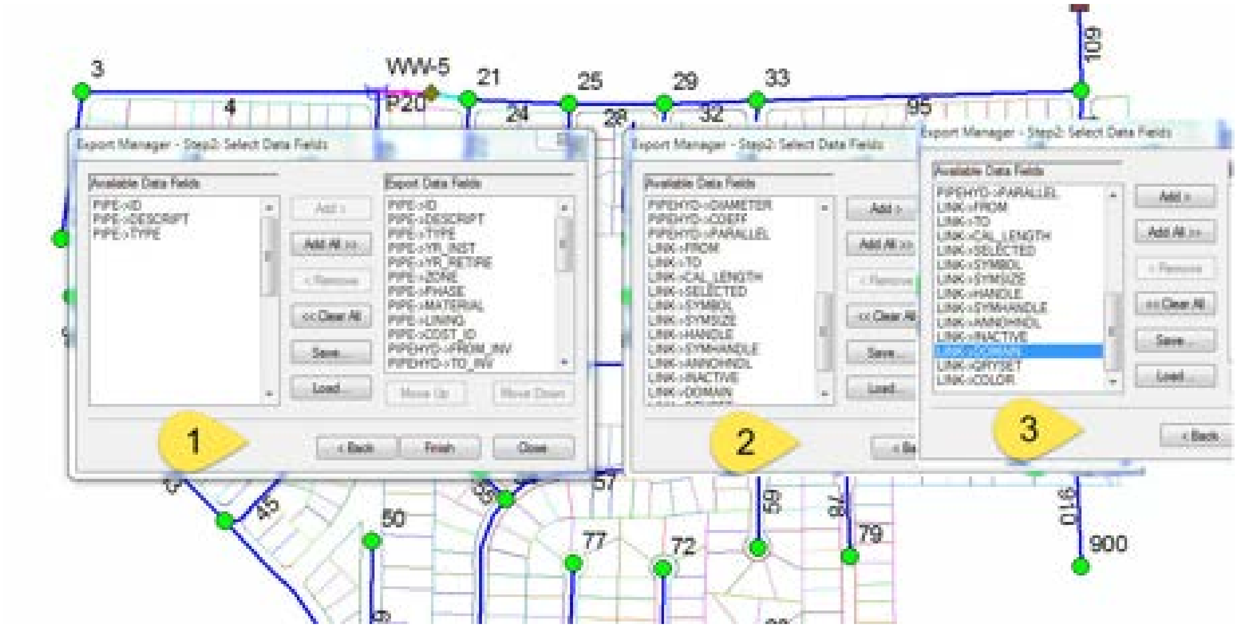
- CSV Files
- Shapefiles
- MIIF/MID
- for Pumps, PIpes, Wet Wells and Manholes



Manhole Export and Import

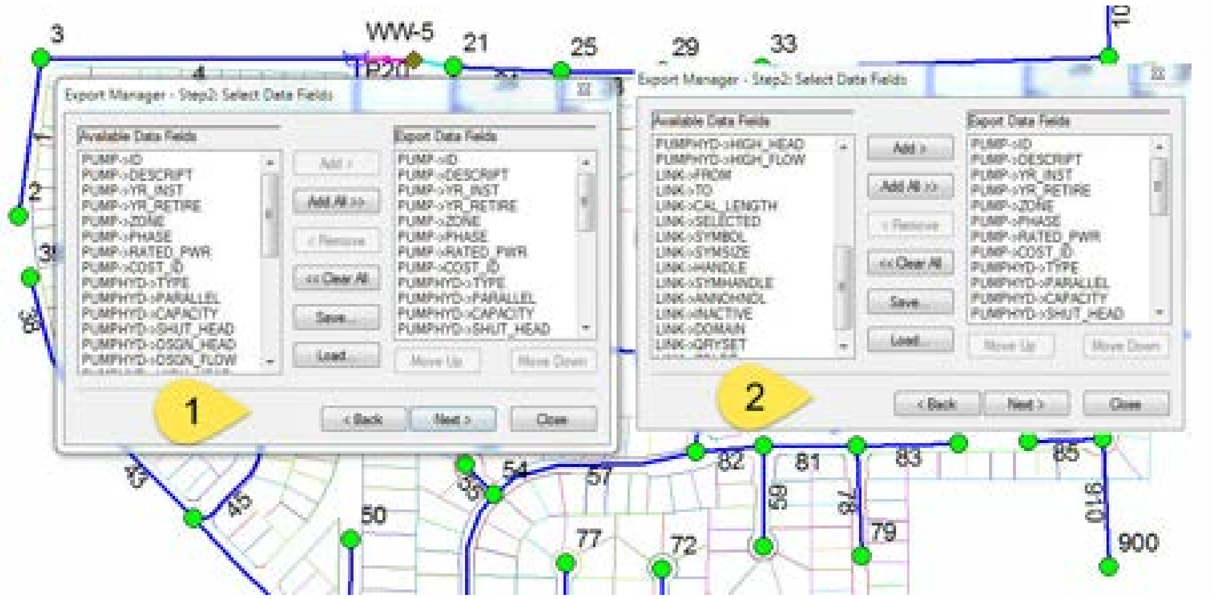


Pipe Export and Import



[Wet Well Export and Import]

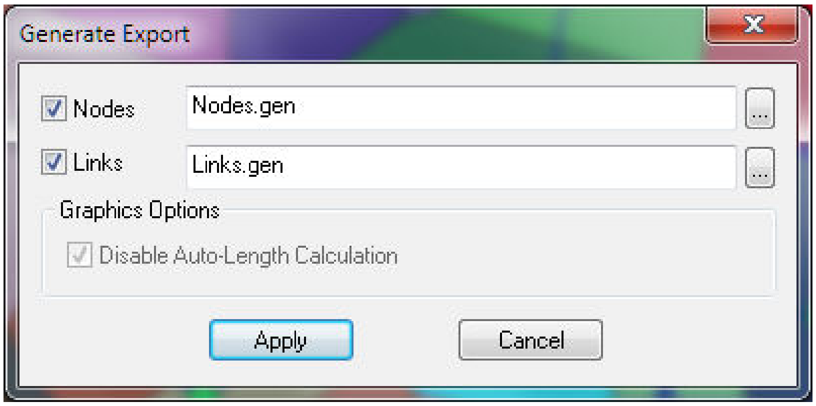
[Pump Export and Import]


Export Manager - Step2: Select Data Fields
Available Data Fields
PUMP->ID
PUMP->DESCRIPT
PUMP->YR_INST
PUMP->YR_RETIRE
PUMP->ZONE
PUMP->PHASE
PUMP->RATED_PWR
PUMP->COST_ID
PUMPHYD->TYPE
PUMPHYD->PARALLEL
PUMPHYD->CAPACITY
PUMPHYD->SHUT_HEAD
PUMPHYD->DSGN_HEAD
PUMPHYD->DSGN_FLOW
Add >
Add All >>
< Remove
<< Clear All
Save...
Load...
Export Data Fields
Move Up
Move Down
< Back
Next >
Close
1
Export Manager - Step2: Select Data Fields
PUMPHYD->HIGH_HEAD
PUMPHYD->HIGH_FLOW
LINK->FROM
LINK->TO
LINK->CAL_LENGTH
LINK->SELECTED
LINK->SYMBOL
LINK->SYMSIZE
LINK->HANDLE
LINK->SYMHANDLE
LINK->ANNOHNDL
LINK->INACTIVE
LINK->DOMAIN
LINK->QRYSET
2
WW-5
3
21
25
29
33
54
57
82
81
83
85
50
77
72
79
900

Generate File Export

Use this to Export to Generate file. Click here to learn more about the Export methodology.

Click on any portion for more details.



Apply

Click on Apply to accept the various changes and selections made in this dialog box.

Cancel

Use this to cancel the selection and close out of the dialog box. Any changes and/or selections made will not be reflected in the project.

Graphics Options

Disable Auto Length Calculation **(not USED)**

- Yes (Checked) - InfoSewer will recalculate the lengths of pipe segments based on junction node and pipe vertices geometry.
- No (Unchecked) - Ensures that the Auto Pipe Calculation will not be used and that pipe lengths, when imported, will be populated as they are in the import file.

Nodes

Check this box if you want to import/export Nodes into/from your project. Specify the location of this file by clicking on the **Browse** button

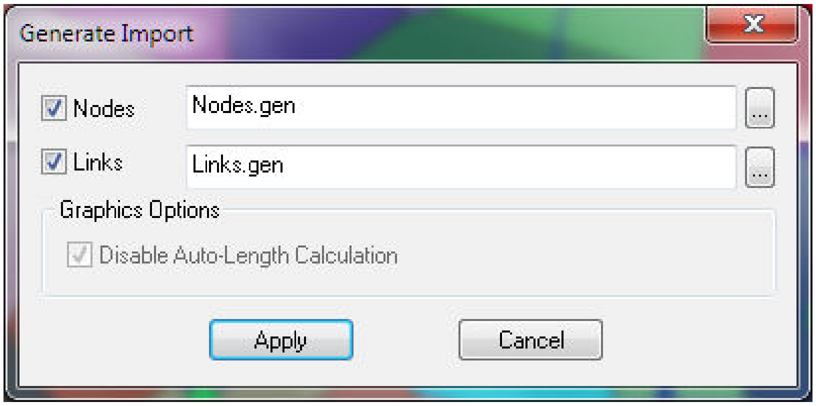
Links

Check this box if you want to import/export Links into/from your project. Specify the location of this file by clicking on the **Browse** button 

Generate File Import

Use this to Import into InfoSewerfrom Generate files. Click here to learn more about the Import methodology.

Click on any portion for more details.



Apply

Click on Apply to accept the various changes and selections made in this dialog box.

Cancel

Use this to cancel the selection and close out of the dialog box. Any changes and/or selections made will not be reflected in the project.

Graphics Options

Disable Auto Length Calculation

- Yes (Checked) - InfoSewer will recalculate the lengths of pipe segments based on junction node and pipe vertices geometry.
- No (Unchecked) - Ensures that the Auto Pipe Calculation will not be used and that pipe lengths, when imported, will be populated as they are in the import file.

Nodes

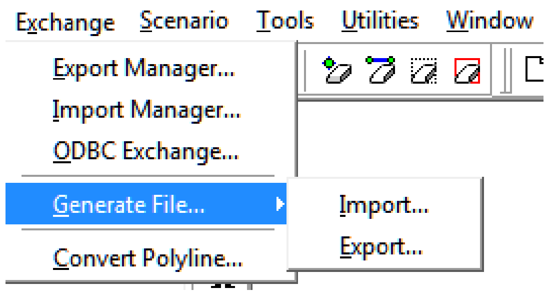
Check this box if you want to import/export Nodes into/from your project. Specify the location of this file by clicking on the **Browse** button .

Links

Check this box if you want to import/export Links into/from your project. Specify the location of this file by clicking on the **Browse** button .

Generate File Methodology

Use this to Export to Generate file or import from Generate file into InfoSewer. Choose the appropriate direction from below:



Export Methodology

For Exporting to Generate File do the following:

- From the **InfoSewerControl Center ->** **InfoSewer** button **-> Exchange** pull down menu choose **Generate File -> Export** option. This launches the Export dialog box as shown below :
- Choose the appropriate generate files you want to create and specify the location on your network or your hard drive you want to save your generate files by clicking on the **Browse** button 
- Click on Apply to create the generate files

Import Methodology

For Importing Generate Files do the following:

- From the **InfoSewerControl Center -> InfoSewer** button -> **Exchange** pull down menu choose **Generate File -> Import** option. This launches the Import dialog box as shown below :
- Choose the appropriate generate files you want to import and specify the location on your network or your hard drive you want to import your generate files from by clicking on the **Browse** button
- Choose to Enable or Disable the InfoSewer Auto-length calculation. Enabling the InfoSewer auto length calculation will over write the lengths with InfoSewercalculated pipe lengths based on the AutoCAD coordinates.
- Click on Apply to create the generate files.

Note: An ArcInfo Generate file typically contains only element coordinates. It is essential to also import a CSV file that contains all the database information.

The following illustrates a sample node and link ESRI generate file. Refer to the ESRI Arc/Info documentation for more information on the Generate file format:

Node Generate File

In the following illustration, several nodes are written to the Generate file. For each node, the **ID**, **X-coordinate**, **Y-coordinate**, and a placeholder for a **Z** value are provided:

1 1678.750000 1703.500000 0.000000

3 1993.750000 1703.500000 0.000000

5 2193.250000 1703.500000 0.000000

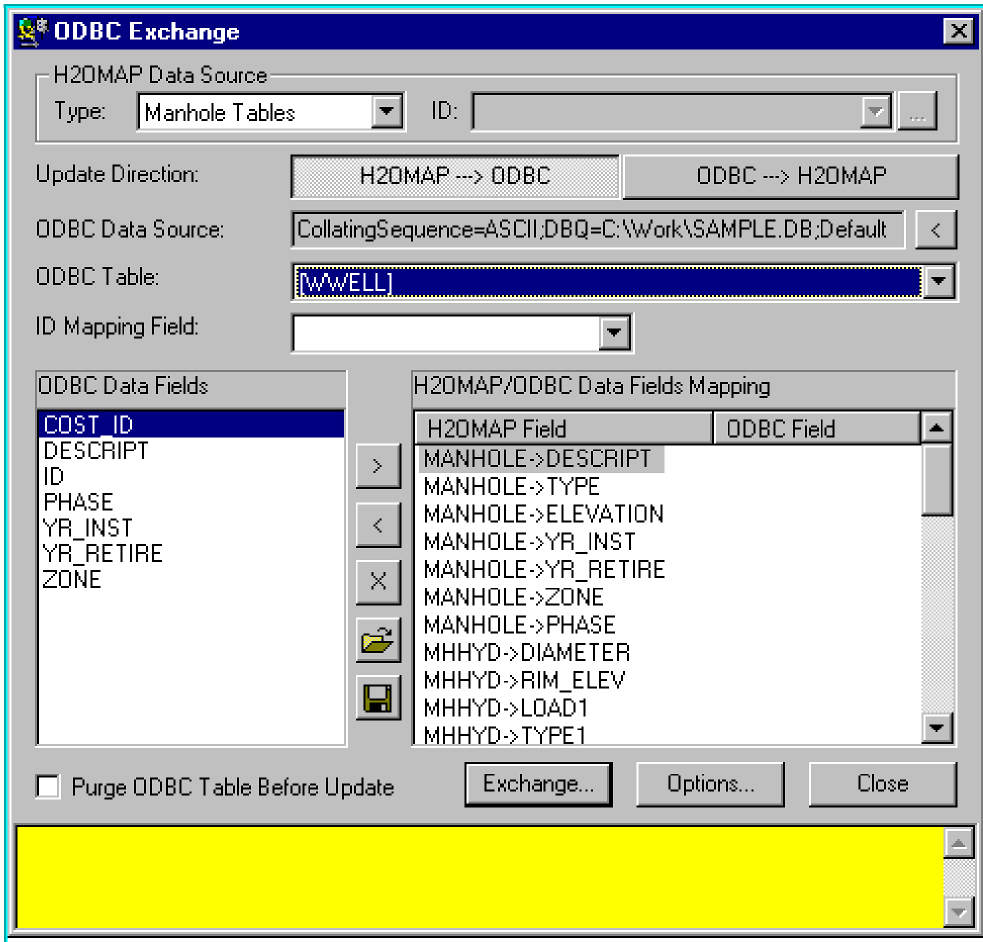
7 3030.000000 1715.000000 0.000000

END

ODBC

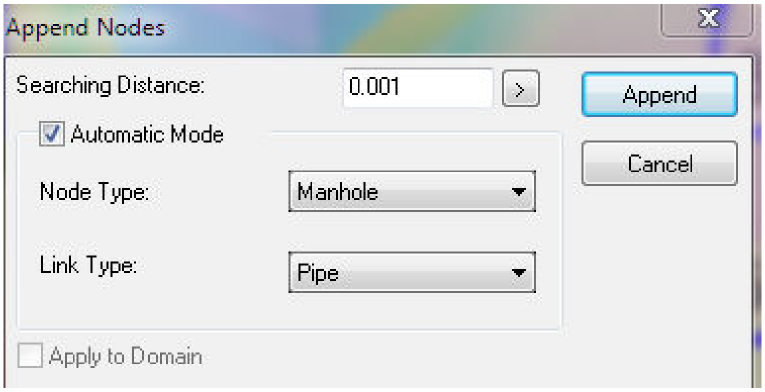
The Open Database Connectivity (ODBC) module of **InfoSewer** allows the user to actively link to any external database and import that data into the current project session. Likewise, the user can also update an external database with InfoSewer data.

To initiate the ODBC Exchange, from the **Exchange** menu, select **ODBC** and the following dialog box will appear. Click on any portion to learn more.



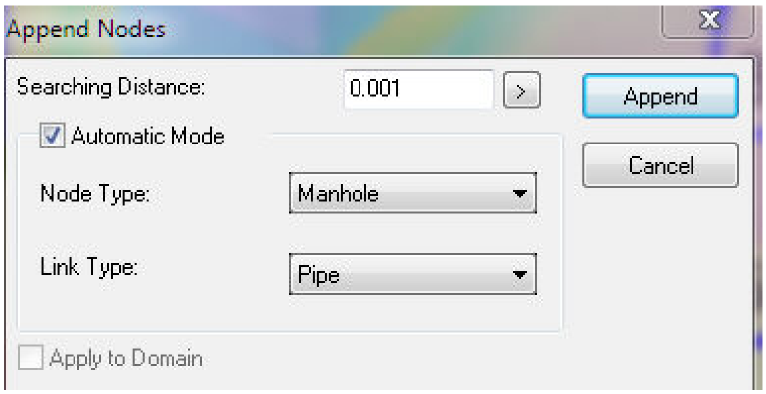
Append Nodes

The Append Nodes command is used to automate the InfoSewer pipe and node network creation. If the user has a pipe network in another third party application like AutoCAD or Microstation, the network can be bought into InfoSewer as a background layer (.dxf or .dwg) and then converted to an InfoSewer pipe and node network. To learn more about the Append Node Methodology click here.



Append Nodes Methodology

The Append Nodes command is used to automate the InfoSewer pipe and node network creation. If the user has a pipe network in another third party application like AutoCAD or Microstation, the network can be bought into InfoSewer as a background layer (.dxf or .dwg) and then converted to an InfoSewer pipe and node network.



Methodology

To Convert a Polyline drawing into an InfoSewer project use the following procedure:

- Import the external CAD drawing or shapefile into InfoSewer. Use the GIS Gateway command under the **InfoSewer Control Center -> InfoSewer** button **-> Exchange** pull down menu to launch the **GIS Gateway** dialog box.

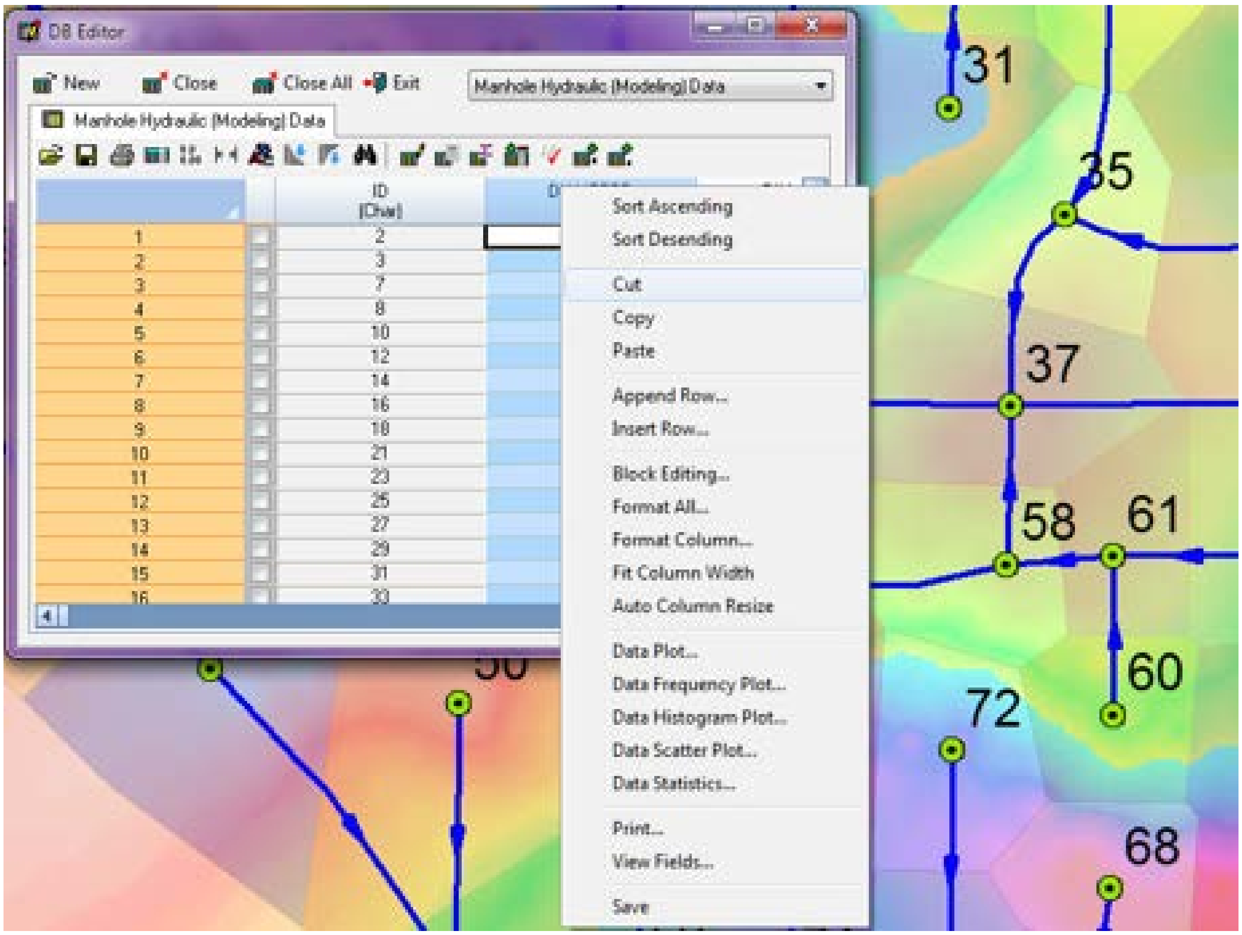
- Create a new GIS Exchange Cluster by clicking on the **Add** button. Click on the Browse button  next to the **GIS Data Source** command and select the AutoCAD drawing that you want to import. For most AutoCAD drawings the **Polyline** choice is the best choice. Select **Pipe Tables** as the **InfoSewer Data Type**. Specify **Tabular Join** as your **Relate Type** and **0: Bi-Direction** as your **Update Direction**.

- Under the **Tabular Join** tab of the **GIS Exchange Cluster** choose to **Create New Records**, **Create Unique IDs** and to **Update Geometry Data** and click on the **OK** button at the bottom of the dialog box. Click here to learn more about GIS Exchange Clusters.

- Now click on the **Load** button on your **GIS Gateway** dialog box to launch the **Load Data from GIS Layer or Table** dialog box. Select the Exchange cluster that you want to load and click on the **Load** button.
- Once the drawing has been inserted into your InfoSewer project choose the **Append Nodes** command from your **InfoSewer Control Center -> InfoSewer** button **->** **Exchange** pull down menu to launch the Append Nodes dialog box.
- Specify the Node Searching radius. It is recommended that half the size of the node symbol size would be a good beginning point. You may find out the size of the node symbol from your **InfoSewer Control Center -> InfoSewer** button **->** **Project Preferences -> Default Symbol Sizes** tab. Also specify your default **Node Type** here. Normally it is good practice to select **Junction** as the default node type.
- Choose the **Automatic Mode** option if you want InfoSewer to automatically convert your polylines for you.
- Once you have entered and selected all your preference choices, click on **Convert** to launch the conversion process.

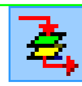
Windows Clipboard

Because InfoSewer is Windows based, any simulation report, graph, or database table that resides in an InfoSewer project is capable of being copied at any time. Just open the desired output report or database table and highlight the data you wish to export. Then, right-mouse click over the highlighted area and select Copy. Open any third party package like Microsoft Excel and select Paste from the Edit menu.



Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

GIS Exchange 

The GIS Exchange allows the user to quickly import and export data between InfoSewer and other GIS formats. It also allows the user to perform spatial joins between an imported GIS layer and an InfoSewerdata table.

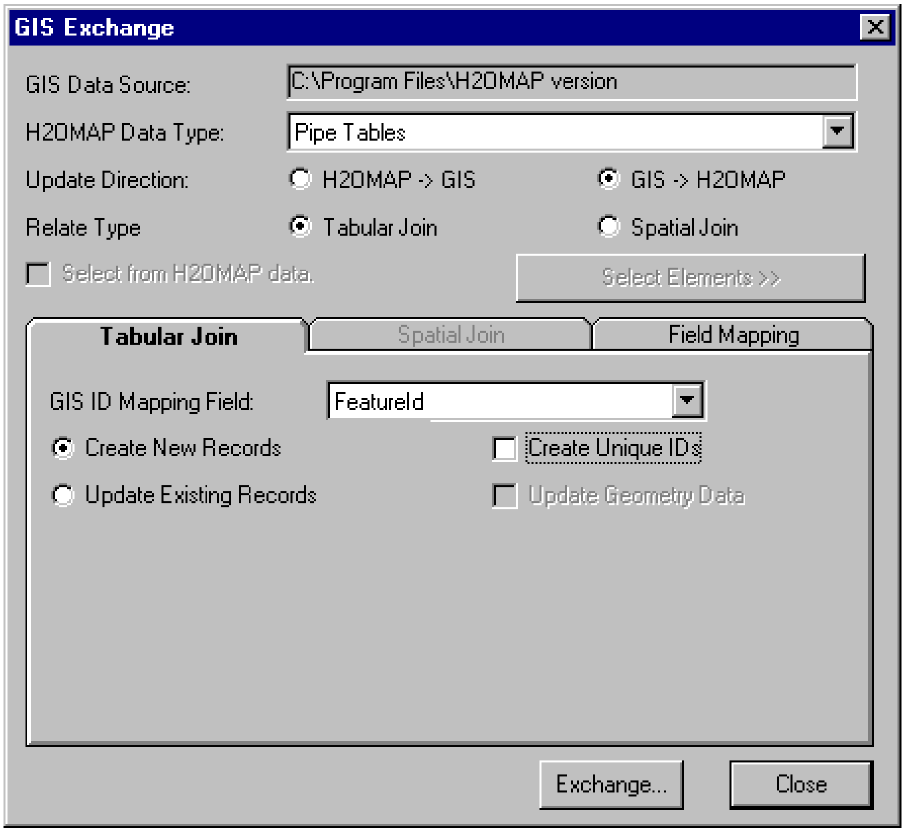
Running GIS Exchange

To run a GIS Exchange, you must first import a GIS layer for which an exchange is desired. From the **View** menu, select **Add Layer**. Once the add layer dialog box opens, point to the GIS layer on the network drive to add it to the InfoSewer session. Add the layer using your mouse and click on the GIS Exchange icon at the top of the control center. You will now see the GIS Exchange dialog box and be able to select the type of join to be performed by the GIS Exchange.

This process is also applicable to ESRI geodatabases and ArcSDE Layers. By adding an SDE layer, you are also to perform a GIS Exchange with the highlighted GIS layer.

Tabular Joins

A tabular join merely exchanges data fields between one GIS data source and another. Use the GIS ID Mapping field to denote which field in the GIS layer will be matched with the respective ID field in the selected InfoSewer data table.



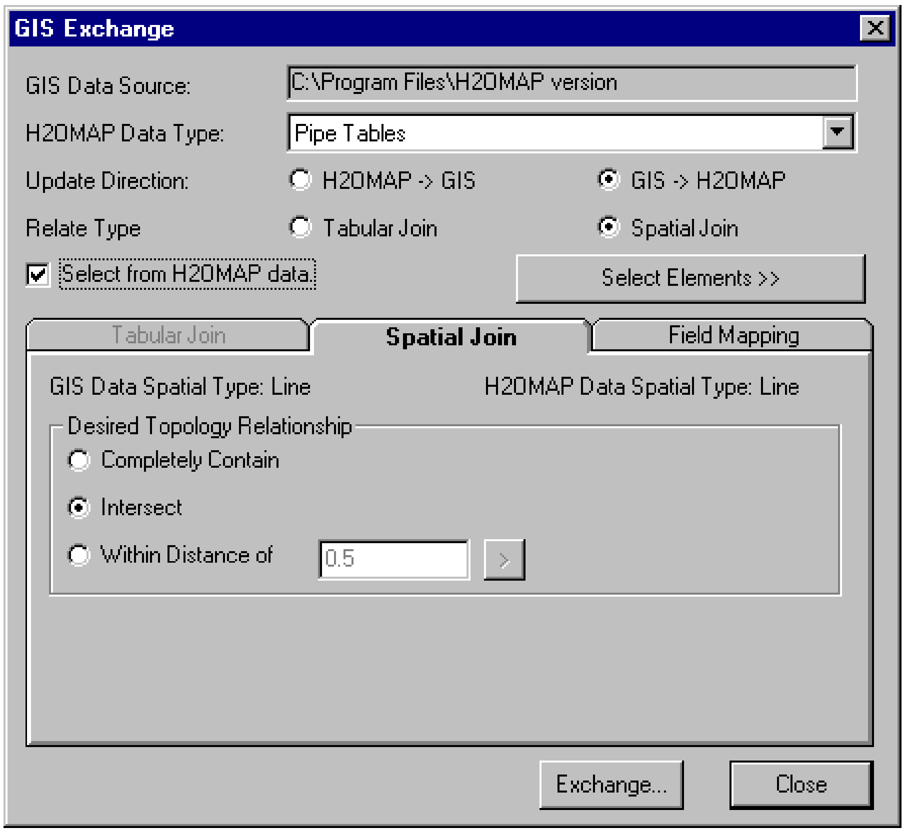
- **GIS Data Source** - The file that will be used for exchanging data.
- **H2OMAP Data Type** - The InfoSewer Element (and associated tables) that will be used in the exchange.
- **Update Direction** - Specify the direction for the GIS Exchange.
- **Relate Type** - During a tabular join, select the Tabular Join radial button.
- **GIS ID Mapping Field** - The Unique ID field that will be mapped to the corresponding ID field in InfoSewer
- **Create New Records** - When this option is selected, new records will be created. Use the Field Mapping tab to map the corresponding fields.
- **Create New ID's** - By checking this box, InfoSewer will automatically assign unique ID's to the elements being exchanged.
- **Update Existing Records** - This option will update matching fields as denoted in the GIS ID Mapping Field. Use the Field Mapping tab to map the corresponding fields.

- **Update Geometry Data** - When this box is checked, the physical geometry of the data file being exchanged will be updated to reflect recent changes.

Once the tabular join options have been specified, proceed to the Field Mapping tab to map the desired fields prior to exchanging the data.

Spatial Joins

A spatial join allows the user to perform a join between two data layers based on their physical relationship to one another. This means that the data contained in one data layer can be assigned to another table by using 'smart topography' features of InfoSewer.

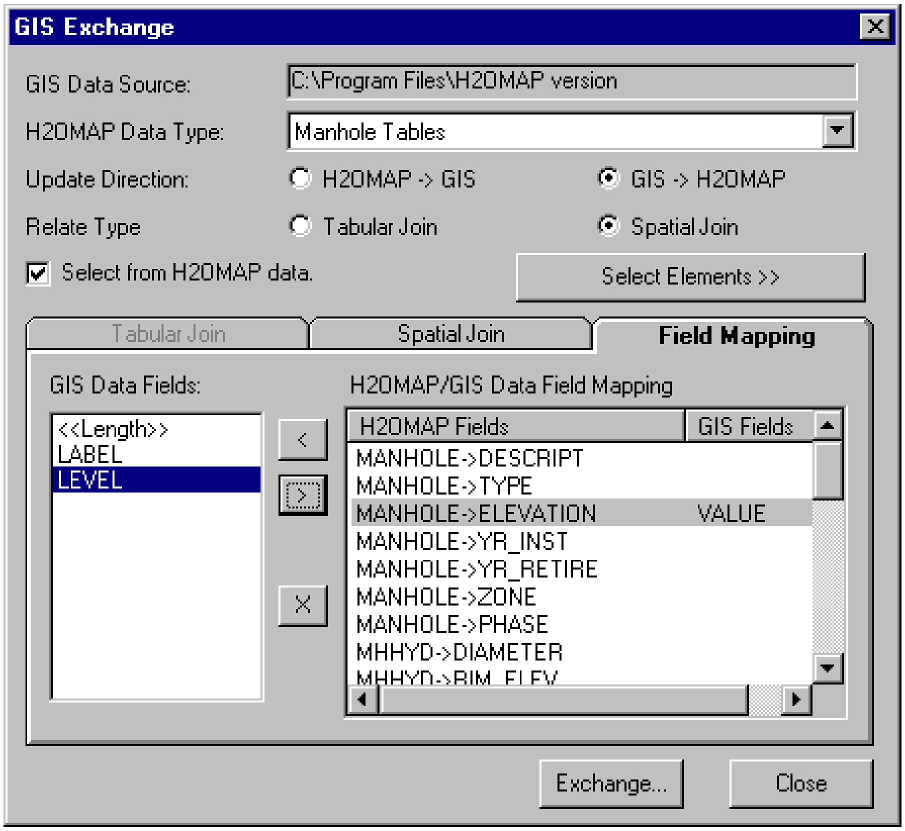


- **GIS Data Source** - The file that will be used for exchanging data.
- **InfoSewer Data Type** - The InfoSewer Element (and associated tables) that will be used in the exchange.
- **Update Direction** - Specify the direction for the GIS Exchange.
- **Relate Type** - During a spatial join, select the Spatial Join radial button.
- **Select from InfoSewer Data** - Using a graphical selection, the user can graphically select which elements are to be included in the spatial join process. When this box is checked, click the Select Elements button to make a graphical selection.

- **Desired Topology Relationship** - The user must specify the type of spatial join to be utilized during the spatial join
- ***Completely Contain*** - Used to exchange data between a point or line data layer and a polygon. Those elements that are 'completely contained' in the polygon data set will be tagged for the spatial join. Those elements not completely contained will be excluded.
- ***Intersect*** - When two data layers are intersected, those elements that are contained within and/or cross the intersected layer are tagged for the spatial join. Those elements that do not have a physical intersection are excluded.
- ***Within Distance of*** - Those elements that are within the user specified distance will be tagged for a spatial join. This option is specified to assign contour elevation data to a point element such as a manhole rim elevation. In other words, any elements of the selected layer that fall within the tolerance level specified will be tagged for a spatial join.
- Once the spatial join options have been specified, proceed to the Field Mapping tab to map the desired fields prior to exchanging the data.

Field Mapping

The field mapping tab allows the user to map corresponding fields to one another prior to performing the GIS Exchange. Merely select fields on the left hand side to be mapped to InfoSewer field on the right. Once all field mapping has been completed, click on the Exchange button to finish the GIS Exchange.



- **GIS Data Fields** - The fields found in the GIS data set that can be linked to the InfoSewer data fields. Use the arrow buttons to map one field to another.
- **InfoSewer/GIS Data Field Mapping** - TheInfoSewer fields that are available to be mapped during a GIS Exchange. By using the arrow buttons, select the desired exchange fields to map to the InfoSewer fields.

Geodatabase Theory

Geodatabases contain geographic information and typically comprise of feature classes and/or tables.

A Feature class stores various geographic features represented as points, lines, or polygons, and their attributes. Tables on the other hand contain additional attributes for a feature class or geographic information such as addresses or x,y,z coordinates.

ArcView® includes a data model for representing geographic information called the geodatabase data model. The geodatabase can be used to define and work with different user - or application - specific models. By defining and implementing behavior on a generic geographic data model, the geodatabase provides a robust platform for virtually any GIS application.

Further ArcView® allows you to access two broad categories of geodatabases: personal geodatabases and multiuser geodatabases managed using ArcSDE™.

Personal Geodatabases

Personal geodatabases support many readers and a single editor. They are stored in a Microsoft Access database. You can create and work with personal geodatabases with ArcGIS™ without the need for any other software.

ESRI Help File -

Creating a personal geodatabase involves creating an .mdb file on disk. This can be done from the Catalog tree in ArcGIS for Desktop or, or using the Create Personal GDB geoprocessing tool.

If you use the geoprocessCreate Personal GDBing tool, you can create a geodatabase that corresponds to an older release of ArcGIS. This ability allows you to share data with people who have older releases of ArcGIS, since older releases of ArcGIS may not be able to open newer releases of the geodatabase. For more information on this, see Client and geodatabase compatibility.

To create a personal geodatabase that corresponds to the same release as the ArcGIS for Desktop client you are using, follow these steps:

Steps:

1. Right-click the file folder in the Catalog tree where you want to create the new personal geodatabase.
2. Point to New.
3. Click Personal Geodatabase.

   A personal geodatabase is created in the location you selected.
4. Type a new name for this personal geodatabase and press Enter

SDE Geodatabases

Multiuser geodatabases can be read and edited by multiple users; they require a DBMS, such as Oracle, SQL Server, Informix, or IBM DB2. Multiuser geodatabases can be used with any ArcGIS product (ArcView®, ArcInfo™, or ArcEditor™) but require ArcSDE for editing and schema management.

InfoSewer allows you work with these Geodatabases. Refer to the section on Geodatabase Import/Export Methodology to learn more. To learn more about the ArcView® Geodatabases refer to the ArcGIS help file. Click here to learn more about **Working with ESRI Geodatabases**.

Other Related Topics - Working with ESRI Geodatabases, GIS Cluster - Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster - Tabular Join, GIS Field Mapping, GIS Gateway

---

Geodatabases

Geodatabases contain geographic information and typically comprise of feature classes and/or tables.

A Feature class stores various geographic features represented as points, lines, or polygons, and their attributes. Tables on the other hand contain additional attributes for a feature class or geographic information such as addresses or x,y,z coordinates.

ArcView® includes a data model for representing geographic information called the geodatabase data model. The geodatabase can be used to define and work with different user - or application - specific models. By defining and implementing behavior on a generic geographic data model, the geodatabase provides a robust platform for virtually any GIS application.

Further ArcView® allows you to access two broad categories of geodatabases: personal geodatabases and multiuser geodatabases managed using ArcSDE™.

Personal Geodatabases

Personal geodatabases support many readers and a single editor. They are stored in a Microsoft Access database. You can create and work with personal geodatabases with ArcGIS™ without the need for any other software.

ESRI Help File -

Creating a personal geodatabase involves creating an .mdb file on disk. This can be done from the Catalog tree in ArcGIS for Desktop or, or using the Create Personal GDB geoprocessing tool.

If you use the geoprocessCreate Personal GDBing tool, you can create a geodatabase that corresponds to an older release of ArcGIS. This ability allows you to share data with people who have older releases of ArcGIS, since older releases of ArcGIS may not be able to open newer releases of the geodatabase. For more information on this, see Client and geodatabase compatibility.

To create a personal geodatabase that corresponds to the same release as the ArcGIS for Desktop client you are using, follow these steps:

Steps:

1. Right-click the file folder in the Catalog tree where you want to create the new personal geodatabase.
2. Point to New.
3. Click Personal Geodatabase.

   A personal geodatabase is created in the location you selected.
4. Type a new name for this personal geodatabase and press Enter

SDE Geodatabases

Multiuser geodatabases can be read and edited by multiple users; they require a DBMS, such as Oracle, SQL Server, Informix, or IBM DB2. Multiuser geodatabases can be used with any ArcGIS product (ArcView®, ArcInfo™, or ArcEditor™) but require ArcSDE for editing and schema management.

InfoSewer allows you work with these Geodatabases. Refer to the section on Geodatabase Import/Export Methodology to learn more. To learn more about the ArcView® Geodatabases refer to the ArcGIS help file. Click here to learn more about **Working with ESRI Geodatabases**.

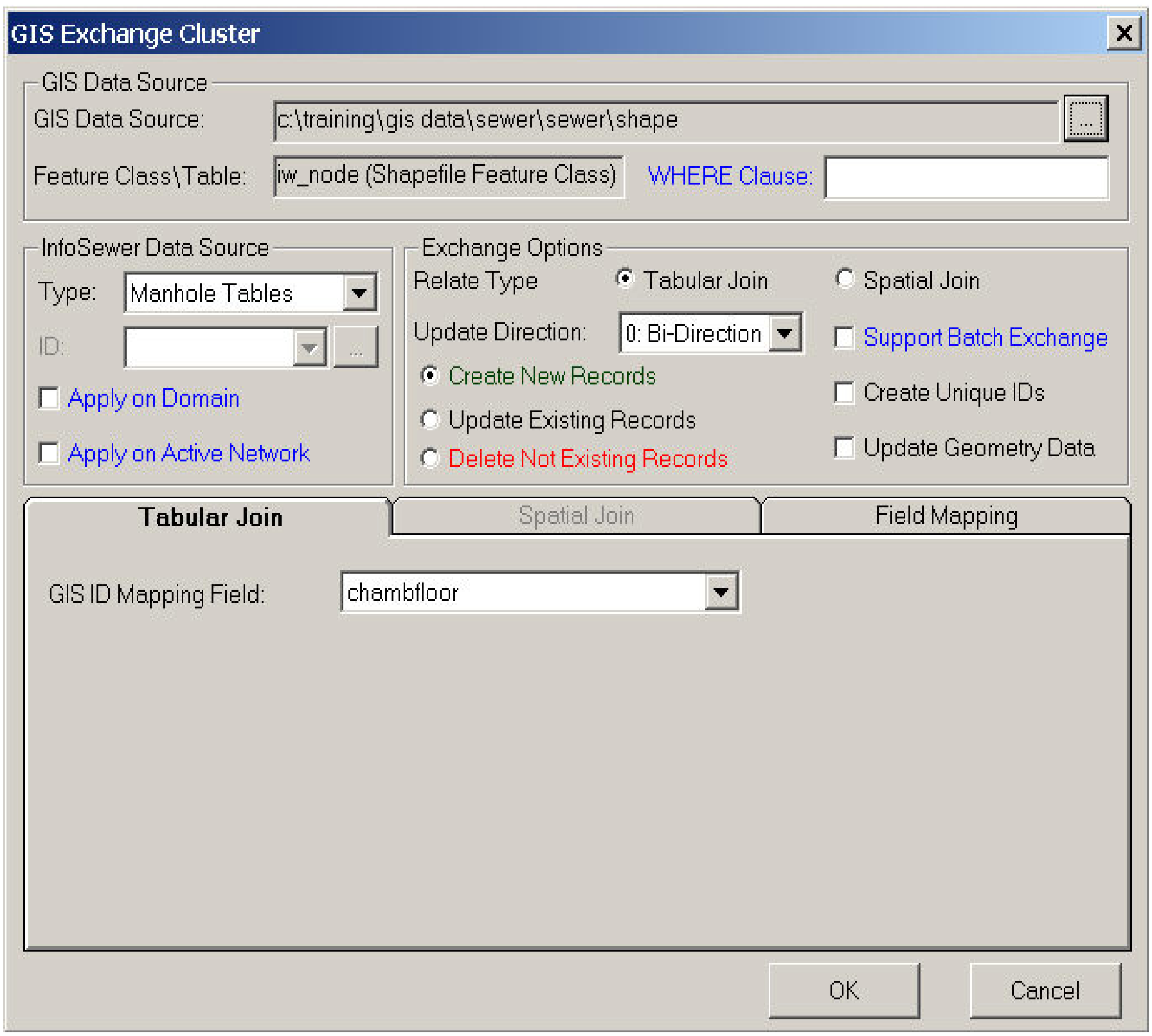
Other Related Topics - Working with ESRI Geodatabases, GIS Cluster - Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster - Tabular Join, GIS Field Mapping, GIS Gateway

---

GIS Exchange Cluster

A GIS Exchange Cluster defines a linkage toward the desired GIS data source and a set of data exchange parameters that allows the user to quickly load and save data when the cluster is applied. This helps to perform data exchange with various GIS data sources supported by ESRI including personal and enterprise Geodatabases. Click here to learn more about Working with ESRI Geodatabases.

Click on any section below to learn more or see the description at the bottom of the page.



Apply on Domain

Apply exchange on elements contained in Domain only.

Apply on Active Network

Apply exchange on elements contained in Active Network only.

Create Unique IDs

By checking this box, InfoSewer will automatically assign unique ID's to the elements being exchanged.

Create New Records

When this option is selected, new records will be created. Use the Field Mapping tab to map the corresponding fields.

Update Existing Records

This option will update matching fields as denoted in the GIS ID Mapping Field. Use the Field Mapping tab to map the corresponding fields.

Delete Not Existing Records

This will delete the elements without any existing records.

Dataset-Feature Class

GIS layer/table that will be used for exchanging data.

GIS Data Source

Specify the location of the external GIS data source. This source file will be used by InfoSewer to exchange data.

GIS ID Mapping Fields

Use the GIS ID Mapping field to denote which field in the GIS layer will be matched with the respective ID field in the selected InfoSewer data table.

Relate Type

Specify the relate type here. When conducting a tabular join, select the Tabular Join radial button. For a spatial join use the Spatial Join radial button.

Support Batch Exchange

Includes current exchange in Batch Exchange.

Update Direction

Select which direction the ODBC linkage is occurring. If you are exporting data from InfoSewer to the external database, select Internal (InfoSewer) --> ODBC. Likewise, if data is being imported into InfoSewer, select ODBC --> Internal (InfoSewer).

Update Geometry Data

When this box is checked, the physical geometry of the data file being exchanged will be updated to reflect recent changes.

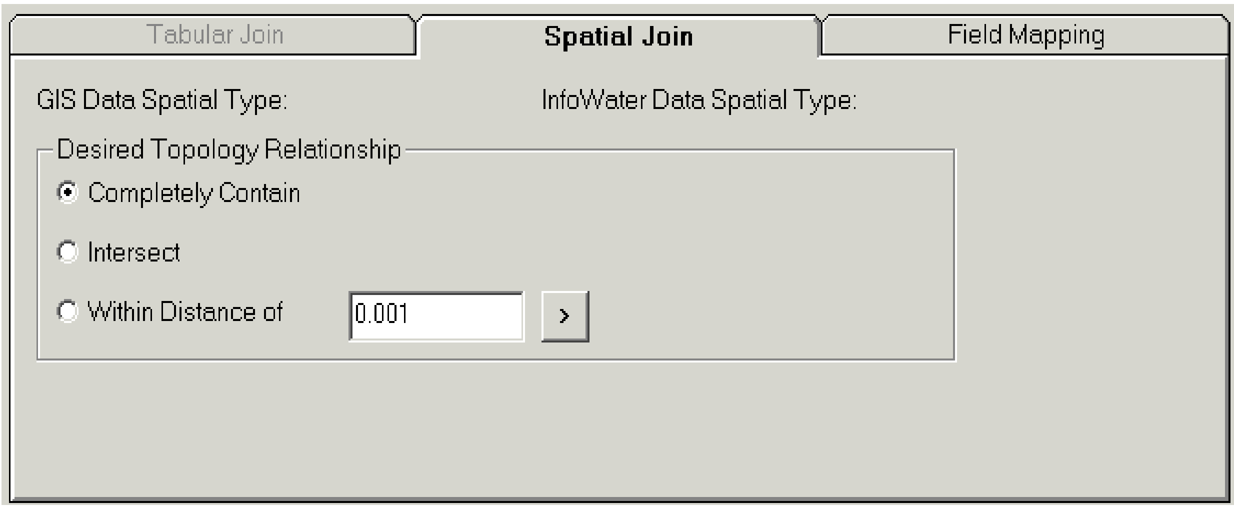
Other Related Topics - Working with ESRI Geodatabases, Geodatabase Theory, GIS Cluster -Spatial Join, GIS Exchange Cluster Tabular Join, GIS Field Mapping, GIS Gateway

---

GIS Exchange Cluster - Spatial Join

A spatial join allows you to perform a join between two data layers based on their physical relationship to one another. This means that the data contained in one data layer can be assigned to another table by using 'smart topography' features of InfoSewer.

Click on any section below to learn more or see the description at the bottom of the page.

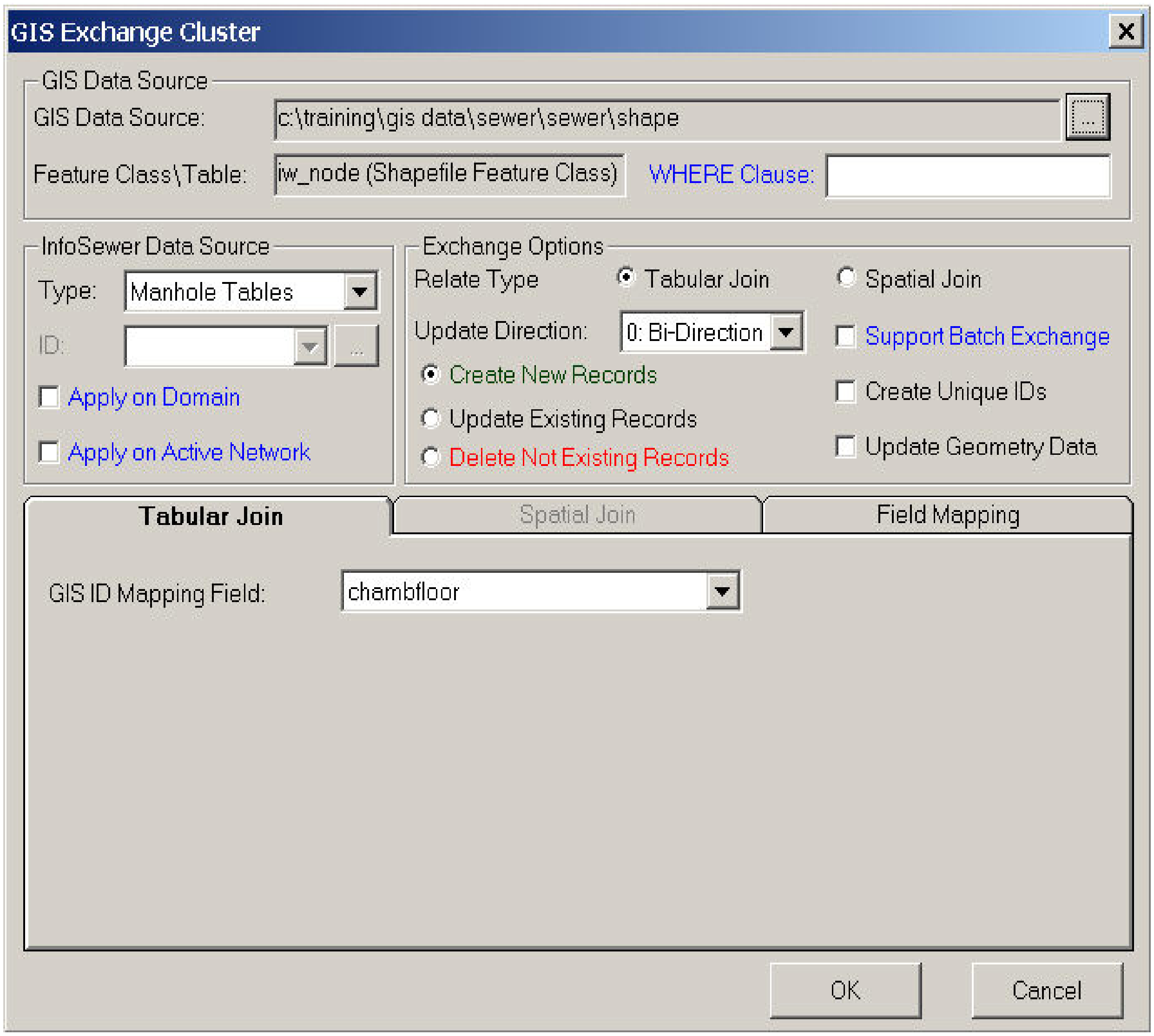


Once the spatial join options have been specified, proceed to the Field Mapping tab to map the desired fields prior to exchanging the data. Click here to learn more about **Working with ESRI Geodatabases**.

GIS Exchange Cluster - Tabular Join

A tabular join merely exchanges data fields between one GIS data source and another. While using the InfoSewer GIS Gateway dialog box you may update your InfoSewer data fields from an external geodatabase or alternatively update the external geodatabase with InfoSewer data. Click here to learn more about Working with ESRI Geodatabases.

Click on any section below to learn more or see the description at the bottom of the page.



Once the tabular join options have been specified, proceed to the Field Mapping tab to map the desired fields prior to exchanging the data.

Apply on Domain

Apply exchange on elements contained in Domain only.

Apply on Active Network

Apply exchange on elements contained in Active Network only.

Create Unique IDs

By checking this box, InfoSewer will automatically assign unique ID's to the elements being exchanged.

Create New Records

When this option is selected, new records will be created. Use the Field Mapping tab to map the corresponding fields.

Update Existing Records

This option will update matching fields as denoted in the GIS ID Mapping Field. Use the Field Mapping tab to map the corresponding fields.

Delete Not Existing Records

This will delete the elements without any existing records.

Dataset-Feature Class

GIS layer/table that will be used for exchanging data.

GIS Data Source

Specify the location of the external GIS data source. This source file will be used by InfoSewer to exchange data.

GIS ID Mapping Fields

Use the GIS ID Mapping field to denote which field in the GIS layer will be matched with the respective ID field in the selected InfoSewer data table.

Relate Type

Specify the relate type here. When conducting a tabular join, select the Tabular Join radial button. For a spatial join use the Spatial Join radial button.

Support Batch Exchange

Includes current exchange in Batch Exchange.

Update Direction

Select which direction the ODBC linkage is occurring. If you are exporting data from InfoSewer to the external database, select Internal (InfoSewer) --> ODBC. Likewise, if data is being imported into InfoSewer, select ODBC --> Internal (InfoSewer).

Update Geometry Data

When this box is checked, the physical geometry of the data file being exchanged will be updated to reflect recent changes.

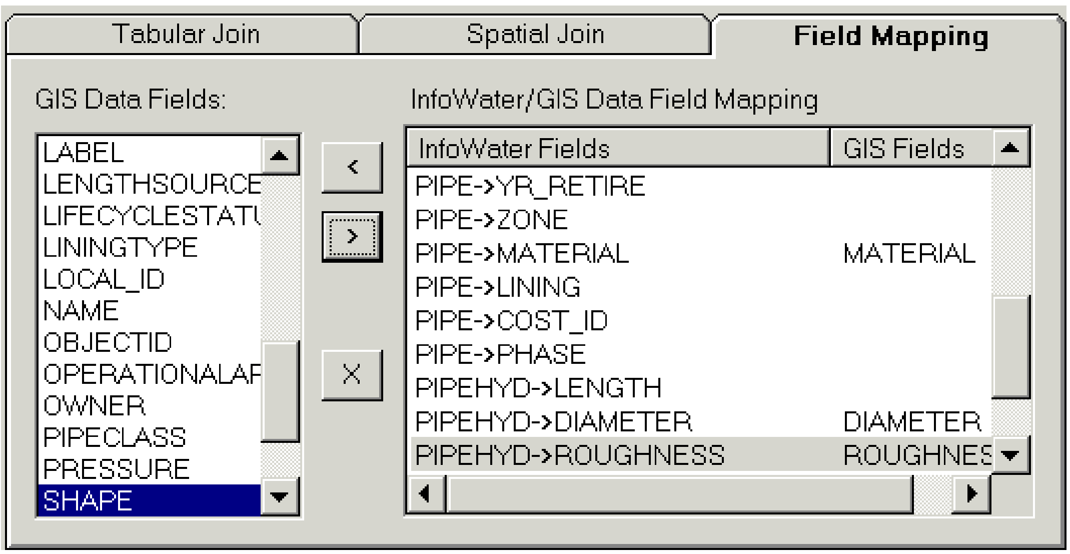
Other Related Topics - Working with ESRI Geodatabases, GIS Cluster - Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster - Tabular Join, GIS Field Mapping, GIS Gateway

---

GIS Field Mapping

The field mapping tab allows you to map corresponding fields to one another prior to performing the GIS Exchange. Merely select fields on the left hand side to be mapped to InfoSewer fields on the right hand side. Once the field mapping has been completed, click on the OK button to finish the GIS Exchange. Click here to learn more about Working with ESRI Geodatabases.

Click on any section below to learn more or see the description at the bottom of the page.



InfoSewer/ GIS Data Field Mapping

The InfoSewer fields that are available to be mapped during a GIS Exchange. By using the arrow buttons, select the desired exchange fields to map to the InfoSewer fields.

GIS ID Mapping Fields

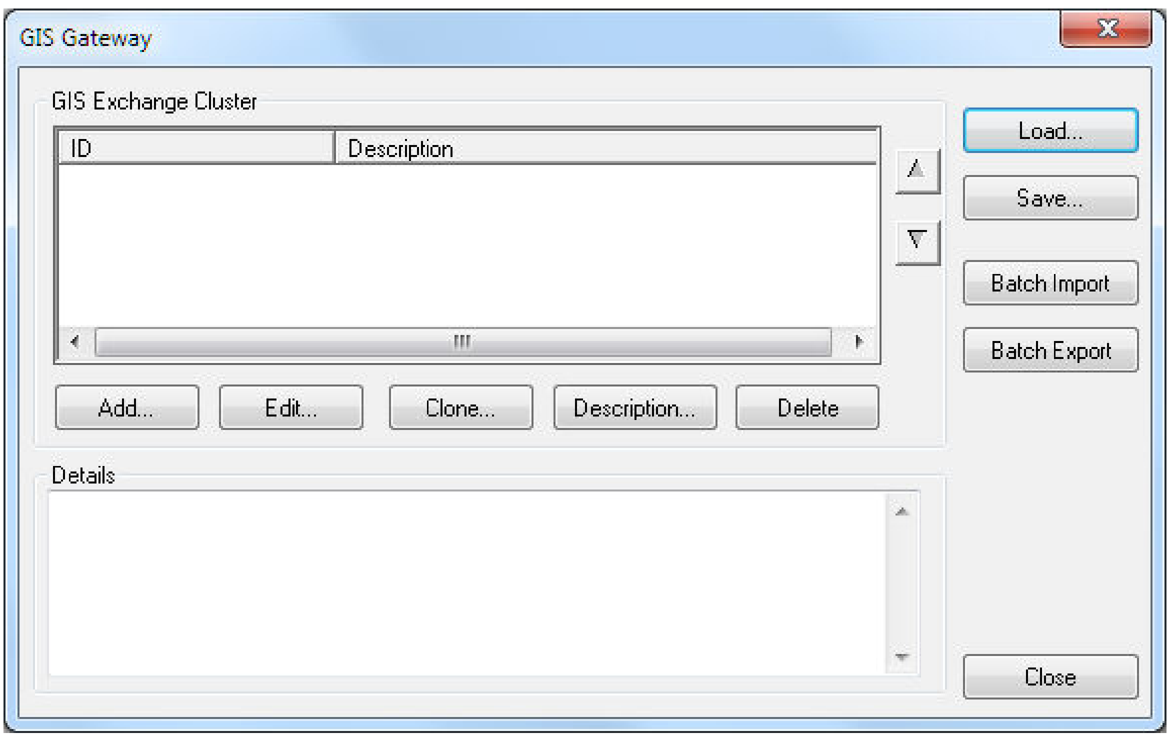
Use the GIS ID Mapping field to denote which field in the GIS layer will be matched with the respective ID field in the selected InfoSewer data table.

Other Related Topics - Working with ESRI Geodatabases, Geodatabase Theory, GIS Cluster -Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster Tabular Join, GIS Gateway

GIS Gateway

GIS Gateway provides a means to create data exchange parameters that allow you to quickly load and save data between InfoSewer and other GIS formats (including Geodatabases and personal Geodatabases).

To run GIS Gateway, from the **InfoSewer Control Center** toolbar **->** **InfoSewer** button **->** **Exchange** menu, select the **GIS Gateway** command. Alternatively, you may launch the GIS Gateway dialog box by clicking on the GIS Gateway icon  on your **InfoSewer Control Center**. You will now see the GIS Gateway dialog box and be ready to manage your GIS Exchange Clusters and perform desired data exchange with various external GIS data sources. A GIS Exchange Cluster defines a set of data exchange parameters which allows you to quickly load and save data between InfoSewer and other GIS formats. Click here to learn more about **Working with ESRI Geodatabases**.



Add

Use this to create a new GIS Exchange Cluster. A GIS Exchange Cluster defines a set of data exchange parameters which allows the user to quickly load and save data between InfoSewer and other GIS formats. It also allows the user to perform spatial joins between an external GIS layer and an InfoSewer data table.

Batch Import

Supports import of Multiple Exchanges.

Batch Export

Supports export of Multiple Exchanges.

Clone

This button will make a copy of a previously created GIS exchange.

Close

Use this to close out of the InfoSewer GIS Gateway dialog box. Make sure to save your settings before you close this dialog box.

Details

This section displays the details of the current selected GIS Exchange Cluster.

Description

Allows the user to modify the description of current selected GIS Exchange Cluster.

Delete

Delete the current selected GIS Exchange Cluster.

Edit

Use this to edit a GIS Exchange Cluster. Use the Add button to create a New GIS Cluster.

A GIS Exchange Cluster defines a set of data exchange parameters which allows the user to quickly load and save data between InfoSewer and other GIS formats. It also allows the user to perform spatial joins between an external GIS layer and an InfoSewer data table.

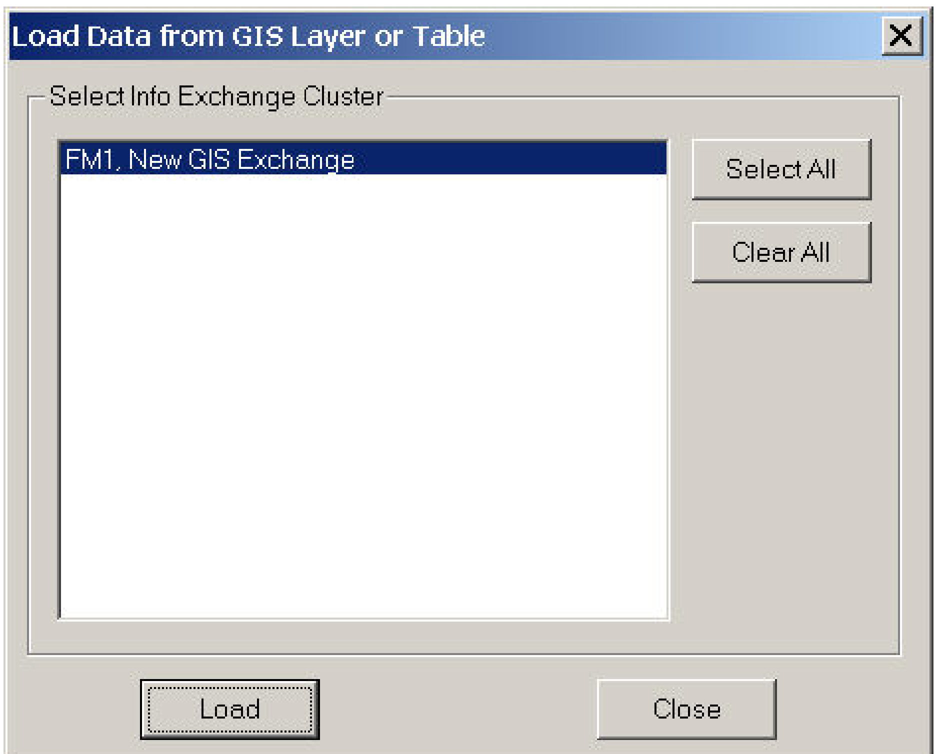
ID & Description

This section displays the GIS cluster ID and Description. The GIS cluster ID is an alpha-numeric entity and may not exceed 20 characters in length. The Description field allows a maximum of 60 characters.

Load

When the user clicks on the "Load" button, "Load Data from GIS Layer of Table" dialog box will appear and the user can select the desired GIS Exchange Clusters to pull data from various external GIS data sources.

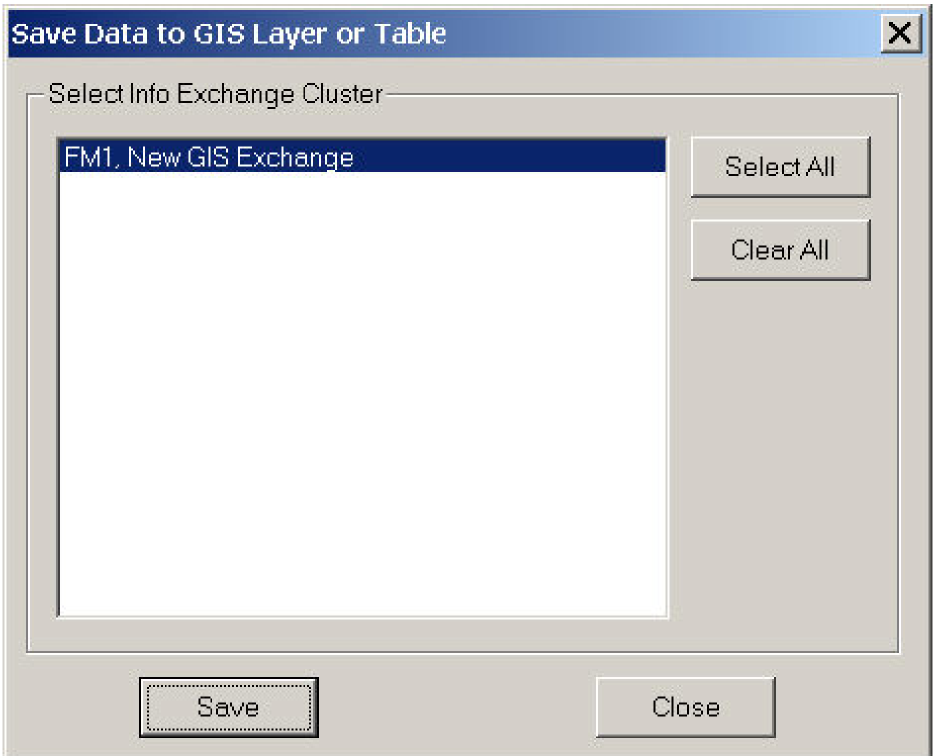
Left click on the Exchange Cluster that you want to load and click on the **Load** button at the bottom of the dialog box. Alternatively click on **Select All** or **Clear All** to select or clear all the selections. Click on **Close** to close out of the **Load Data from GIS Layer or Table** dialog box.



Save

Click on the **Save** button to launch the **Save Data to GIS Layer of Table** dialog box. You can now select the desired GIS Exchange Clusters to push data to various external GIS data sources.

Left click on the Exchange Cluster that you want to save and click on the **Save** button at the bottom of the dialog box. Alternatively click on **Select All** button or **Clear All** button to select or clear all the selections. Click on **Close** button to close out of the **Save Data to GIS Layer or Table** dialog box.



Other Related Topics - Working with ESRI Geodatabases, GIS Cluster - Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster - Tabular Join, GIS Field Mapping, GIS Gateway

---

Working with ESRI Geodatabases

InfoSewer allows you to work with both Personal Geodatabases as well as SDE Geodatabases served by enterprise ArcSDE servers. To import/export data from/to Geodatabases use the GIS Gateway tool (InfoSewer Control Center -> InfoSewer button -> Exchange pull down menu -> GIS Gateway command). Click here to learn more about Geodatabases.

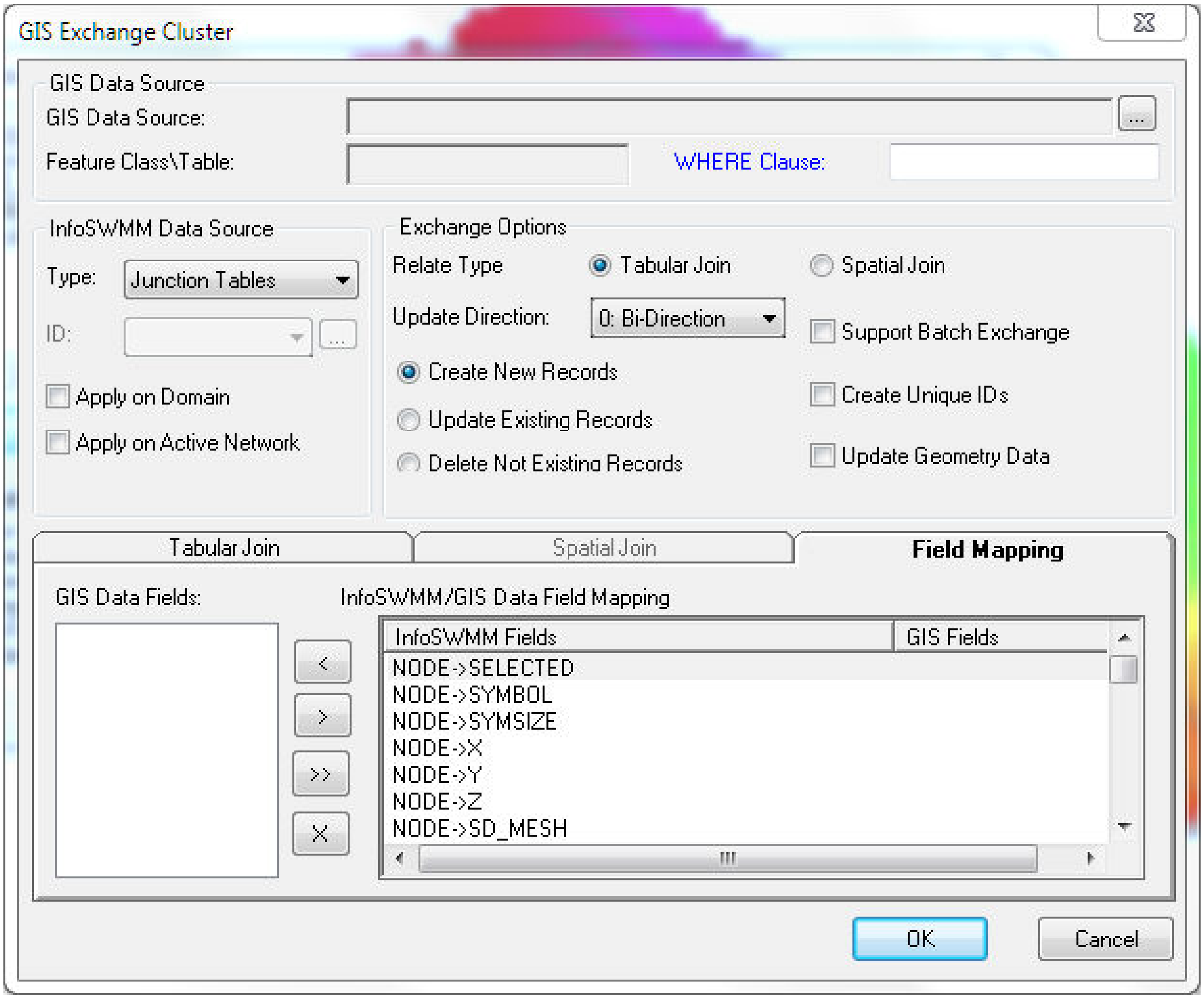
GIS Gateway utilizes GIS Exchange Clusters to perform data exchange with various GIS data sources supported by ESRI including personal and enterprise Geodatabases. A GIS Exchange Cluster defines a linkage toward the desired GIS data source and a set of data exchange parameters that allows the user to quickly load and save data when the cluster is applied.

Define a GIS Exchange Cluster

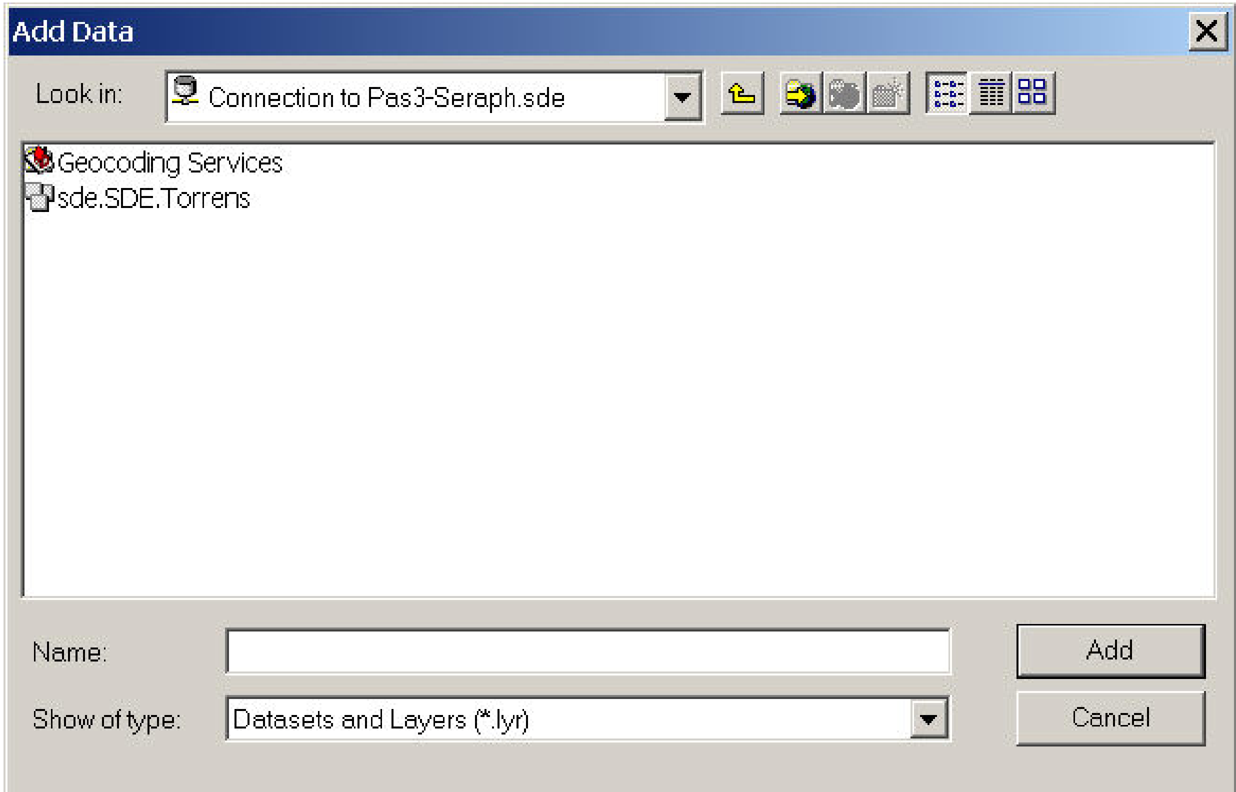
For defining a new GIS Exchange Cluster for the desired Geodatabase do the following:

- From the **InfoSewer -> Exchange** pull-down menu, choose the **GIS Gateway** command. This launches the GIS Gateway dialog box.
- Click on the **Add** button on the **GIS Gateway** dialog box and the **Identification** dialog box appears as shown below. Enter the new GIS Cluster ID and description as shown below.

- Click on **OK** to launch the **GIS Exchange Cluster** dialog box as shown below.



- On the **GIS Exchange Cluster** dialog box specify the GIS Data Source to connect the desired Geodatabase to your project by clicking on the **Browse** button ... next to the GIS Data Source: field.
- Locate your Geodatabase using the **Select a Table or Layer** dialog box and click on the **Add** button Add as shown below.



- From the **GIS Exchange Cluster** dialog box, select the appropriate Exchange type form the **InfoSewer Data Source Type:** InfoWater Data Source section of the dialog box. InfoSewer provides 6 different data element types including Junctions, Tanks, Reservoirs, Pumps, Valves and Pipes as well as any other model attribute.

- Specify the **Relate Type** in the Relate Type section of the **GIS Exchange Cluster** dialog box. You may choose either Tabular Join or Spatial Join depending on the type of join that you want to create.

- Choose the appropriate Update Direction. You may choose *0: Bi-Direction*, *1: Load Only* or *2: Save Only* as shown below. To load data from Geodatabase choose either *0: Bi-Direction* or *1: Load Only* option. To update Geodatabase choose either *0: Bi-Direction* or *2: Save Only* option.

- 

- Depending on the **Relate Type** that you specified choose either the Tabular Join or the Spatial Join tab.

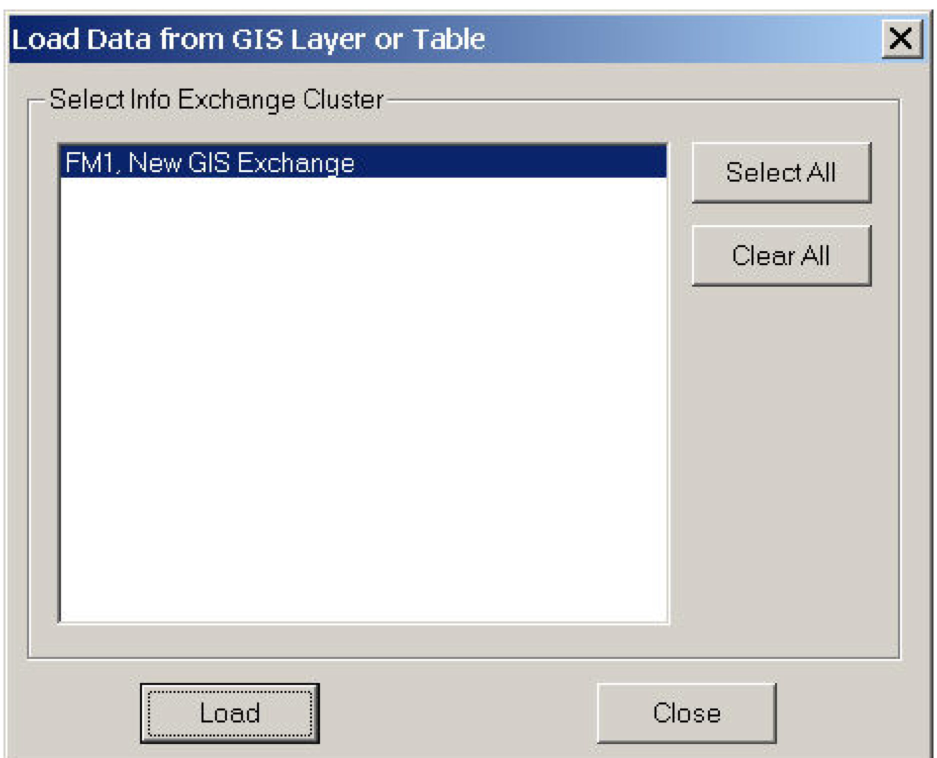
- For Tabular join, specify the GIS ID Mapping Field. InfoSewer will assume that this field contains the unique element IDs for hydraulic modeling.
- Choose **Create New Records** or **Update Existing Records** to create or update InfoSewer data.
- Depending on the Geodatabase that you are loading you can choose to check the 

option. If this option is checked then InfoSewer will update the element geometry as well. Elements will then be updated with the new coordinate geometry.
- Click on the Field Mapping tab and map the appropriate GIS data fields with the InfoSewer fields and finally click on **OK**. This will create a new GIS Exchange Cluster definition.

Load Data from Geodatabases

You can update InfoSewer data directly from your Geodatabases. Once the GIS Exchange Clusters have been defined and linked with the desired Geodatabases, click on the **Load** button 

on your **GIS Gateway** dialog box to load data from the desired Geodatabases.



For **Loading Geodatabases** perform the following:

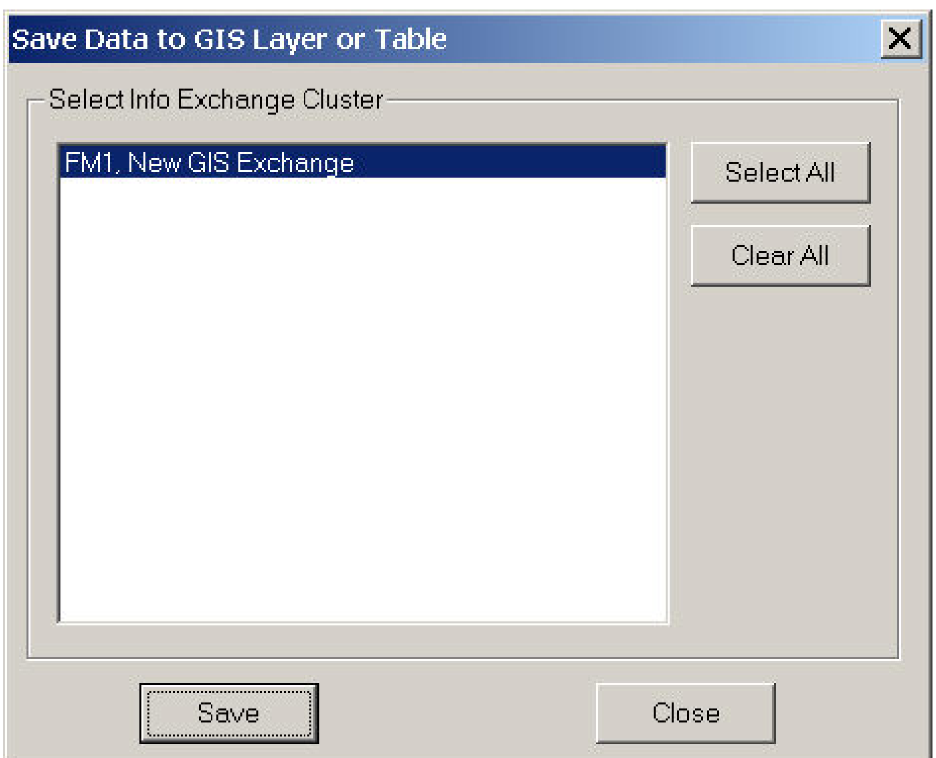
- Launch the **GIS Gateway** dialog box from the **InfoSewer Control Center -> InfoSewer** button **-> Exchange** pull down menu.
- Click on the **Load** button 

to launch the **Load Data from GIS Layer or Table** dialog box and select the desired GIS Exchange Cluster(s). Click on the **Load** button (from the **Load Data to GIS Layer or Table** dialog box) at the bottom to read data from the specified Geodatabases.

Save Data to Geodatabases

You can use InfoSewer to make any desired changes to your Geodatabases. Once the GIS Exchange Clusters have been defined and linked with the desired Geodatabase, click on the **Save** button 

from **GIS Gateway** dialog box to update the desired Geodatabases.



For **saving Geodatabases** perform the following:

- Launch the **GIS Gateway** dialog box from the **InfoSewer Control Center -> InfoSewer** button **->** **Exchange** pull down menu.

- Click on the **Save** button 

to launch the **Save Data to GIS Layer or Table** dialog box and select the desired GIS Exchange Cluster(s). Click on the **Save** button (from the **Save Data to GIS Layer or Table** dialog box) at the bottom to update the specified Geodatabases.

Note: You can only save to a geodatabase that has been loaded in InfoSewer.

Other Related Topics - Geodatabase Methodology, Geodatabase Theory, GIS Cluster -Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster Tabular Join, GIS Field Mapping, GIS Gateway

---

Desired Topology Relationship

Specify the type of spatial join to be utilized during the spatial join here:

- **Completely Contain** - Used to exchange data between a point or line data layer and a polygon. Those elements that are 'completely contained' in the polygon data set will be tagged for the spatial join. Those elements not completely contained will be excluded.
- **Intersect** - When two data layers are intersected, those elements that are contained within and/or cross the intersected layer are tagged for the spatial join. Those elements that do not have a physical intersection are excluded.
- **Within Distance of** - Those elements that are within the user specified distance will be tagged for a spatial join. This option is specified to assign contour elevation data to a point element such as a manhole elevation. In other words, any elements of the selected layer that fall within the tolerance level specified will be tagged for a spatial join.

Once the spatial join options have been specified, proceed to the Field Mapping tab below to map the desired fields prior to exchanging the data. Click here to learn more about **Working with ESRI Geodatabases**.

Where Clause

Allows filtered data exchange, and is used to specify a selection criterion. To conditionally select data from a table, a WHERE clause can be used in the definition of a GIS cluster.

General Usage Format : WHERE column_name operator value

With the WHERE clause, the following operators can be used.

Operator Description

= Equal

<> Not equal. **Note:** In some versions of SQL the <> operator may be written as !=

> Greater than

< Less than

>= Greater than or equal

<= Less than or equal

Example Usage

Using the WHERE Clause, to select only the water meters installed in the city called "Sandnes", we add a WHERE clause as follows:

WHERE City = 'Sandnes'

Note that, in the above example, we have used single quotes around the conditional values. SQL uses single quotes around text values. Numeric values should not be enclosed in quotes.

For text values,

Correct Wrong

WHERE City = 'Sandnes' WHERE City = Sandnes

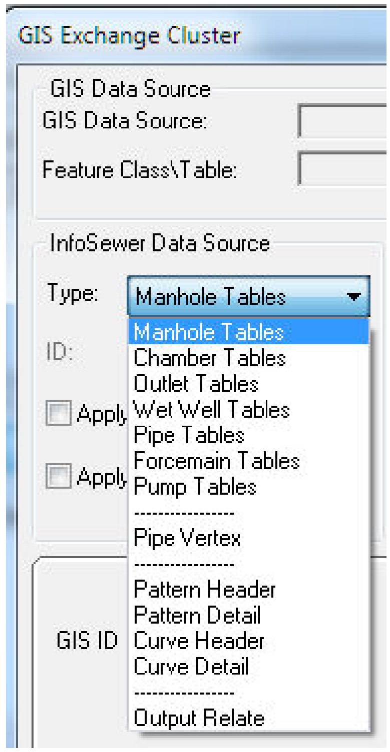
For numeric values,

Correct Wrong

WHERE Year>1965 WHERE Year>'1965'

InfoSewer Data Type

The InfoSewer Element (and associated tables) that will be used in the exchange. You may choose from one of the following element types for the exchange.



us/support-overview

InfoSewer™

Modeling Made Easy

Scenario Menu

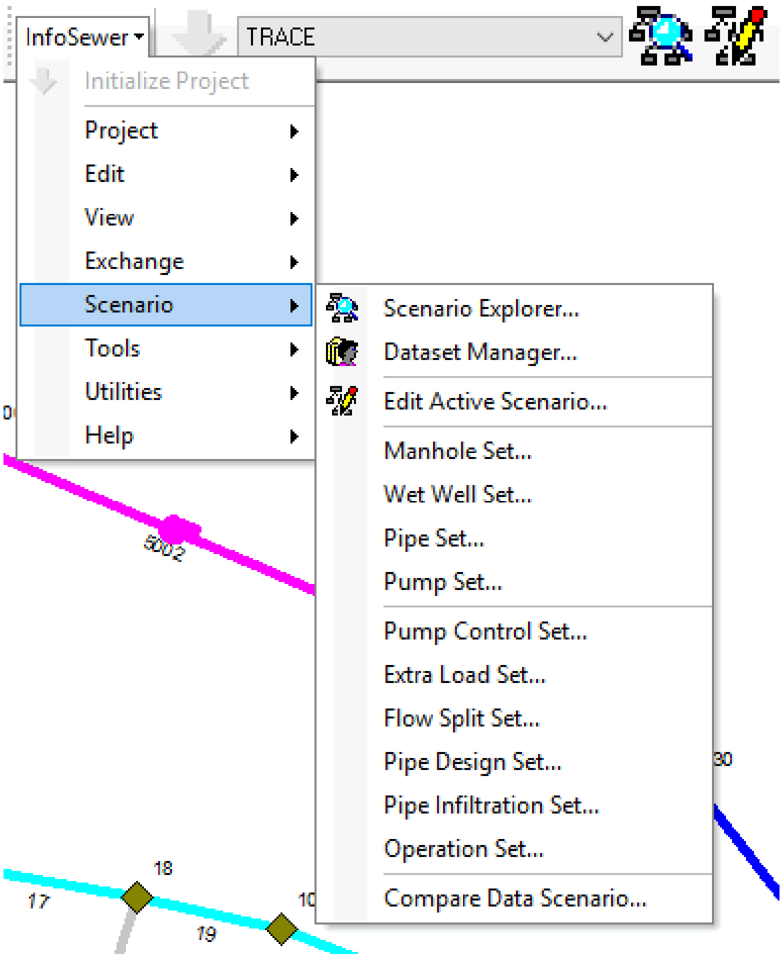
Descriptions of All Scenario Menu commands in InfoSewer.

The **Scenario Manger** is where InfoSewer allows the user to create, delete and modify scenarios. The Scenario Manager is what allows the user to create "what if" situations throughout a sewage collection system.

The **Dataset Manager** is a "quick view" dialog box that allows the user to create, view and edit data sets in one easy to use location. As shown below, the Dataset Manager is used by clicking on any of the tabs to view and edit the listed data set. Use any of the icons at the top of the dialog box to create, edit, clone, copy or delete data sets

A **Data Set** is one of three components that comprise a scenario. Data sets provide the capability to take a one-time “snapshot” of data (database information) in the <u>active</u> model and store that information separate from the network itself.

InfoSewer **Compare Scenario** command allows the user to monitor the differences between any two scenarios in a model.

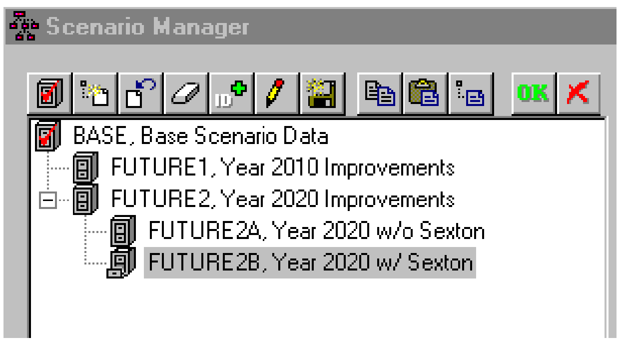

InfoSewer
TRACE
Initialize Project
Project
Edit
View
Exchange
Scenario
Tools
Utilities
Help
Scenario Explorer...
Dataset Manager...
Edit Active Scenario...
Manhole Set...
Wet Well Set...
Pipe Set...
Pump Set...
Pump Control Set...
Extra Load Set...
Flow Split Set...
Pipe Design Set...
Pipe Infiltration Set...
Operation Set...
Compare Data Scenario...
5002
18
17
19
30

Scenario Manager

The Scenario Manger is where InfoSewer allows the user to create, delete and modify scenarios. The Scenario Manager is what allows the user to create "what if" situations throughout a sewage collection system.

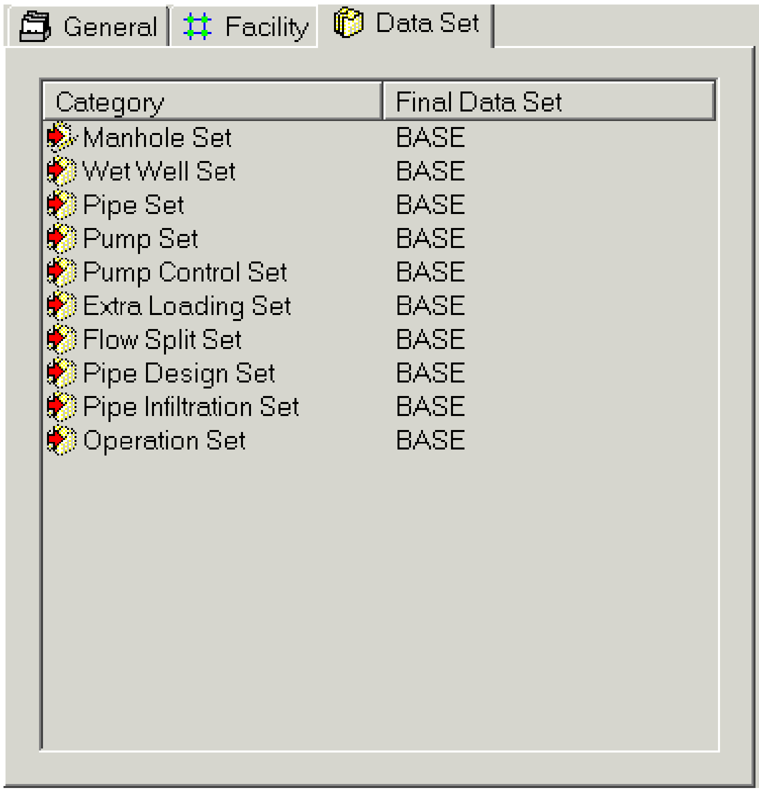
What is a Scenario?

A scenario is a group of InfoSewer facilities, data sets and conditions that are created to reflect a specific modeling situation. With a scenario, you can develop multiple models that are specific to your sewage collection system (ex. Average day loading for a specific service area with unique reporting options).



A scenario is comprised of the following three items:

- **Simulation sets** - General tab
- **Facility set** - Facility tab
- **Data sets** - Data Set tab



Components of an InfoSewer scenario

Each of the three components of a scenario can be further defined as follows:

- **Simulation set** – Created through the General tab, they define the simulation options (durations, timesteps, analysis parameters, etc.) associated with the scenario. There are three different option set types, each storing a logical grouping of simulation options. Click here for more information on simulation sets.

- **Facility set** – Defines the network facilities (components such as pipes, pumps, wet wells and manholes) to be used in a simulation. Only one facility set can be active at a time (facility sets are created through the Facility tab). Click here for more information on facility sets.

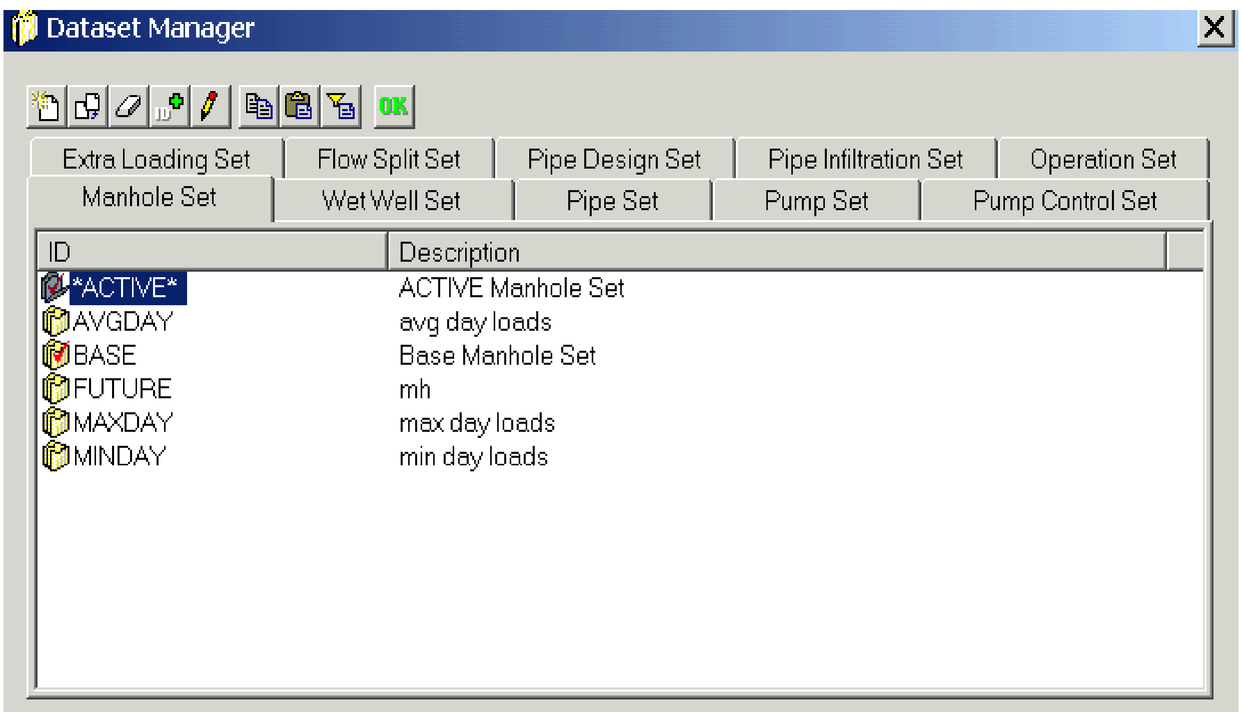
- **Data set** – Stores modeling data (pipe diameter, invert elevations, manhole loadings, etc.) associated with each facility in a separate external database. There are nine different data set types (as seen above), each storing its own unique logical grouping of modeling data. Click here for more information on data sets.

When you define a scenario, you pick the facility, data, and option sets that comprise that scenario. When picking data sets for inclusion in a scenario, you may either specify that a data set associated with a given scenario is included in that scenario independent from other scenarios or alternately may specify that the given data set inherits its contents – properties – from a "parent" scenario. Click here to learn more about Parent-Child relationships and how they are built.

Once you have configured and created a scenario, you can activate that scenario at any time. Once a scenario is activated, any modifications made to any of the databases related to InfoSewer facilities will be changed, but only for the data sets that are related to, and dependent upon, the active scenario. Click here to learn how to activate a scenario.

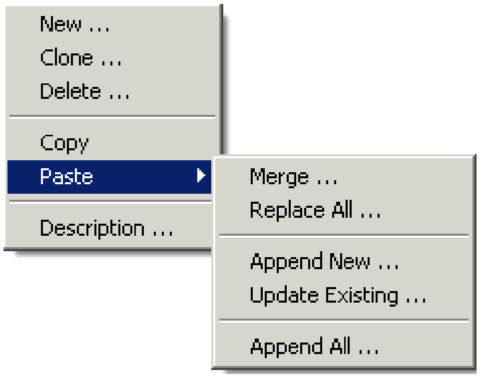
Dataset Manager

The Dataset Manager is a "quick view" dialog box that allows the user to create, view and edit data sets in one easy to use location. As shown below, the Dataset Manager is used by clicking on any of the tabs to view and edit the listed data set. Use any of the icons at the top of the dialog box to create, edit, clone, copy or delete data sets. Click here to learn more about data sets.



Dataset Paste Options

Right clicking on any existing data set brings up a menu with toolbar options. However, the **Paste** function has some options that are unique to this menu. Please see below.



Saved Data Sets may contain different objects. These "paste" options are used to reconcile the difference. A saved conduit set may contain a conduit which does not exist on anther set, since "New" operation works only with the "ACTIVE" conduit set. It becomes even more obvious for data set with multiple detail records (pattern, curve, control, …). By default, if a data set does not contain a particular object (ID), the "active" one will be assumed for network type object or the buffer will be cleared for data type object when the data set gets activated. Now here are what various "paste" operations do:

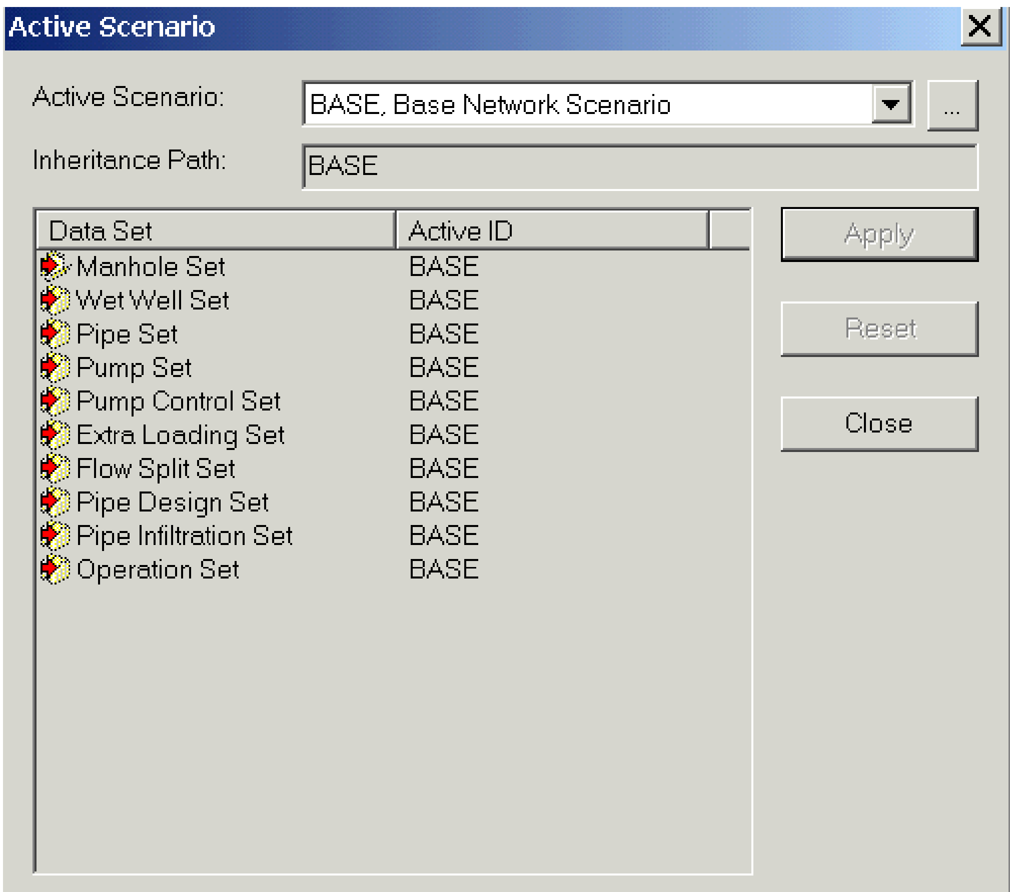
For Target Data Set:

- **Append New**: added only new objects from the source
- **Update Existing**: update objects which can also be found in source
- **Merge**: Append New + Update Existing
- **Replace All**: replace target entirely with source
- **Append All**: blindly append all records from source (i.e. control set aggregation)

Active Scenario

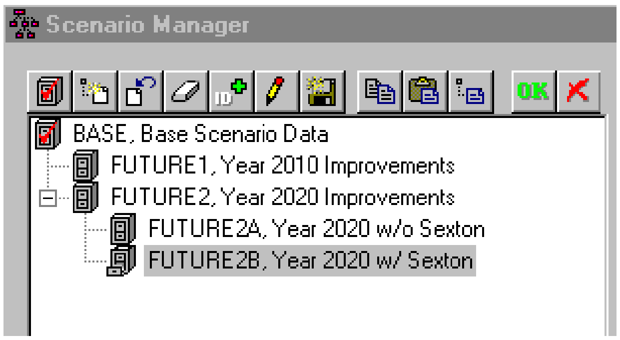
The Active Scenario dialog box allows the user to select any of the scenarios created in an InfoSewer project and make it active. Once a scenario is made active, each of the three facets that comprise a scenario (facility sets, data sets, simulation sets) also become active.

The Active Scenario shows you which scenario is currently active while the Inheritance Path shows the user which parent this scenario is dependent upon. The Data Sets also show which data sets are currently being assigned to this particular scenario. Here, the user can change the Data Sets of the active scenario by doubling-clicking on the desired data set. A dialog box will appear showing all of the data sets available to the user.



Parent - Child Relationships

The relationship between a parent scenario and its child is defined as inheritance. InfoSewer shows this relationship in the form of a directory tree. All scenarios are children of the *BASE* scenario until some element of the child is made unique from the parent (such uniqueness comes in the form of data sets, facility sets, or option sets). In the example below, FUTURE2 is a child of BASE and FUTURE2B is a child of FUTURE2. When a change is made to a parent, unless some facet of a child is unique, it will inherit the change through inheritance.



What is Inheritance?

Inheritance refers to the relationship between a parent scenario and one or more of its children. Rather than each scenario existing independently of other scenarios, a scenario may inherit one or more of its properties from a parent scenario. With this capability, there is no need to enter redundant information to numerous scenarios that share the same data. Instead, you simply develop a master scenario (referred to as the parent) and develop one or more scenarios whose properties are dependent on the parent scenario.

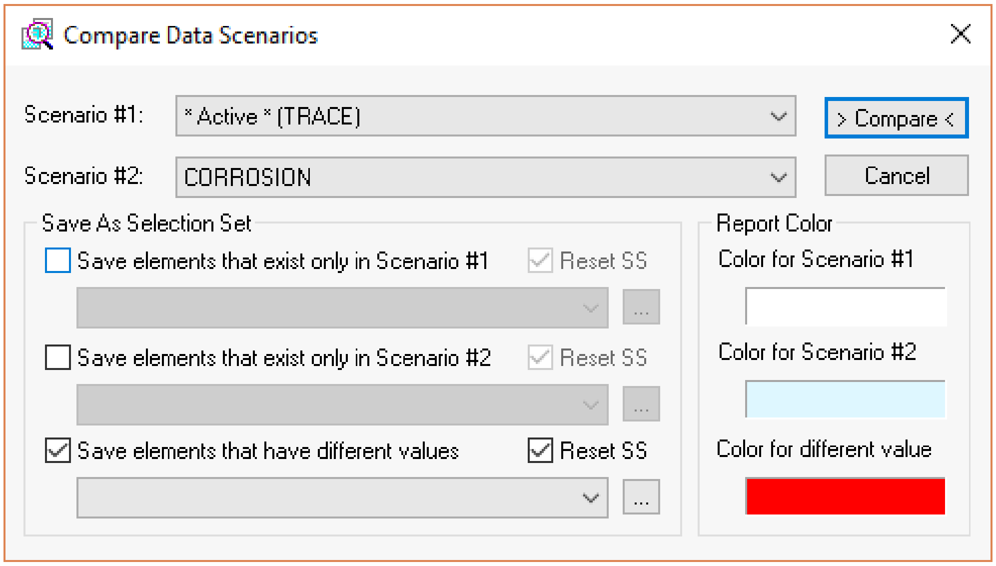
For example, supposing that a child has a manhole set that is exactly the same as the parent. When a change is made to the parent manhole set, the change will also be reflected in the child. To make the child independent with respect to system loadings, create a new manhole set specifically for the child and assign the new manhole set to the child scenario. In this case, it will ensure that the child is independent of the parent with respect to its manhole set.

Explained further, once a child is created from a parent, the relationship is dynamic, meaning that when some piece of data is changed in the parent (like a pipe set), the child reflects that change as well. However, where a property is explicitly changed in a child scenario, the inheritance relationship for that property is broken and the change is reflected only in the child.

Compare Data Scenarios

InfoSewer Compare Scenario command allows the user to monitor the differences between any two scenarios in a model.

To compare two scenarios, from the **Scenario** menu, select **Compare Scenarios**. The following dialog box will be displayed:



- **Scenario #1** - The first scenario to be compared.
- **Scenario #2** - The scenario to compare against Scenario #1.
- **Save As Selection Sets:** You can save selection sets based on the three analysis results. These can be used for later reference if you wish to make amendments based on these results.
  - Elements only in Scenario #1: For any elements that exist in the first scenario, but do not have a matching element in Scenario #2
  - Elements only in Scenario #2: For any elements that exist in the second scenario, but do not have a matching element in Scenario #1
  - Elements with different values: When comparing elements in either scenario, a value is found to differ. This may not be undesirable, but will serve to highlight specific data items,

such as different pipe diameters or manhole loadings between the two scenarios, for checking purposes.

- **Reset SS:** Stands for Reset Selection Set. You may want to use a regular set of selection sets for Compare Data Scenarios. If this is the case, each time the Compare Scenario command is carried out, the existing selection set is cleared.
- **Report Color:** You can select how the elements will be displayed in Compare Scenario's output report. This provides a quick way of comparing data between scenarios.
- **Compare:** When two scenarios have been specified, click the compare button to perform the comparison.
- **Cancel:** Cancel the compare operation.

Comparing Scenarios:

Once the Compare Data Scenarios dialog box appears, the user will see that the Active scenario (in this case, BASE) is selected. You can compare the Active scenario and another scenario in a model. Alternatively, you can select another scenarios other than the Active scenarios to compare to another scenario.

To compare two scenarios, perform the following:

- Select the first scenario you wish to compare from the Scenario #1 drop down list.
- From the Scenario #2 drop down list, select the scenario you wish to compare to the first scenario.

If you wish to save the location of any elements that have data flagged by the Compare Scenario feature, you can save selection sets to refer to at a later time (i.e. from the Domain Manager). For information on creating a selection set, refer to selection sets.

Save As Selection Sets:

You can select an existing Selection Set to use, or you can create a new selection set. To save the Selection Sets, perform the following:

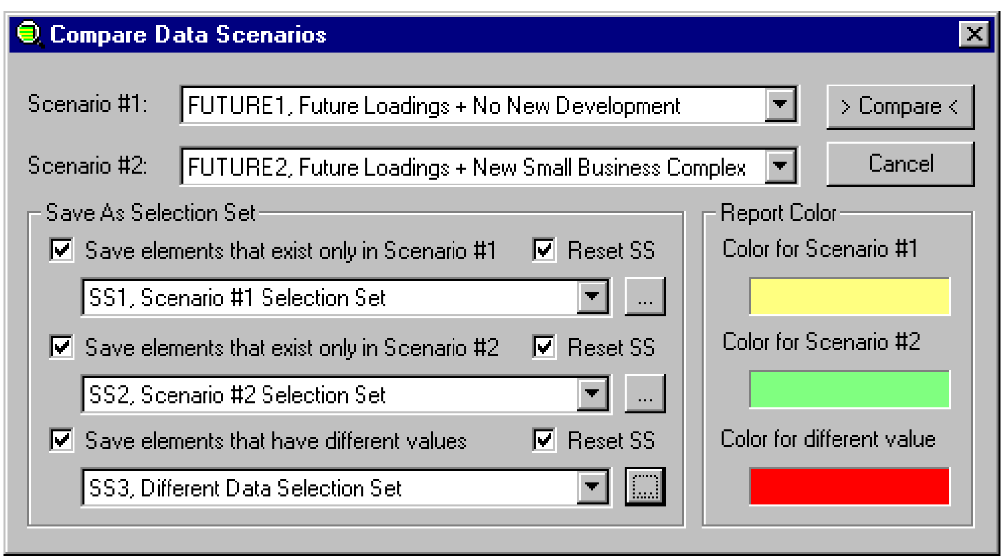
- If you wish to save a Selection Set based on the elements that are in Scenario #1, but not in Scenario #2, check the Save elements that exist in Scenario #1 only option.
- If you wish to save a Selection Set based on the elements that are in Scenario #1, but not in Scenario #2, check the Save elements that exist in Scenario #2 only option.
- If you wish to save a Selection Set based on elements that differ between the two scenarios, check the Save elements that have different values option.

Please note, you must have different selection sets selected for each comparison result that is to be reported. If you wish to reset an existing Selection Set, ensure that the Reset SS option is checked.

Changing Report Colors

To alter the colors selected for each of the Compare Scenario outputs, click on the color swatch you wish to change, and a color pick dialog will appear for you to select a new color. Each color that is selected in this dialog will be reflected in the Data Scenario Comparison Report legend.

Once you have completed the configuration of the Compare Scenario dialog, you will see something like this:

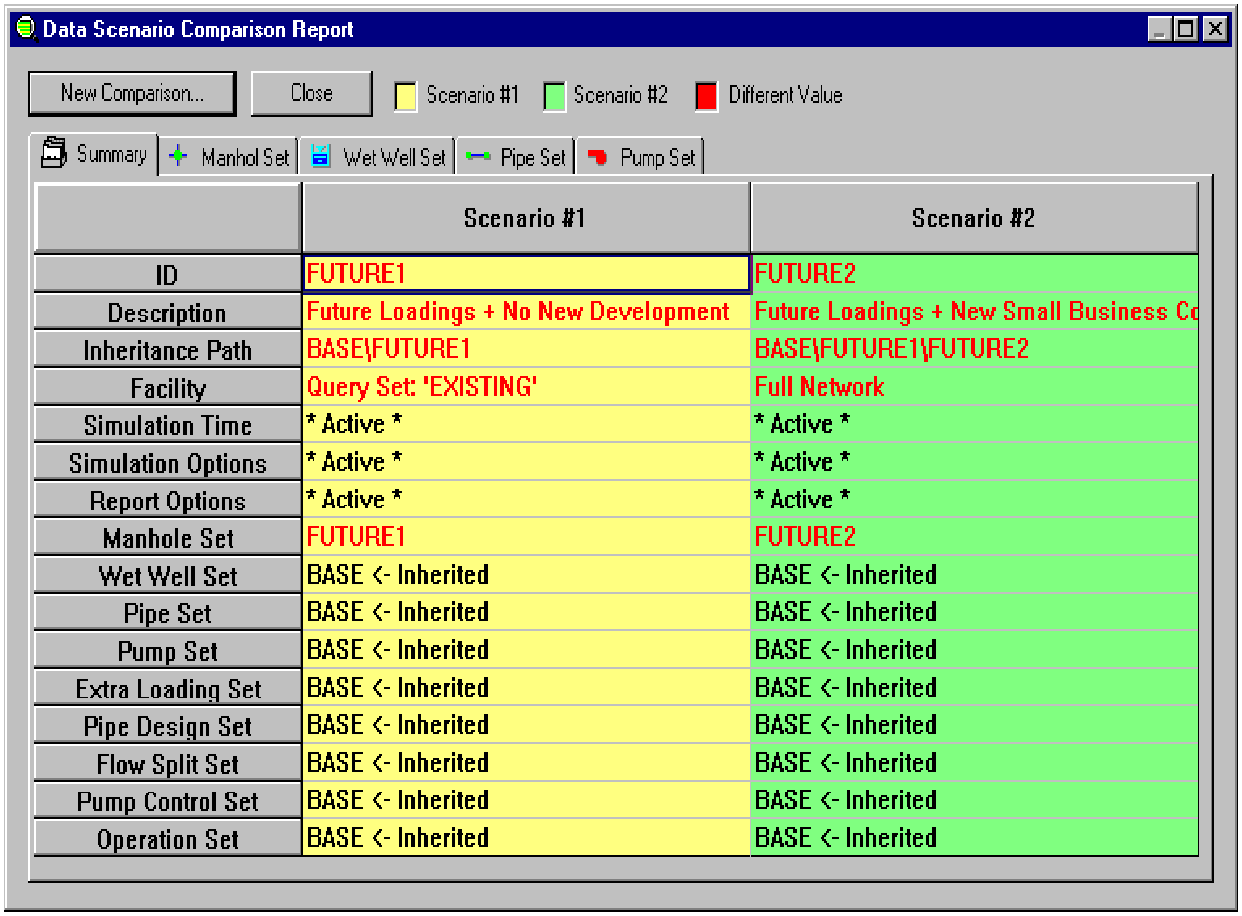


To conduct a comparison of scenarios, press the 

button.

Viewing results

Compare Scenarios presents the results in a format that allows to readily compare each scenario's data. When the comparison analysis is completed, you will be presented with a display similar to the Graph Reports.



Features of the Data Scenarios Comparison Report:

- **New Comparison:** This button will re-open the Compare Scenarios dialog to enable the user to conduct another comparison of scenarios.
- **Close:** Closes the report window
- **Report Color Legend:** Each of the colors selected in the Compare Scenarios dialog are featured for easy reference
- **Summary Tab:** This provides users with a summary of all the data associated with the scenarios. The following is what each of the

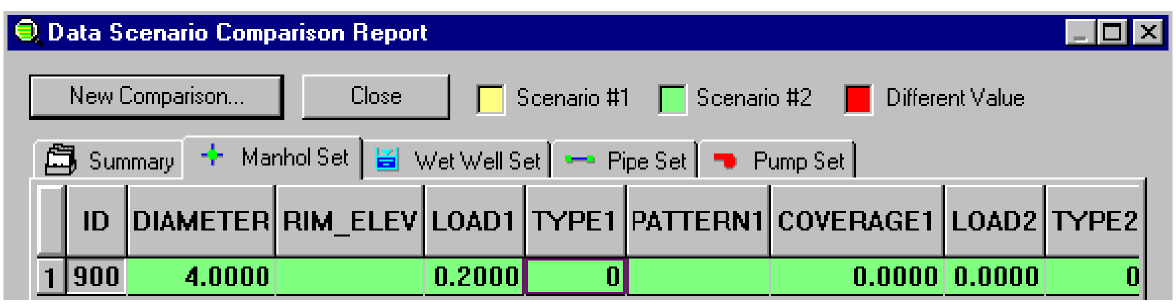
summaries describe:

- ID – Each scenario's ID
- Description – Each scenario's description
- Inheritance Path – Shows each scenario's inheritance (from each of it's parent scenarios, and their respective parent scenario). For further explanation of inheritance, refer to Scenarios.
- Facility – Describes how the scenario has been created, ie. With a query set, existing network, or Intelliselect. Refer to Facility Sets for further information.
- Simulation Time, Simulation Options, Report Options – Settings made under the Run Manager.
- Data Sets – Each data set that is found within a scenario, configuring what data appears within the scenario.

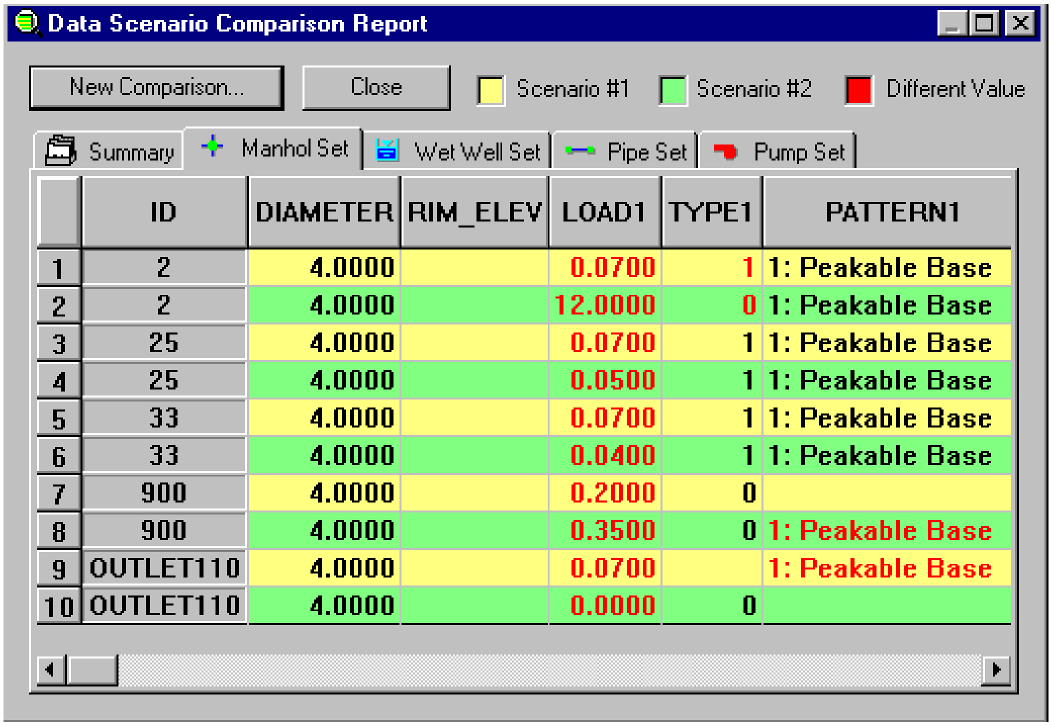
Data Set Tabs: For each data set tab, data that is considered different between scenarios will be presented. To view the comparison results of the scenarios for each facility data set, click on a tab.

Interpreting Data Set Comparison Results:

When viewing data in the data set tabs, you will be presented with three possible results. For this example, we will refer to the following figures:



In this first example, you can see that Scenario #2 has a manhole with ID 900 and a loading of 0.2 cfs (note the coloring of the row).



	ID	DIAMETER	RIM_ELEV	LOAD1	TYPE1	PATTERN1
1	2	4.0000		0.0700	1	1: Peakable Base
2	2	4.0000		12.0000	0	1: Peakable Base
3	25	4.0000		0.0700	1	1: Peakable Base
4	25	4.0000		0.0500	1	1: Peakable Base
5	33	4.0000		0.0700	1	1: Peakable Base
6	33	4.0000		0.0400	1	1: Peakable Base
7	900	4.0000		0.2000	0	
8	900	4.0000		0.3500	0	1: Peakable Base
9	OUTLET110	4.0000		0.0700		1: Peakable Base
10	OUTLET110	4.0000		0.0000	0	

In this next example, between scenario #2 and #3, the manhole with ID 900 has a loading in the first scenario of 0.2 cfs, but in the second scenario, it has a loading of 0.35 cfs.

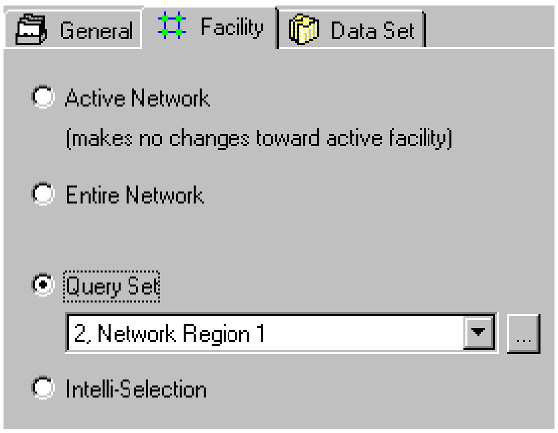
The process of comparing scenarios can be applied to any number of facility attributes, including:

- Manhole Data
- Pipe Data
- Wet Well Data
- Pump Data

Facility Sets

The facility set defines those network components (pipes, pumps, manholes, wet wells) that will be considered during the next simulation. To model a subset of components, the user has many options to choose from.

*(Note: It is very important to understand the relationship between the Facility Manager, the *active* facility set and the facility set utilized by the Scenario Manager. Click here to read a detailed, step-by-step process for associating facility sets with scenarios.)*



(Note: The default facility set is the entire network.)

The facility set may include the entire network model or a subset of network components. Only facilities in the active facility set are displayed in H2OMAP Sewer. Facilities that are not active (i.e., those not in the current facility set) are removed from the map display.

Activating Facility Sets

Facility sets are activated immediately as you choose the Add or Activate All buttons on the Facility Manager dialog box. All network components that meet the criteria of your selection method (see Create an Active Facility Set) will remain displayed. Those components not included in your facility set become inactive and are removed from the map display.

Inactive components remain in the InfoSewer project, however their associated graphics are either hidden or displayed depending if the user has the "View Inactive Facility" option checked.

To activate the entire network (i.e., restore all network components in your project for displaying, editing, and inclusion in a simulation), open the Facility Manager dialog box and choose the "Activate All" button.

Associating a Facility Set with a Scenario

When you create a new scenario, one of the options that must be chosen is how you define the network components to be included in the scenario. There are four separate and unique options available for associating a facility set with a scenario. They are as follows:

- **Active Network** – When this option is chosen, InfoSewer will include the current facility set (what the user has currently created and activated via the Facility Manager) in a scenario. Again, all currently activated components (i.e., those in the active facility set - and visible on the map display) at the time the scenario is activated will be included in the scenario.

*(Note: With this option, loading a scenario that has the "Active Network" option will not change the current active facility set. In other words, the facility set included with an "Active Network" will vary every time according to the *active* facility set created from the Facility Manager.)*

- **Entire Network** – When this option is chosen InfoSewer will always include the entire network in a given scenario. Therefore, no matter what facilities are active (in the current facility set from the Facility Manager) at the time you load the scenario, that facility set will be replaced with a new facility set representing the entire suite of network components in the open InfoSewer project.

- **Query Set** – When this option is chosen InfoSewer will evaluate one or more database statements you specify and all network components meeting those criteria at the time the scenario is activated will be included in the scenario's facility set. Those facilities will be displayed and all facilities not meeting the entered criteria will be disregarded from the simulation and will be removed from the map display. With this option, the user can either select a query set or assign a previously created query set. Click here to learn more about query sets.

(Note: This option is useful for creating facility sets based on database attributes. For instance, you could assign service area designations to all components in the current InfoSewer project and then specify database query statements to select and include components in each service area in different modeling scenarios.)

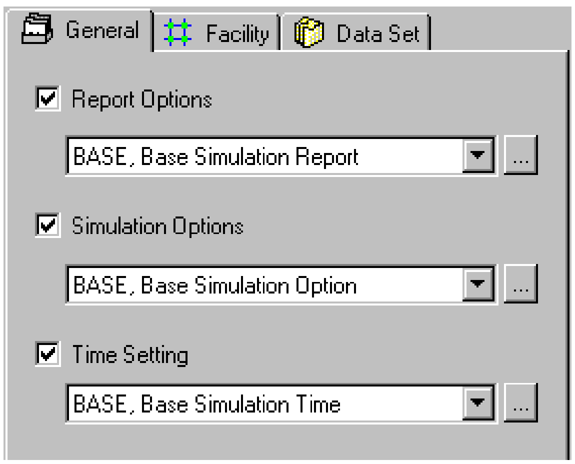
- **Intelli-Selection** – This option is like the "Active Network" option, but differs in that the Intelli-Selection remembers the *active* facility set from the Facility Manager and will reinstate that facility set every time the scenario is activated.

Simulation Set

The Simulation Set is defined by the following:

- **Report Option Set** – Standard reporting options associated with a scenario.
- **Simulation Option Set** – Simulation options associated with a scenario, including hydraulic analysis criteria, peaking equations, design constraints, etc.
- **Time Option Set** – Simulation duration and timesteps associated with a scenario (EPS simulations only).

Click on any of the three option sets below to learn more...



Data Sets

A data set is one of three components that comprise a scenario. Data sets provide the capability to take a one-time “snapshot” of data (database information) in the active model and store that information separate from the network itself.

In essence, InfoSewer creates separate database tables for each data set created, allowing the user to manipulate the database characteristics of the data set, separate from the "Base" data set. Once new data is stored in a data set, it may be reloaded back into the active model (via a scenario) at any time.

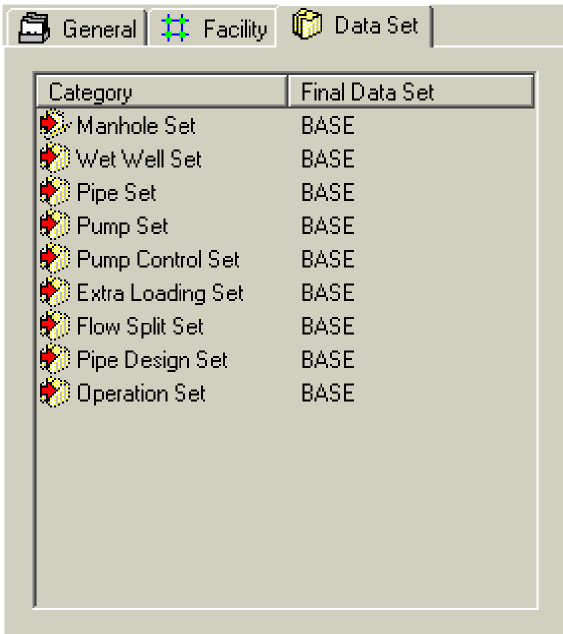
To learn more about data sets, click on the following subjects.

- **Data Sets Defined**
- **Step to Create a Data Set (Tutorial)**
- **How do I Activate a Data Set?**
- **What Happens During a Data Set Activation?**

Data Sets Defined

Category - Represents the specific subset of a data set.

Final Data Set - Represents the data set created and selected by the user or inherited from the parent scenario. Click here to learn more about inheritance.



- **Manhole Set** – Modeling data associated with manholes. Manhole sets are used to save modeling data on manhole loadings and pattern identifiers and retrieve those data as part of a scenario. The attributes stored in a manhole set are the baseline loads, the type of load, associative patterns, and the coverage. The Auto-Manhole Reset preference governs how loadings are assigned when activating manhole sets.

- **Wet Well Set** – Modeling data associated with wet wells. Wet wells are used to save modeling data on wet wells and retrieve those data as part of a scenario. Wet well sets contain modeling information including type of wet well (constant or variable-area) and related wet well node characteristic data such as bottom elevation, initial water level, diameter, minimum and maximum levels, and any curve defining a variable area wet well.

- **Pipe Set** – Modeling data associated with pipes. Pipe sets are used to save modeling data on pipes and retrieve those data as part of a scenario. Pipe sets contain modeling information including pipe diameters, lengths, Manning's roughness coefficients, upstream and downstream inverts and presence or absence of parallel pipes.

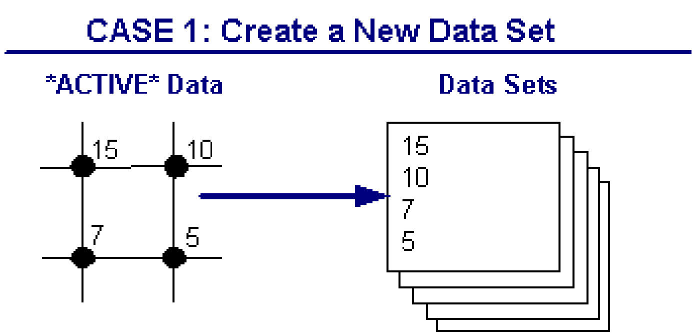
- **Pump Set** – Modeling data associated with pumps. Pump sets are used to save modeling data on pumps and retrieve those data as part of a scenario. Pump sets contain modeling information including curve type and associative parameters.

- **Control Set** – Simple controls used for pumps located at wet wells. Control sets contain logical controls for pumps (on/off settings) as part of a logic set that are stored and recalled as part of a scenario.
- **Extra Loading Set** – An extension of a manhole set, the XLoad set is used to assign additional loadings to a manhole node and include the same data parameters as a manhole set.
- **Flow Split Set** – Flow split percentages or patterns assigned by the user. Flow split sets are only assigned if the automatic calculation is overridden by the user.
- **Pipe Design Set** – Design and analysis criteria curves as well as replacement and duplicate curves assigned by the user to various facilities.
- **Pipe Infiltration set** – Depending on the infiltration modeling option selected, infiltration rate and pattern needs to be created.
- **Operation Set** – All patterns and curves in the H2OMAP Sewer project at the time the operation set is created.

Steps to Create a Data Set

Data sets can be created in one of two fashions:

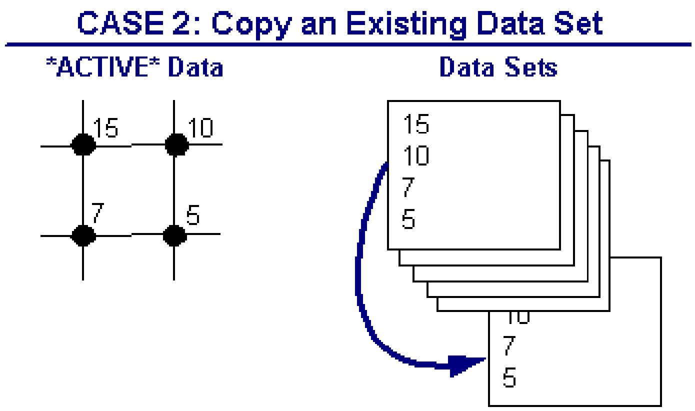
- Create a new data set from the current *ACTIVE* or currently loaded data set into a new data set.



(Note: For all new data sets (with the exception of manholes sets), creating a new set will copy the original database table of the

**ACTIVE* set to the new data set database table. When a new Manhole Set is created (not cloned), the database field for system loadings will be empty.)*

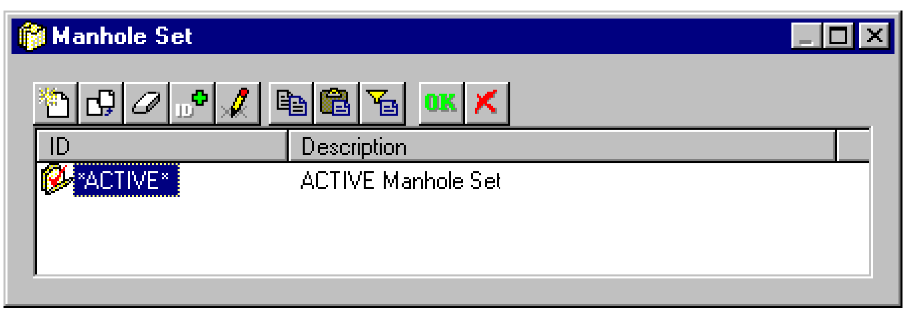
- Clone data from a previously-defined data set into a new data set.



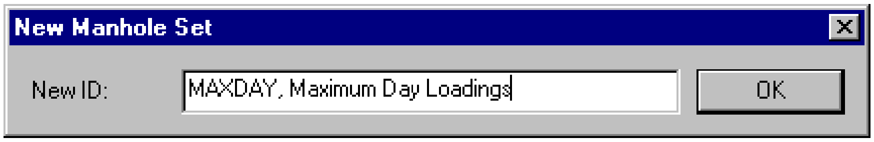
[Example Data Set Creation (Tutorial)]()

In this example, steps for the creation of a cloned manhole set are provided. For a more detailed tutorial, please see Chapter 3 of the hard-bound User's Guide.

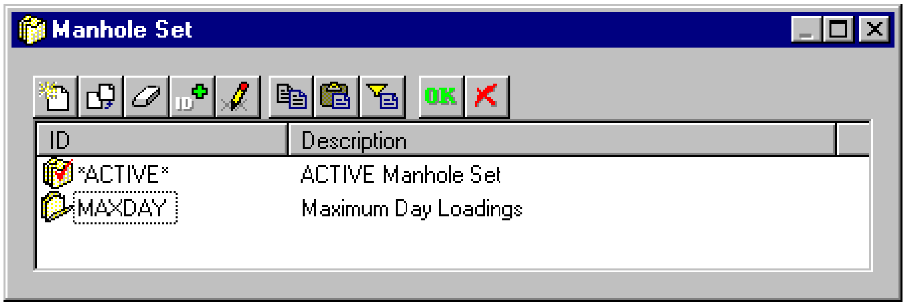
- From the **Scenario** menu, select **Manhole Set**. Once selected, the following dialog box will appear:



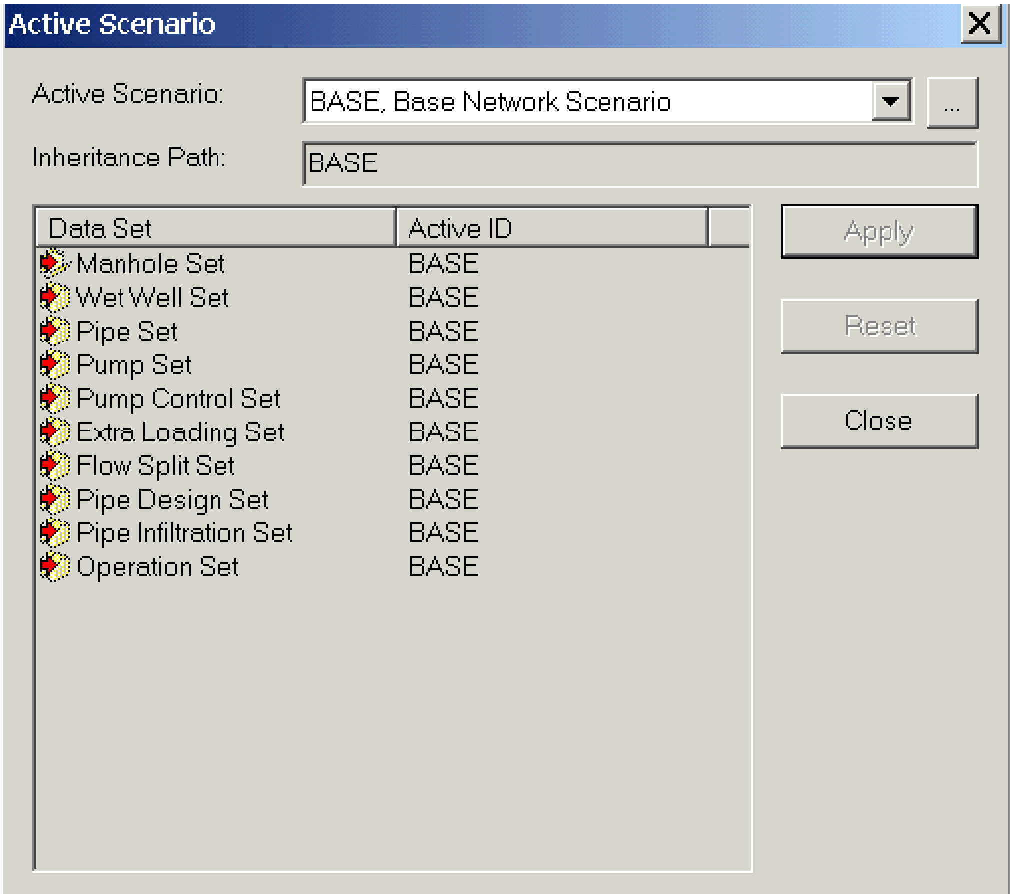
- The *ACTIVE* manhole set represents the Base loading set. Select the Clone icon to replicate the existing *ACTIVE* manhole set. (By selecting clone, the user is requesting that all information in the *Active* database be copied to the new manhole set database - including system loadings.) Once selected, the following dialog box will appear:



- Enter in the unique ID for the manhole set (in this case, MAXDAY) and then provide a description for future reference. Once entered, select OK and the following dialog box will appear:



- Notice that there is no red check mark next to the new MAXDAY manhole set. The reason is because the red check mark indicates whether a data set is currently active or not. To edit the data related to the MAXDAY manhole set it must first become associated with a scenario and made *active*. (Once a data set becomes active, any edits made to a data set will be saved and stored ONLY in the *active* data set.) The next step is to click OK to exit the manhole set dialog box and associate your new data set with a new (or existing) scenario. Go to the **Scenario** menu and select **Active Scenario**. Once selected, the following dialog box will appear:



- The currently *active* scenario will appear in the Active Scenario drop down box (in the example above, FUTURE1). To create a new scenario, select the [...] icon next to the Active Scenario drop down box to see the Scenario Manager dialog box. Highlight the BASE scenario and click on the "New Child" icon. Specify an ID and description for the new scenario and click the OK button. The user is now able to edit any data set for the new scenario by double-clicking on the Category Name (in our case, the Manhole Set) and selecting the MAXDAY manhole set. Once the MAXDAY manhole set is associated with your new scenario, select the Activate icon to activate this scenario.

- At this point, any modifications made to the manhole loadings will be saved to the MAXDAY manhole set (not the original BASE set). So, in essence, by creating this new scenario that now contains the new MAXDAY manhole set, we have created a new manhole database table that is only related to this manhole set (and to the new scenario). At any time the user can go back to the Scenario Manager and specify a different manhole set (or any other data set). To make it active and edit the database related to the active data set, use the Active Scenario command. Any changes made will be saved and are ready to be "recalled" by the user once any specific scenario is made active.

Activating Another Data Set

There are two methods for loading new data or option sets into the current InfoSewer project:

- Associate a data set with a custom scenario and then activate that scenario – The primary method for activating data sets is by associating them with one or more custom scenarios and then activating one of your scenarios. Definition of custom scenarios and related data sets is accomplished using the Scenario Manager command.
- Load a data set at any time – To change a data set for the currently active scenario, from the **Scenario** menu, choose the **Active Scenario** command. You may load one or more data sets into the active scenario. By doing so, the current modeling information in the open H2OMAP Sewer project will be copied to the active data sets before switching to the new data sets you selected. The contents of the newly-selected data sets are then copied into the open H2OMAP Sewer project and are immediately available.

(Note: The user cannot switch data sets for the BASE (default) scenario. The user may only switch data sets for custom-developed scenarios.)

The second option is most commonly used when the user has not developed multiple scenarios - yet still wishes to interchange data sets for modeling purposes (trial and error before scenarios are created).

What Happens During a Data Set Activation

InfoSewer activates different data sets in different fashions. The following describes how each data set type is activated and data from those data sets loaded into the *ACTIVE* scenario:

Pipe, Pump, and Wet Well Sets

When these data sets are activated and where a match exists between a record in the data set and a network component in the activated facility set (the network you see on the map display), the data from the data set overwrite any data currently loaded for those network components. Where there is no match between records in the data set and activated network components, those components retain their current data values.

Control, XLoad, Split, Design and Operation Sets

When these data sets are activated, H2OMAP Sewer first clears all data (related to these data sets) from network components in the activated facility set (the network you see on the map display). Then, where a match exists between a record in the data set and a network component in the activated facility set, the data from the data set are assigned to those network components. Where there is no match between records in the data set and activated network components, those components remain with zero or null data values.

Manhole Sets

By default, manhole set activation follows the same rules as Pipe, Pump, and Wet Well Sets above, where data in the *ACTIVE* data sets are retained and only overwritten if there is a match between records in the data set and activate network component facility set. However, you may specify that manhole set activation follow a similar procedure as Control, XLoad, Split, Design and Operation Sets, where data in the *ACTIVE* data sets are first cleared before loading data in the data sets.

The setting of the Auto-Loading Set preference controls the loading of the manhole set procedure. When ON (checked), loading sets are loaded similar to Pipe, Pump, and Wet Well Sets. When OFF, manhole sets are loaded in a fashion similar to all other data sets. To use this function - from the **Tools** menu, select **Preferences**. Under the Map Operations tab, select Auto Manhole Set to control manhole set activation.

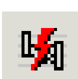
Tools Menu

Descriptions of All Tool Menu commands in InfoSewer.

*Use the different options provided here to run your model (**Run Manager** and **Batch Simulation Manager**) and analyze your results through the different options such as the **Customized Report, Query report, Output Report Manager**. Modify your output Units through the **Output Unit manager**. Create **Domains** and **Facility** sets through the **Domain** and **Facility** Managers respectively. Activate the InfoSewer modules through the **AddOn Extension Manager**. Also create and view your **Animations** through the **Animation Viewer**.*

The Run Manager is used to perform simulations and to manage simulation output sources (results). Three individual simulation types are available from the Run Manager: Steady-State , Design, and Extended Period Simulation/Dynamic simulation (including quality analyses). You may also use the Batch Run Manager to run multiple standard simulations with one operation.

The Batch Run Manager is used to run models for numerous user-selected scenarios in a single operation. This command is especially useful where several simulations are simultaneously required for a large model. With the Batch Run Manager, the user can select the desired scenarios and run each model in a "batch" process.

 **Pipe Profile for Output** - Profiles the Output data for a series of links.

Like a customized report, a Query Report is used to selectively choose which input and output data will appear in a report. However, unlike a customized report, a query report is required to relate data from an Output Relate with output data in a report format. In other words, the only way to see output data in a report format is to create a query report that contains a desired output relate.

The **Output Report Manager** provides you with a means to access all your Output variables in report and graph form for all your simulations.

A **Domain** is a *temporarily* selected subset of network components. The domain can be used for a variety of purposes including group editing operations, mapping, contouring, etc. The Domain Manager is used to create, edit and delete elements contained within a domain. In conjunction with the Domain Manager, InfoSewer also has the Edit Selection Attributes and Edit Domain Attribute icons.

The **Facility Manager** is used to create and maintain the active facility set. The active facility set defines the network components in a current model that will be considered during the next simulation run(s). Facility sets can also be associated with a scenario via the **Scenario Manager**.

The ability to determine costs for replacing sewer collection infrastructure can be one of the most timely and difficult tasks in preparing a master plan. The **Pipeline Costing** function has been provided to assist the user in establishing these costs for replacing certain pipes.

The **Project Preferences** command is used to control the InfoSewersystem settings. All changes made on the Preferences dialog box must be applied prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

- **Run Manager** - The Run Manager is used by InfoSewer to specify hydraulic options, create output sources and to run a hydraulic simulation (Steady State Analysis, Steady State Design or EPS).
- **Batch Simulation** - A batch simulation can be established by the user to run a series of model scenarios in a batch process. Click here to learn more.
- **Pipe Profile for Output** - Allows the user to create a hydraulic profile of the pipe output data. Click here to learn more.
- **Query Report** - A query report is used to further customize a report with output model data. Features of a query report include second level queries and output relates which contain report features. Click here to learn more.

- **Output Report / Graph** - The Output Report Manager allows the user to create, view and edit the output data created by simulations within InfoSewer. Specifically, this feature enables the user to view reports and graphs for any output sources created through the Run Manager.
- **Output Unit Manager** - The Output Unit Manager is used to alter any of the modeling units required by InfoSewer. This is where a user can change English units for a project to Metric by redefining the units used for length, pressure, head, etc.
- **Domain** - Domains are used by InfoSewer to quickly select and de-select elements for quick editing and mapping purposes. The creation of domains are required prior to using the Map Legend under the Control Center. To learn more about domains, see the Domain Manager. To learn more about mapping and graphical modifications in an InfoSewer Sewer project, see the Map Legend.
- **Facility** - Facilities are used to create active elements prior to running simulations or creating scenarios that are dependent upon certain "active" elements. See the Facility Manager to learn more.
- A**nimation Editor** - The Animation Editor allows the user to create a "movie" of specific elements within the collection system.
- **Animation Viewer** - The Animation Viewer allows the user to see a "movie" of specific elements within the collection system.
- **Pipe Invert Calculator** - Allows the user to "back-calculate" pipe invert elevations from known pipe slopes. Click here to learn more.
- **Pipe Invert Translator** - Provides a way to "translate" invert information from node to pipe. Click here to learn Mode.
- **Pipe Direction Calculator** - Click here to learn more.
- **Pipeline Costing** - The Pipeline Costing tool allows the user to assign a costing equation to either a user selection or to a domain. Click here to learn more.
- **Elevation Extractor**, as the name implies, extracts rim elevation of manholes and outlets from digital topographic maps such as the DEM

thus simplifying the effort it takes to create this data. The elevation extractor interpolates ground surface elevations for the nodes in the drainage system from digital maps of various formats including vector data, raster data, and spot elevation data and other digital contour data formats compatible with ESRI. Based on the notion that the commonly available elevation maps are for ground surfaces, not for invert elevations of nodes, the invert elevation saved to InfoSewer's node database by the elevation extractor is the ground surface elevation interpolated from the digital map for the node less the maximum depth of the node. In other words, before running the elevation extractor, it is desirable for the modeler to enter maximum depth data for the nodes in the collection system.

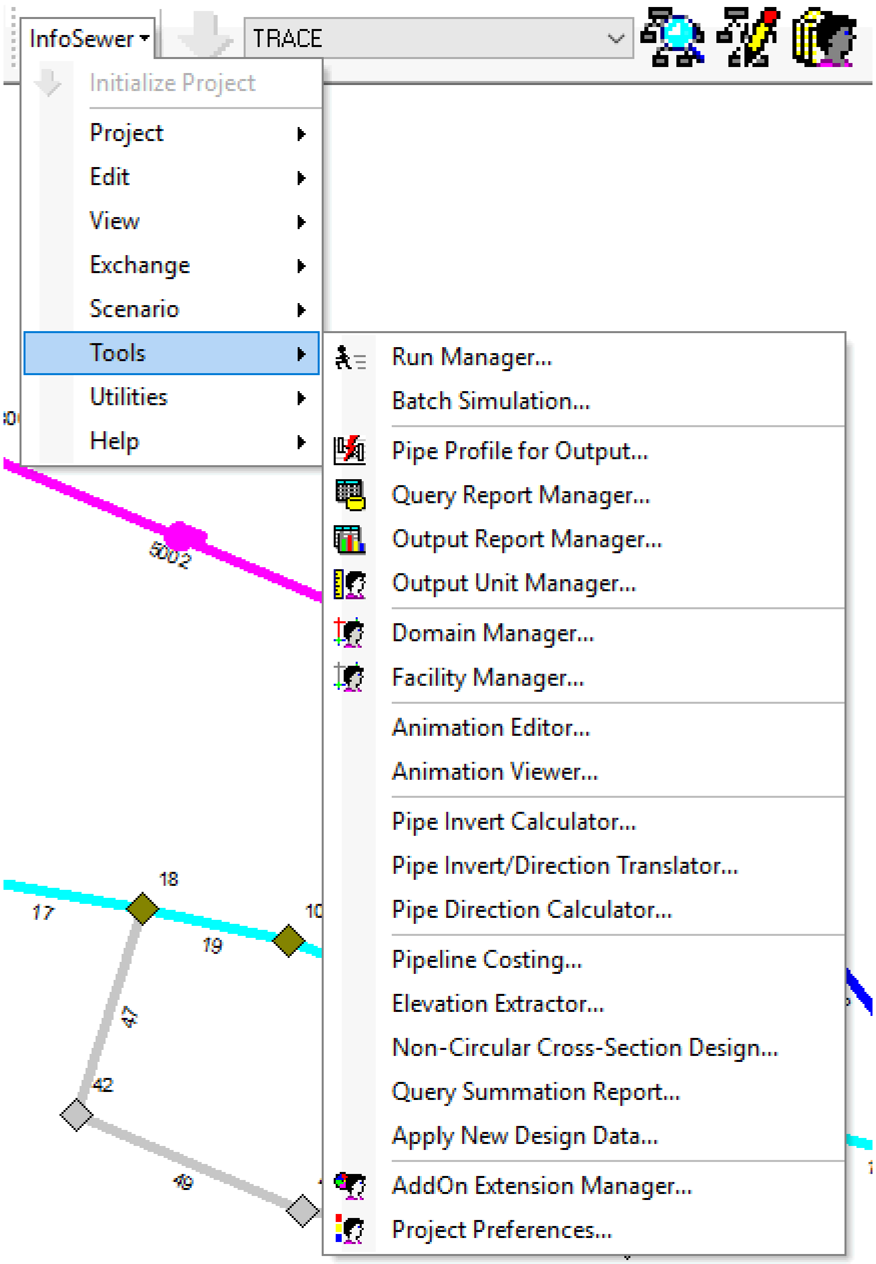
- **Non-Circular Cross-Section Design** - This is a design table editor that enables the user to provide data required during design simulations of non-circular cross-sections. Click here to learn more about the design table.

- **Query Summation Report** - Allows presentation of existing DB queries in report or chart format.

- **Apply New Design Data** - This tool exports design results - specifically pipe diameter and pipe invert elevations and manning's n for new pipes - to the specified model database enabling the user to build a working model from the design result. Selection sets can also be created from pipes that succeeded and failed during the design simulation. Click here for more information

- **Add-On Manager** - This command allows the user to enable add-on features to InfoSewer like the Sewer Load Allocator program.

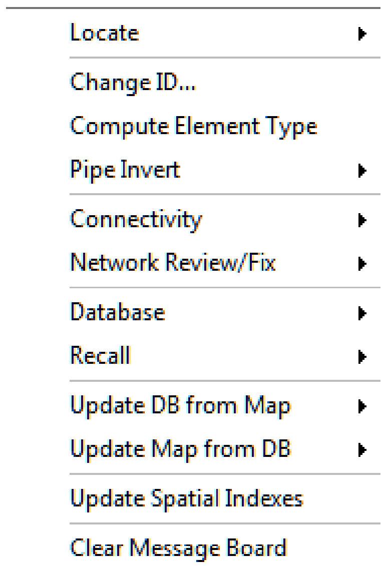
- **Preferences** - The Preferences section is where all project specific options are set.


InfoSewer
TRACE
Initialize Project
Project
Edit
View
Exchange
Scenario
Tools
Utilities
Help
Run Manager...
Batch Simulation...
Pipe Profile for Output...
Query Report Manager...
Output Report Manager...
Output Unit Manager...
Domain Manager...
Facility Manager...
Animation Editor...
Animation Viewer...
Pipe Invert Calculator...
Pipe Invert/Direction Translator...
Pipe Direction Calculator...
Pipeline Costing...
Elevation Extractor...
Non-Circular Cross-Section Design...
Query Summation Report...
Apply New Design Data...
AddOn Extension Manager...
Project Preferences...

us/support-overview

Utilities

The Utilities menu provides various advanced editing tools and operating tools such as the **Locate** command to locate your elements graphically on your Map. Use the **Insert Node, Move Node, Merge Node and Switch Node** commands to move and modify your data elements graphically. Create **Pipe Jumps**, manage the InfoSewer **Database**. Additionally verify **Connectivity** issues and **Recall** deleted elements. Another very useful command available from the Utilities menu is the **Change ID** command that allows you to change InfoSewer element IDs.



- **Locate** - The Locate command is used to find a node/pipe/element by ID within an InfoSewer project. This function is also available from the Model Explorer - Attribute Tab toolbar . See: Locate
- **Change ID** - The Change ID command allows the user to change theInfoSewer database identification for any element from one value to another (as long as the new value for that element is unique). Merely select the Desired Element Type to change, type in the Old ID then the New ID. Clicking Apply will change the ID for that element. Click here to learn more.

- **Pipe Invert** - To verify the invert order of gravity mains, go to the **InfoSewer** button, **Utilities** menu and select **Pipe Invert Order**. Select any of the commands, and then the user can either select pipes individually or drag a window across all desired elements. Once the pipes have been selected, right mouse click and choose the Enter option or hit the Enter key on your keyboard.

- **Connectivity** - The connectivity feature has many elements that assist the user in ensuring that connectivity is established prior to a model being run. It is important to note that connectivity is required for InfoSewer to run a hydraulic simulation. Click here to learn more.

- **Network Review/Fix** - The Network Review/Fix Tool is a comprehensive network drawing examination and correction application for use in constructing reliable, credible working models ready for analysis. It offers users functionality to quickly identify and correct network topology problems and data flaws that may arise from digitizing a model or building it using pre-existing GIS and CAD datasets. See: Network Review/Fix Tool

- **Database** - The database feature allows the user to utilize database management from within InfoSewer to find and correct database flaws or problems. Click here to learn more.

- **Recall** - The recall command (or Undelete) allows the user to recall a deleted pipe or node from the project database. It is important to note that when a record is deleted from a database, it is only "marked" for deletion (unless the user has turned on the **Auto Database Packing** feature in the **Project Preferences)**. Click here to learn more.

- **Update DB from Map** - This feature allows the user to recreate some or all of the project databases from the graphics in the InfoSewer project from the project databases. Click here to learn more.

- **Update Map from DB** - This feature allows the user to regenerate the map graphics from the project database to ensure that the map view and the database contain the same data. Click here to learn more.

- **Update Spatial Indexes** - The Update Spatial Index command will help to speed up the map refresh in ArcMap. When there are tables joined to an InfoSewer layer, ArcMap will not update the spatial indexes when those tables have been modified. The Update Spatial Indexes command will update the joined tables for an improved map refresh rate.

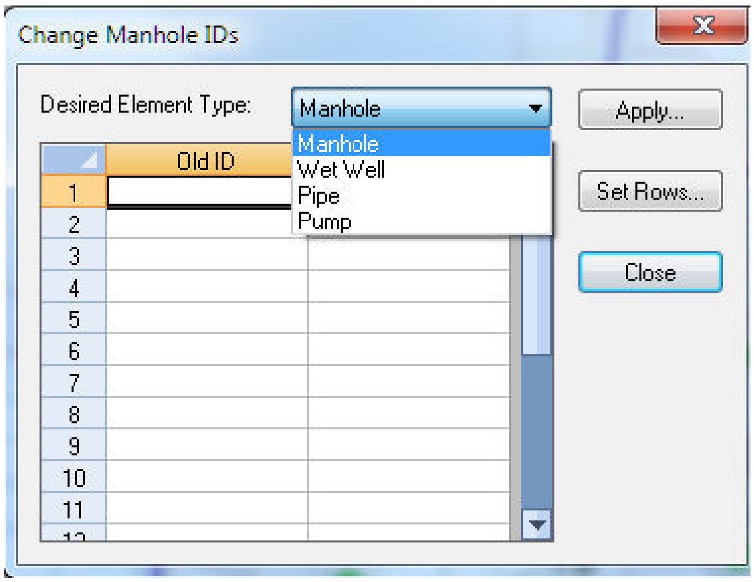
- **Clear Message Board** - Use this command to clear the InfoSewer Message Board. The InfoSewer message board displays messages, warnings and/or errors during, before and after an InfoSewer simulation.

Other Related Topics - Edit, Exchange, Help, Project, Scenario, Tools, Utilities

---

Change ID

The Change ID dialog box allows the user to change the ID for any selected data element. Click on any portion to learn more.

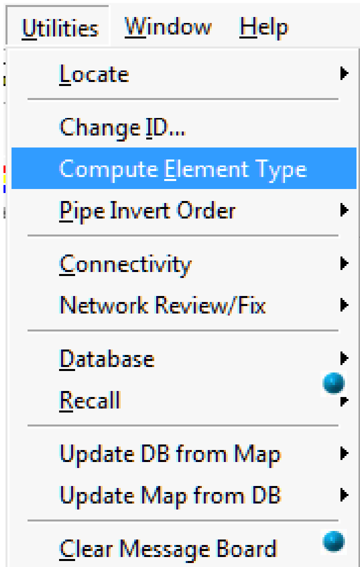


Desired Element Type - From the drop down box, select the InfoSewer data element for which an ID change is to occur.

- **Old ID** - Enter the current ID.
- **New ID** - Enter the new ID.
- **Apply** - Click apply to make the ID change.
- **Set Rows** - Specify the number of rows for mass editing (twenty is the default).
- **Close** - Close the current dialog box.

Note - To perform a mass edit, use the DB Editor and open the database where the subject ID's are stored. Highlight and copy the ID's into the Windows clipboard and paste the values into a third-party software like Microsoft Excel. Next to each ID, enter the new value for the ID - Using Excel functions like "mid" and "concatenate" to help you in the mass edit. Once you have the old and new ID's, determine how many rows are being used in Excel. Highlight all old and new ID's in Excel and use Ctrl+C to copy the highlighted area. Go back to InfoSewer, use the Set Rows command to make the rows the same as those in the Windows clipboard. Once this is done, highlight the first cell in the Change ID dialog box and use the Ctrl+V function to paste the values from the clipboard. You have now greatly reduced your time from having to edit each ID individually. Click Apply to change the ID's and then Close to close the dialog box.

Compute Element Type

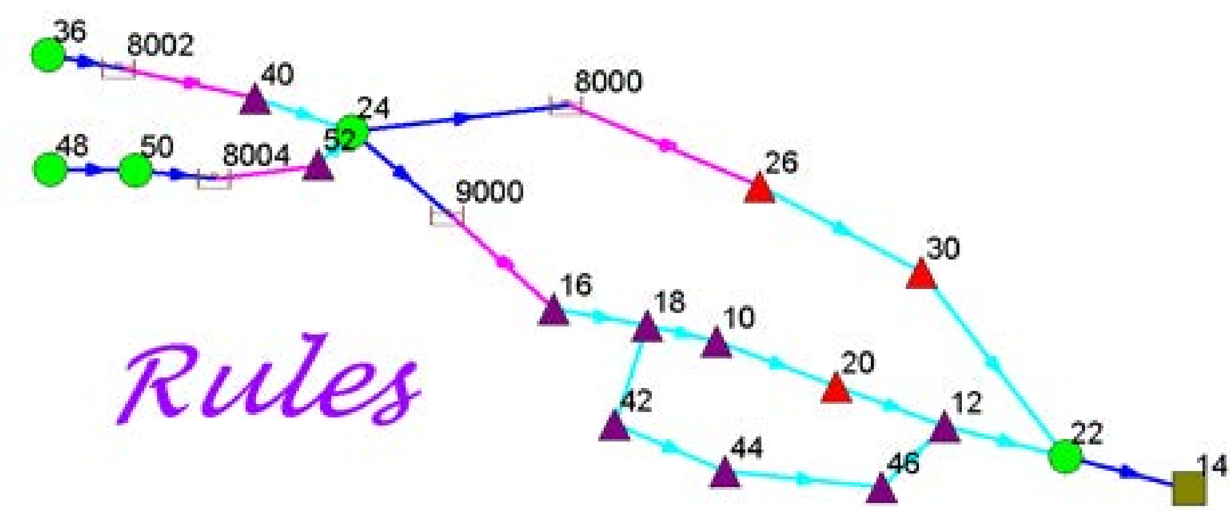


If the user should incorrectly create an element or import a set of elements that do not adhere to sewer collection system requirements, this command will allow the program to automatically determine which element should be inserted in lieu of the existing element.

For example:

If the most downstream node is digitized as a manhole, this command will convert that element to an outlet.

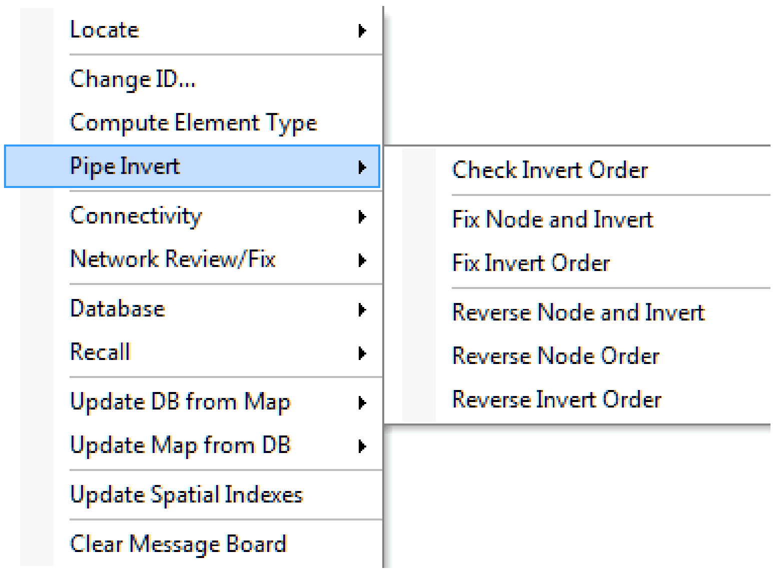
If a pipe is imported as the downstream element from a wet well, this command will convert that pipe to a pump. Likewise, it will change the next node to a chamber node and the next link to a force main.



Pipe Invert Order

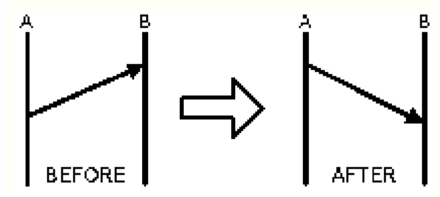
To verify the invert order of gravity mains, go to the **InfoSewer** button, **Utilities** menu and select **Pipe Invert Order**. Select any of the commands, and then the user can either select pipes individually or drag a window across all desired elements. Once the pipes have been selected, right mouse click and choose the Enter option or hit the Enter key on your keyboard.

Click anywhere on the dialog box to view more information.

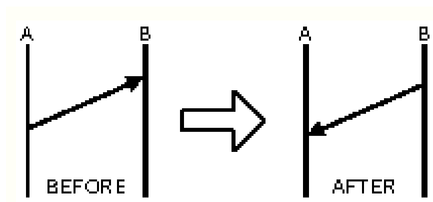


Pipe Invert Order Options

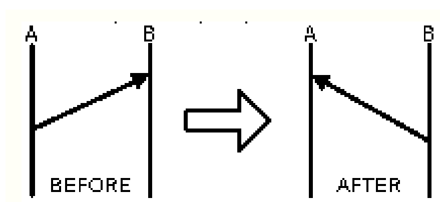
- **Check Invert Order** - For every pipe that has been found to have invert elevations in the opposite direction of flow (adverse slopes - upstream elevation lower than the downstream), a statement will appear in the Message Window located at the bottom of the display. The user can either locate and edit the slopes for each of these pipes or use either of the following two commands:
- **Fix Node and Invert** - This command will reverse the direction of the pipe to correlate with the upstream and downstream invert elevations. If a pipe has been found to be adverse, this command will reverse the direction of pipe to match the FROM INVERT and TO INVERT fields.



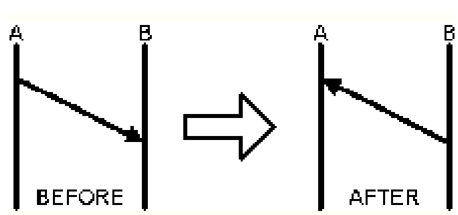
- **Fix Invert Order** - This command is used to have the upstream and downstream fields switched for all pipes that have been found to have adverse slopes. The message box will also inform the user which pipes the inverts were switched. This command will only work on adverse slope pipes.



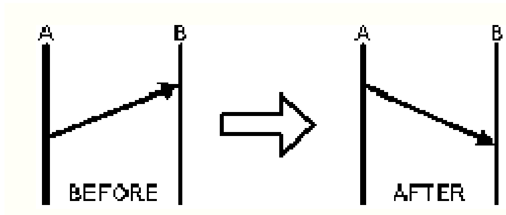
- **Reverse Node and Invert** - This command will reverse both the direction of the pipe and the invert elevations between the two nodes. Once the command is completed, the link will be graphically reversed and the upstream and downstream invert elevations will also be switched. This command will only work on adverse slope pipes.



- **Reverse Node Order** - This command will reverse the direction of the pipe between the two nodes. The link will be graphically reversed, but the upstream and downstream inverts will remain the same. This command will work on any pipe.



- **Reverse Invert Order** - This command will reverse the inverts between two nodes. The graphic for the link will remain the same, only the FROM INVERT and the TO INVERT elevations will be switched. This command will work on any pipe.

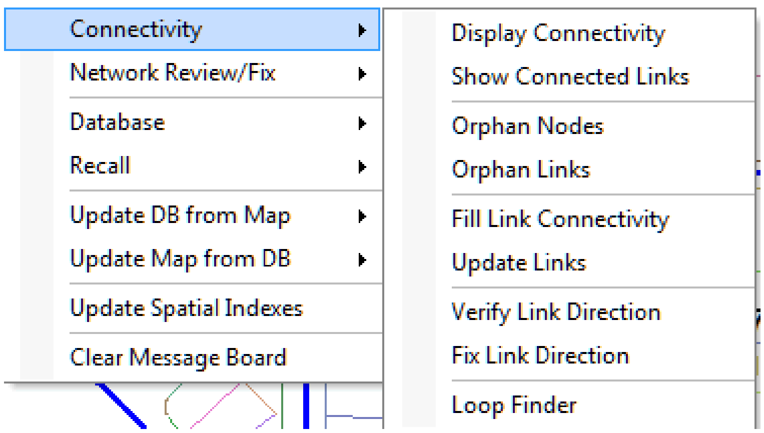


2

Connectivity Menu

The Connectivity sub-menu is used to verify network connectivity before running a simulation. When importing a model network from an external data source (GIS, infrastructure inventory, other hydraulic model, etc.), it is critical that the network representation be properly formed (each pipe is connected to exactly two nodes, each node is connected to at least one pipe). Otherwise, errors will be reported when attempting to run a simulation.

To verify network connectivity, go to the **InfoSewer Control Center -> InfoSewer** button **-> Utilities** menu and select **Connectivity**. There the user will see the following options:



- **Display Connectivity** - Draws a simple straight-line to graphically represent the connectivity for the selected pipe(s). The connectivity line drawn represents to FROM and TO nodes in the database. This allows the user to graphically confirm that the database matches the graphics. If it appears that a match does not exist, use either the DB from Map or the Map from DB commands to rectify the discrepancy.
- **Show Connected Pipes** - Highlights pipes connected to the selected node.
- **Orphan Nodes** - Lists the ID for any orphan nodes. An orphan node is defined as a node not connected to any links in the current InfoSewer model.

- **Orphan Pipes** - Lists the ID for any orphan pipes. An orphan pipe is defined as a pipe without either a FROM or TO node, or more specifically, a pipe whose from and to-node ID's do not reference existing nodes in the current InfoSewer model.
- **Fill Link Connectivity** - Based on a user defined tolerance and subsequent parameters, fills the connectivity information for all InfoSewer pipes based on the graphic overlap of the selected pipes. This tool is used to "weed out" very small pipes that connect two nodes that are very close together. After running this command, use the Orphan Nodes and Orphan Pipes commands.
- **Update Links** - Updates the connectivity for the selected pipe(s) by connecting that pipe to the correct nodes as indicated by the FROM and TO node IDs assigned to that pipe.
- **Verify/Fix Link Direction** - Checks/fixes the graphic direction of the selected pipes. Any differences will result in having the graphic direction automatically reversed to match the connectivity as listed in the InfoSewer databases.
- **Loop Finder** - This command will search the entire network and locate potential cyclic loops (clockwise or counter-clockwise flow circulation). These are areas in the system where flow is being routed back to a manhole of original. Since circular flow is not permissible in a sewer system, a hydraulic model with such an occurrence will not converge.

Note - If pipe-node connectivity is determined to be incorrect using one of the Connectivity command options, it is the responsibility of the user to determine which representation is correct - the graphic representation appearing on the map display or the FROM and TO node ID designations as stored in the database. You may use any InfoSewer data modification command to correct pipe-node connectivity.

If the database representation is correct (i.e., the from- and to-node IDs are correct, but the graphic is drawn incorrectly on the network map display), use the DB to Map command. Likewise, if the graphic is correct and the database is not, use the Map to DB command.

Network Review/Fix Tool

The Network Review/Fix Tool is a comprehensive network drawing examination and correction application for use in constructing reliable, credible working models ready for analysis. It offers users complete functionality to quickly identify and automatically correct any network topology problems (e.g., disconnected nodes) and data flaws (e.g., duplicated pipes or nodes) that may arise from digitizing a model or building it using pre-existing GIS and CAD datasets.

Locate/Fix Nodes in Close Proximity (Overlapping/Duplicate Nodes)

Nodes in close proximity designate nodes that overlap (accidentally duplicated) as shown in the figure below. The Network Review/Fix Tool allows the user to instantly view every node in close vicinity of another node (based on any specified distance) and automatically merge the identified nodes into one where necessary.

Locate/Fix Pipe-Split Candidates

Pipe-split candidates represent separate pipe sections that should be connected by a common node as depicted in the figure below. The Network Review/Fix Tool allows the user to rapidly locate all pipe-split candidates in the network that are within a specified distance of their end nodes and automatically make the connection where appropriate.

Locate/Fix Crossing/Intersecting Pipes

Crossing pipes refer to those pipes that cross but do not intersect at a common node as shown in the figure below.

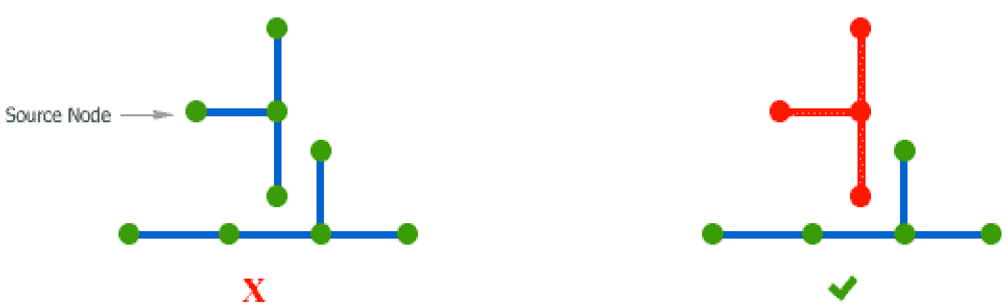
The Network Review/Fix Tool allows the user to rapidly view all crossing pipe candidates in the network and automatically create the intersection where necessary.

Trace Connected Nodes

This command allows the users to trace all connected nodes.

Trace Network

The Network Trace function launches a spanning tree to identify all pipes and nodes in the network that can be reached from any specified location (source node) via a connected path. This function allows the user to instantly view all portions of the network model that are disconnected from any node as shown in the figure below. The user can then easily determine if a hydraulic connection actually exists and, if so, make the connection in the model.



Trace Upstream Network

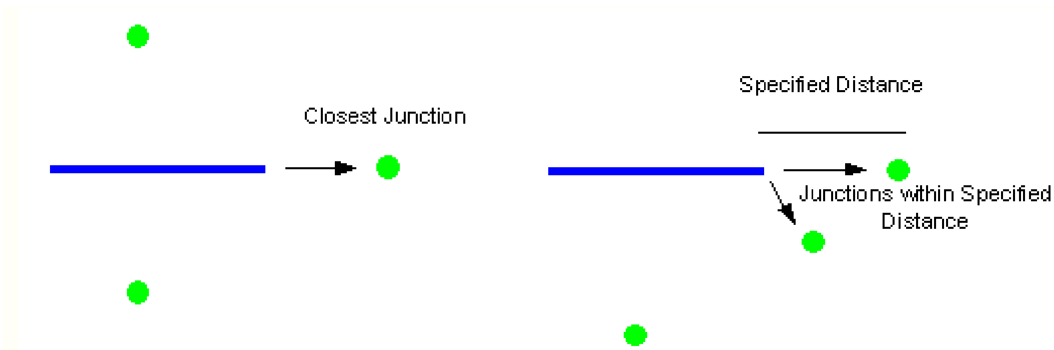
This command allows the user to trace all connected networks upstream of a selected node.

Trace Downstream Network

This command allows the user to trace all connected networks downstream of a selected node.

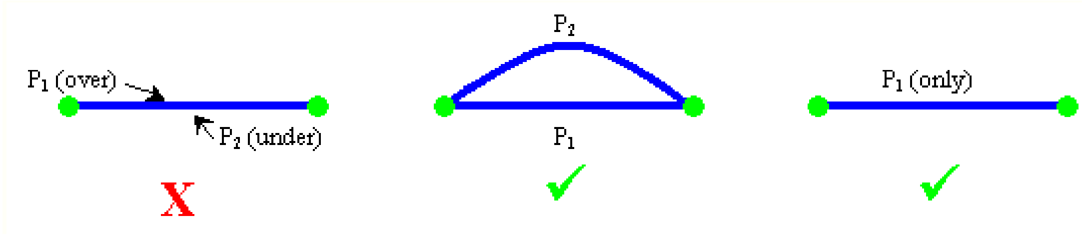
Locate Manhole Closest to Pipe

Manhole(s) closest to a pipe may be located using this feature. This is especially useful when data is imported from a GIS into InfoSewer/Pro and may be used to identify the closest manhole(s) to the pipe. Additionally you may also use this functionality to specify a distance from the pipe in map units and locate all the junctions within this specified distance.



Locate Parallel Pipes

Duplicate pipes are superimposed/parallel pipes that share the same curvilinear shape. The Network Review/Fix Tool is able to rapidly identify all duplicate pipe candidates in the network model. The user can then determine if a parallel pipe actually exists and, if so, re-route (redraw) the pipe in a noticeable manner or remove/delete it from the model if such a duplicate pipe does not actually exist as shown in the figure below.



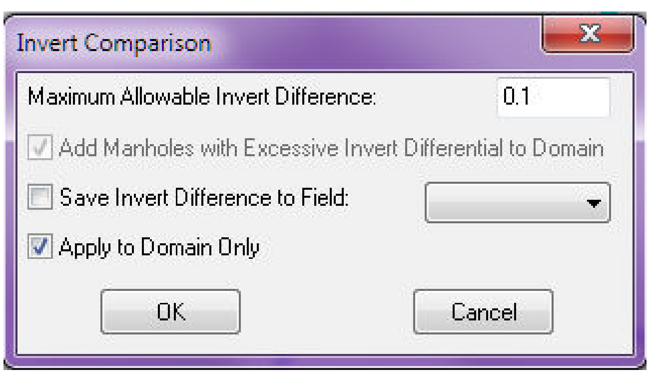
Join Disconnected Nodes

Disconnected nodes are stray nodes (i.e., are not connected to any pipe) that are separated from the rest of the network as shown in the figure below.

The Network Review/Fix Tool allows the user to quickly locate and highlight all junction nodes in the network that are not connected to any pipe. The user can then choose to connect those junctions to the system or remove them.

Invert Comparison

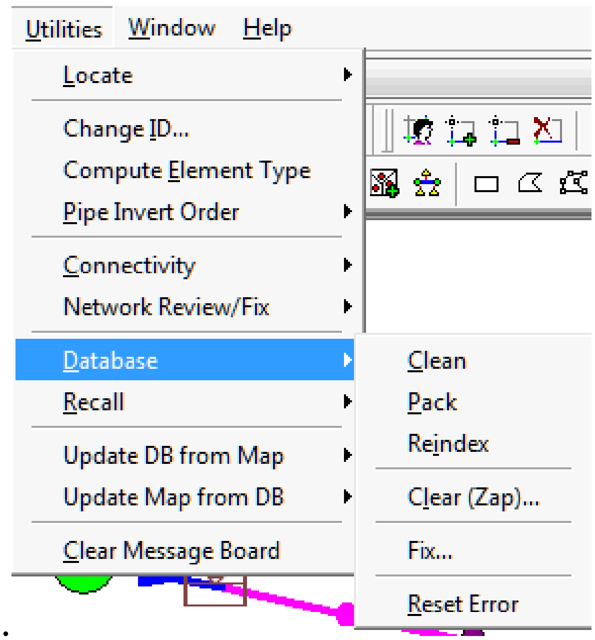
Flat slope and adverse/negative slope pipes could be quickly identified using this feature. Starting and destination invert elevations of a pipe are compared, and the routine locates elements whose differences in these elevations are less than the allowable tolerance. The comparison could be performed for all pipes in the network, or could be limited to pipes in a domain.



Database Tools Menu

The database tools submenu is provided to assist the user in database management. Remembering the InfoSewer is based on having graphical links to external databases, the data contained in those databases may, at times, require maintenance due to forces beyond the control of the user.

To run database management, from the **InfoSewer -> Utilities** menu, select **Database**. There you may select from the following options.



- **Clean** - The Clean Database command removes database records without a matching network component in the network map display. While it is unlikely this may occur, there are instances where a corrupt condition may occur do to inadvertent computer reboot or power outage.
- **Pack** - When data elements are graphically deleted, the database records associated with each component are marked for deletion and must be packed for those records to be permanently removed. When Pack is chosen, select the table or tables to pack, then select OK.
- **Clear (Zap)** - The Clear (ZAP) command reinitializes InfoSewer non-component database tables (demands, patterns, initial status, operational controls, water quality sources and parameters, and simulation options).

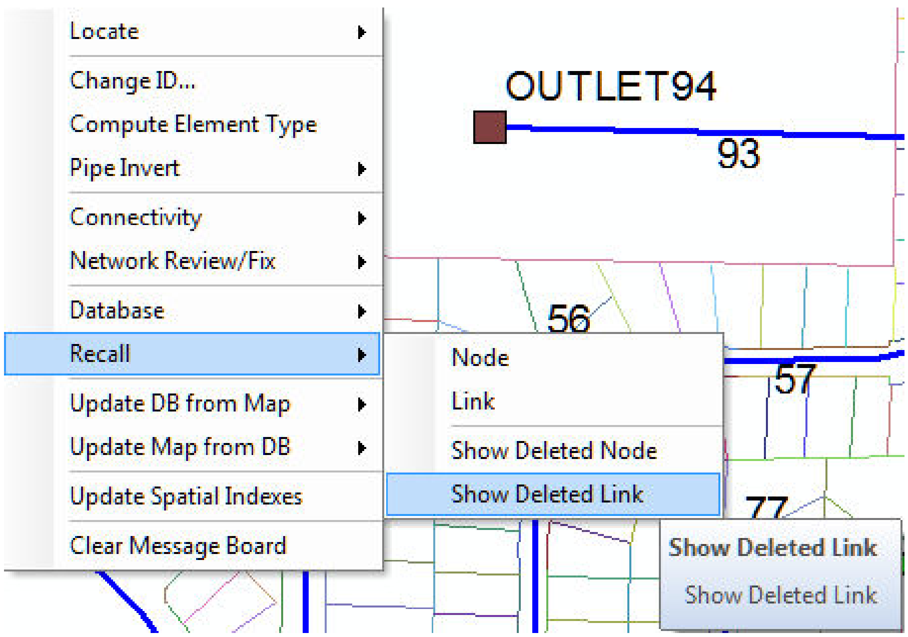
The user picks the database table to clear from an on-screen pop-up window of tables. All records are removed from the selected tables.

- **Reindex** -The Reindex Database command reconstructs internal database table indexes. This command is recommended when you use applications external to InfoSewer (such as custom GIS or relational database programs) to populate and/or maintain InfoSewer database tables.
- **Fix DB** - The Fix DB function allows the user to select and repair a corrupted database table.
- **Reset Error** - The Reset Database command resets the internal InfoSewer error flag. If an unexpected error occurs during an InfoSewer database operation, an error flag will be set preventing the user to further modify, and potentially corrupt, the project database. The Reset Database command clears the flag and allows the user to continue modifying the InfoSewer project.
- **Join All Layer Tables** - Joins all database tables to their respective feature classes. This function enables the InfoSewer database tables to be accessed with ArcGIS tools. For more information on Joins, please refer to the ArcGIS help file.
- **Remove All Layer Joins** - Removes all database tables to their respective feature classes. For more information on Joins, please refer to the ArcGIS helpfile.

Recall

The Recall command is used to restore – or undelete – network components that have been deleted. This command will not work if the user has enabled Auto Database Packing from **InfoSewer Tools -> Project Preferences -> Operation Settings** tab or has recently packed the database. Recalled components will be redrawn with their graphical properties (node size, pipe connectivity and shape) at the time of deletion. All database values assigned to those components will be restored.

To run a Recall, from the **InfoSewer -> Utilities** menu, select **Recall.** Click on any portion to learn more.



Recall Options

- **Node** - Recalls one or more previously deleted nodes. If the user does not know the ID of the deleted node, use the Show Deleted Nodes command to display a listing of deleted nodes.
- **Link** - Recalls one or more previously deleted links. If the user does not know the ID of the deleted link, use the Show Deleted Links command to display a listing of deleted links.
- **Show Deleted Nodes** - Used to get a listing of all nodes marked for deleting in the nodes database.

- **Show Deleted Links** - Used to get a listing of all links marked for deleting in the links database.

Update DB to MAP

The DB to MAP command is used to redraw the current **Sewer** project's network map based on the following values in the **Sewer** project database:

- X,Y Coordinates stored with manholes and wet wells.
- Connectivity (from and to node designations) and shape (intermediate X,Y shape-defining vertices) for pipes, pumps, and valves. The length value in the model input database is not used to update the network map.

The Update MAP from Database command should be used to update the **Sewer** project map in the following situations:

- When records from a non- **Sewer** database (GIS, facility inventory databases, etc.) are imported into **Sewer** using the delimited text (CSV) option. Importing comma-delimited text files updates existing records and/or adds new records to the database for the current **Sewer** project but does not update the corresponding elements on the map display.

To run an update, from the **Utilities** menu, select **Update DB to MAP**. There the user will see the following options:

DB to MAP Options

Single Node - Updates a single manhole or wet well based on a record in the associated database table.

Single Link - Updates a single link (gravity, force or pump) based on a record in the associated database table.

All Manholes - Updates all manholes based on records in the manholes database table.

All Wet Wells - Updates all wet well nodes based on records in the wet well database table.

All Pipes - Updates all pipes based on records in the pipes database table.

All Pumps - Updates all pumps based on records in the pumps database table.

All Network - Updates the entire network map based on records in the database tables.

Force All Network - Same as the All Network option, but with this option, all drawing elements are removed from display and redrawn.

If You Have Developed Custom Scenarios

If you have developed custom scenarios in the current **Sewer** project or alternately, are taking advantage of **Sewer**'s facility set activation feature, you should be aware of the following:

- The Update MAP from Database command updates all network components, not just those activated as part of the current scenario. Changes to inactive components will be shown when those components are activated as part of another custom scenario.
- The information required by the Update MAP from Database command are stored in geometry tables stored external to scenario data sets. Therefore the command disregards custom scenario definitions and related data.

Update Map to DB

The MAP to DB command is used to update the current project's database tables based on the current state of the **Sewer** network drawing. This command should be used in the following situations:

- If an ESRI generate file containing a different configuration than the original network is imported into the active **Sewer** project, replacing existing components in the network drawing.
- If a database record appears to be corrupted or has erroneous data that differs from the H2OMAP Sewer project.

Upon terminating the selection process, **Sewer** updates the database records corresponding to the selected components. The Update Database from MAP command updates the following in the **Sewer** project database:

- Nodes X,Y coordinate location.
- Links Connectivity (from and to node designation), shape (intermediate X,Y shape-defining vertices), and length for pipes. To have **Sewer** recalculate link lengths, be sure that the Auto-Length Calculation preference is set to ON prior to running the Update Database from ACAD command.

To run an update, from the **Utilities** menu, select **Update MAP to DB**. There the user will see the following options:

MAP to DB Options

All Nodes - Updates the **Sewer** database records for all manholes and wet wells.

Nodes in Selection - Updates the **Sewer**database records for selected manholes and/or wet wells.

Nodes in Domain - Updates the **Sewer** database records for selected manholes and/or wet wells that are in the current domain.

All Links - Updates the **Sewer** database records for all pipes and pumps.

Links in Selection - Updates the **Sewer** database records for selected links (pipes or pumps).

Links in Domain - Updates the **Sewer** database records for selected links (pipes or pumps) that are in the current domain.

If You Have Developed Custom Scenarios

If you have developed custom scenarios in the current **Sewer** project or alternately, are taking advantage of **Sewer**'s facility set activation feature, you should be aware of the following:

- You can only select network components from the active facility set when updating the **Sewer** database tables from the network drawing.
- **Sewer** considers pipe length a modeling attribute and therefore stores pipe length in scenario pipe sets. The Update Database from MAP command updates the records in the active pipe set only. Records corresponding to the selected pipes in other (inactive) user-defined pipe sets will not be updated with this command.

Update Map to DB (Database)

The update Map to Database command is used to update the current project's database tables based on the current state of the InfoSewer network drawing. This command should be used in the following situations:

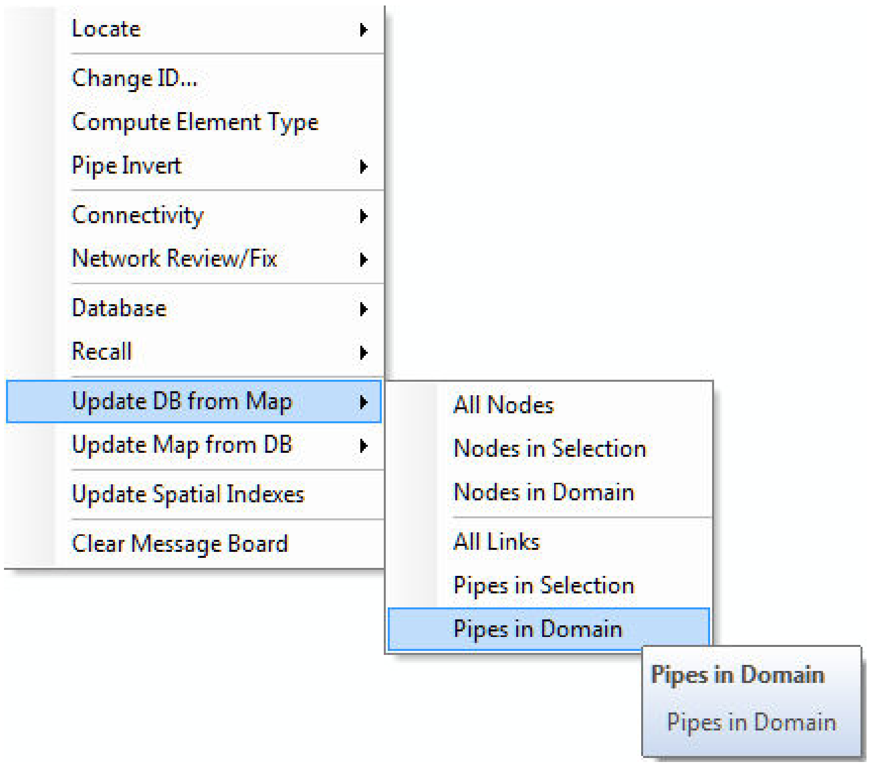
- If an ESRI generate file containing a different configuration than the original network is imported into the active InfoSewer project, replacing existing components in the network drawing.
- If a database record appears to be corrupted or has erroneous data that differs from the InfoSewer project.

Upon terminating the selection process, InfoSewer updates the database records corresponding to the selected components. The Update MAP to Database command updates the following in the InfoSewer project database:

- Nodes X,Y coordinate location.
- Pipes Connectivity (from and to node designation), shape (intermediate X,Y shape-defining vertices), and length for pipes. To have InfoSewer recalculate pipe lengths, be sure that the Auto-Length Calculation preference is set to ON prior to running the Update Database from ACAD command.

To run an update, from theInfoSewer-> **Utilities** menu, select **Update Map to DB**.

Click on any section below to learn more:



- **Note -** If you have developed custom scenarios in the current InfoSewer project or alternately, are taking advantage of InfoSewer's facility set activation feature, you should be aware of the following:
  - You can only select network components from the active facility set when updating the InfoSewer database tables from the network drawing.

InfoSewer considers pipe length a modeling attribute and therefore stores pipe length in scenario pipe sets. The update DB from Map command updates the records in the active pipe set only. Records corresponding to the selected pipes in other (inactive) user-defined pipe sets will not be updated with this command.

- **All Nodes** - Updates theInfoSewer database records for all nodes including junctions, tanks, reservoirs, pumps and valves.
- **Nodes in Selection** - Updates the InfoSewer database records for selected nodes such as junctions, tanks, reservoirs, pumps and valves.
- **Nodes in Domain** - Updates the InfoSewer database records for selected nodes such as junctions, tanks, reservoirs, pumps and

valves that are included in the currently active domain.

- **All Pipes** - Updates the InfoSewer database records for all pipes.
- **Pipes in Selection** - Updates the InfoSewer database records for selected pipes.
- **Pipes in Domain** - Updates theInfoSewer database records for selected pipes that are included in the currently active domain.

Note - If you have developed custom scenarios in the current InfoSewer project or alternately, are taking advantage of InfoSewer's facility set activation feature, you should be aware of the following:

- You can only select network components from the active facility set when updating theInfoSewer database tables from the network drawing.
- InfoSewer considers pipe length a modeling attribute and therefore stores pipe length in scenario pipe sets. The update Map to DB command updates the records in the active pipe set only. Records corresponding to the selected pipes in other (inactive) user-defined pipe sets will not be updated with this command.

Other Related Topics - Update DB to MAP

InfoSewer™

Modeling Made Easy

Element Attribute Browser

The Attribute Browser dialog box allows the user to view and edit the database information related to a selected link or node facility. Shown below, the dialog box contains all information related to links and nodes when selected by the user.

Unless "grayed" out, any of the records under the section headers (Geometry, Modeling, Information) are editable at any time. While using the Attribute Browser, click the red check box next to the title to expand and contract the section headers. The Output data section is available only after a successful model run. Additionally other features allow you to locate (search) elements, create default values and assign specific values through the Tools menu.

To learn more about the multitude of icons related to the Attribute Browser, click here or select any of the icons below.

The Link Element

The link tab allows the user to see the database information related to a graphically selected link.

Click on any of the drop down links below to learn more about the database information contained in the Attribute Browser.

Pipe Data

Pipe ID

- **ID** - The unique InfoSewer identification of the pipe.
- **Description** - A user defined description of the pipe.

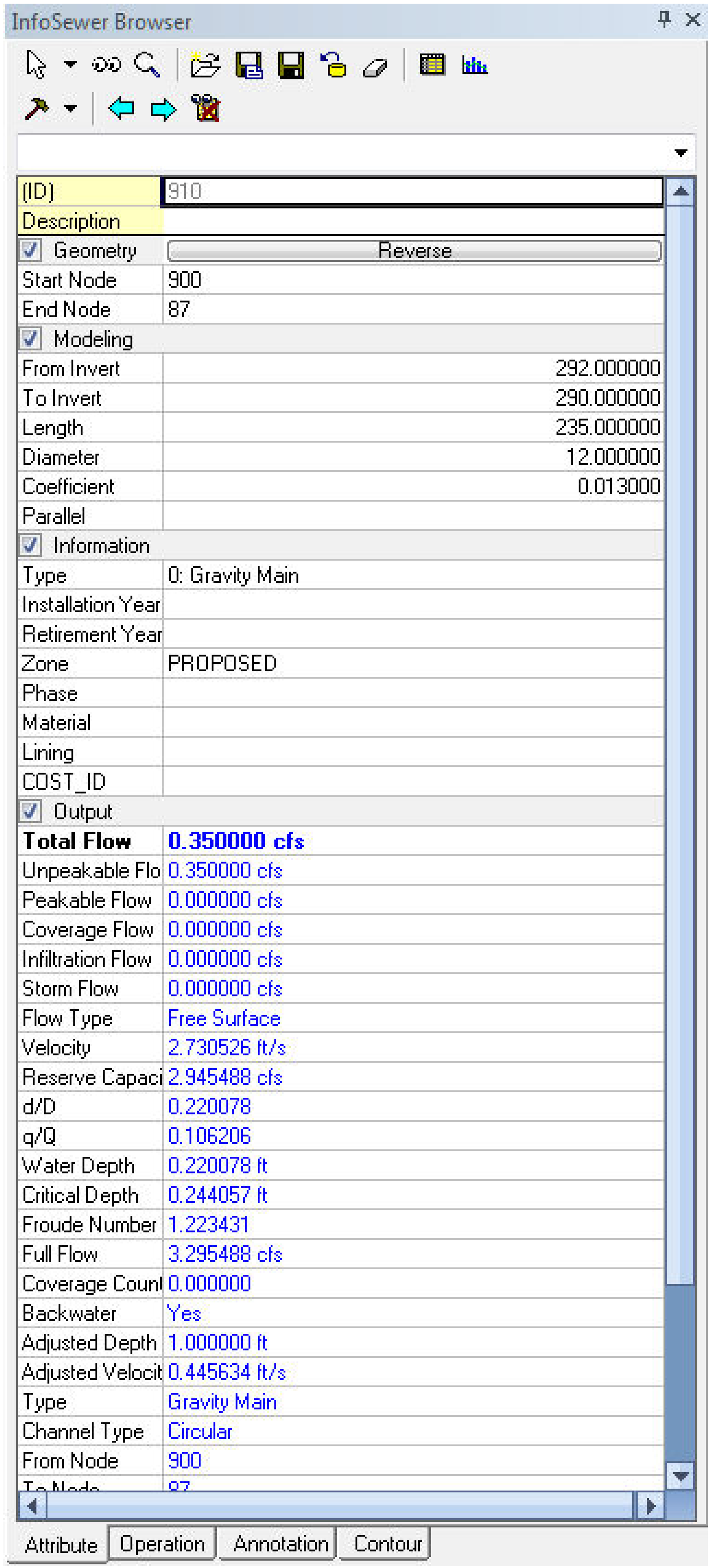
Geometry

- **Start Node** - The upstream node for the pipe.

End Node - The downstream node for the pipe.

Modeling

- **From Invert** - The upstream invert elevation of the pipe. Units are feet or meters.
- **To Invert** - The downstream invert elevation of the pipe. Units are feet or meters.
- **Length** - The length of the pipe. Created automatically if Tools -> Preferences ->



Operation -> Auto Length Calculation is checked on. Units are feet or meters.

- **Diameter** - The diameter of the channel (for circular pipes) as specified by the user. For non-circular conduits, required input parameters may be one or more of channel depth, channel width, and side slopes. Units are inches or millimeters.
- **Coefficient** - The Manning (gravity flow) or Hazen-Williams (force main) roughness coefficient as assigned by the user.
- **Parallel** - The number of pipes in parallel with the existing pipe. Each parallel pipe is given the exact same parameters as the existing pipe. Unitless.

Output

- **Output** - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a pipe output report.

Pump Data

Pump ID

- **ID** - The unique InfoSewer identification of the pump.

Description - A user defined description of the pump.

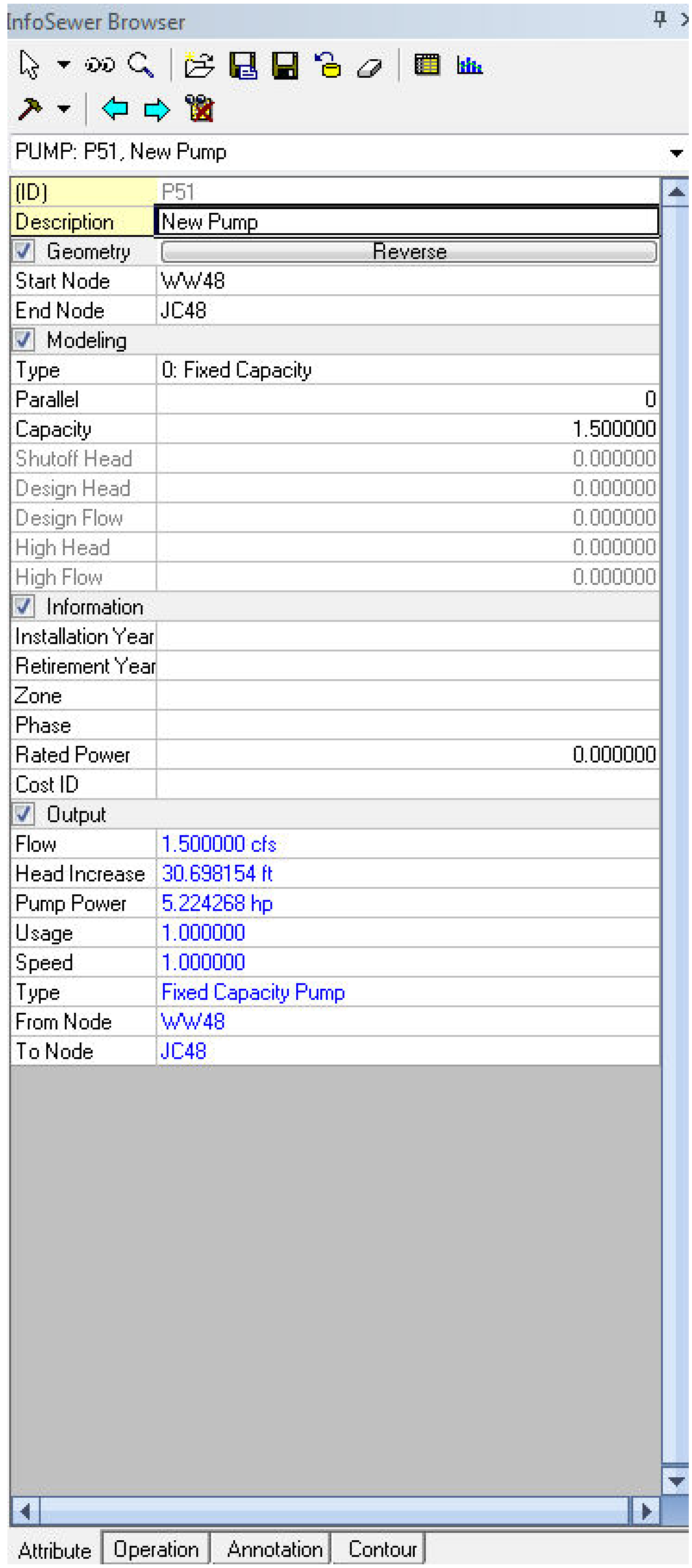
Geometry

- **Start Node** - The upstream node for the pump.

- **End Node** - The downstream node for the pump.

Modeling

- **Type** - The type of pump to be used by InfoSewer.. To change the type, merely click in the drop down box



and select from the list. Click here to learn more about each type and the fields required.

- **Parallel** - The number of pumps in parallel to the existing pump. Parallel pumps are given the exact specifications of the existing pump. Unitless.

Click here to learn more about the types of pumps available

Information

- **Installation Year** - The year the pump was installed.
- **Retirement Year** - The year the pump is expected to be retired from service.
- **Zone** - The service area of the selected pump.
- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to

modeling a simulation.

- **Rated Power** - The rated horsepower of the pump in the field.
- **Cost ID** - A database field used to assign differing cost identifiers to differing facilities.
- **(Other User Created Fields)** - Any fields added by the user will appear at the end of the Information tab. Click here to learn how to add a field.

Output

- **Output** - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a pump output report.

The Node Element

The node tab allows the user to see the database information related to a graphically selected node.

Manhole Data (Normal, Chamber, Outlet)

Node ID

ID - The unique H2OMAP Sewer identification of the manhole.

Description - A user defined description of the node.

Geometry

X - The x-coordinate of the node.

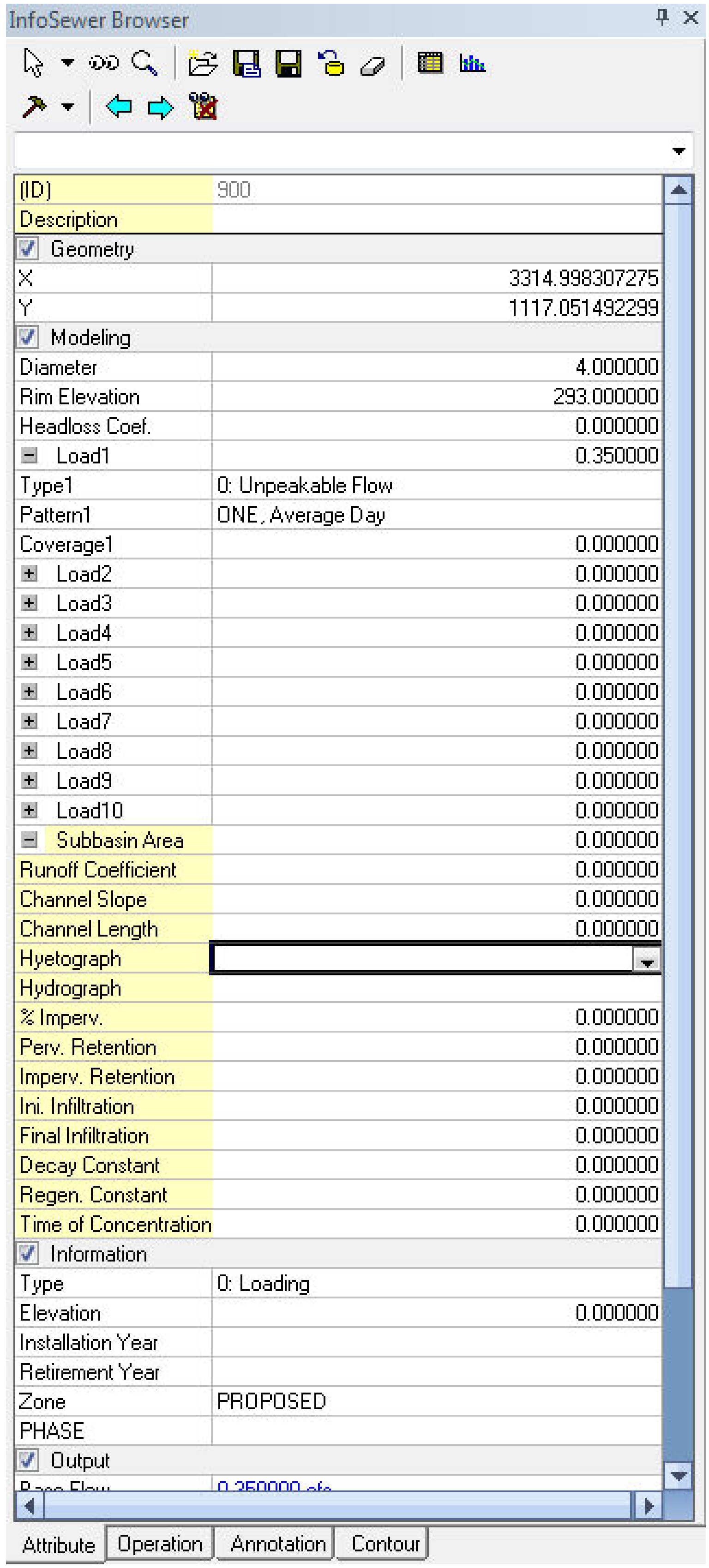
Y - The y-coordinate of the node.

Modeling

Diameter - The internal diameter of the manhole. Feet or Meters.

Rim Elevation - The elevation of the rim of the manhole as specified by the user (usually a value denoted above sea level). Units are feet or meters.

Headloss Coef – Used to calculate headloss in the



manhole itself using the exit pipe velocity (see link for more details on Headloss Coefficients and manhole headloss)

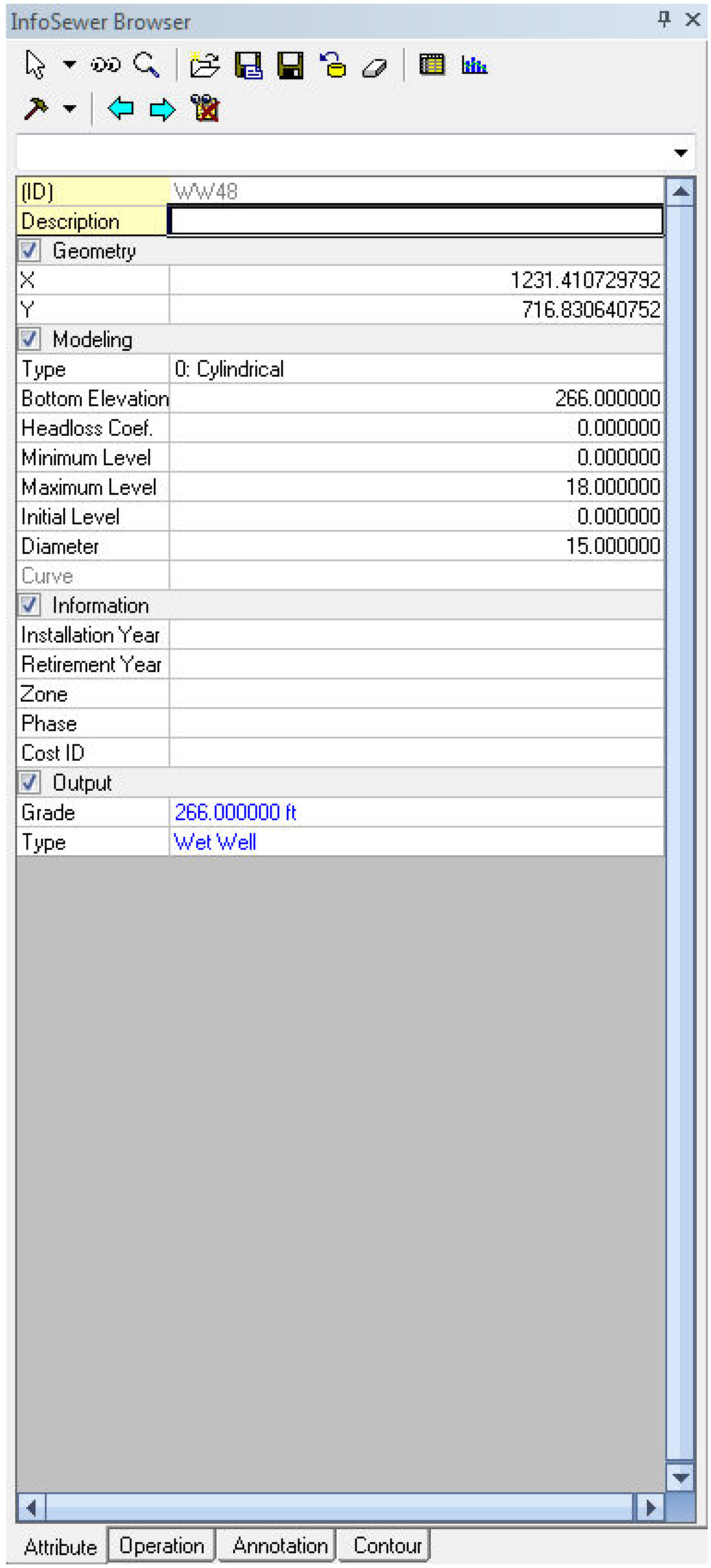
- **Load1** - The base sewage loading demand at the manhole. Up to ten base loadings can be entered for a single manhole. The extra load option allows additional loadings to be added as needed. Units from the Run Manager.

- **Type1** - Specify whether the load is an unpeakable flow, an peakable base flow or a peakable coverage flow. Click here to learn more about each type.

- **Pattern1** - The user defined diurnal pattern for the manhole loading. Required for an EPS analysis.

- **Coverage1** - The base population to be used during a peakable coverage flow determination.

- **Subbasin Area** - Area of the subwatershed that drains to the loading manhole. This variable is required for stormwater modeling.
- **Runoff Coefficient** - Ratio of storm runoff to rainfall volume. This variable is required during steady state simulation of stormwater.
- **Channel Slope** - Average slope of the longest flow channel for the subwatershed that drains to the loading manhole. This variable is required during steady state simulation of stormwater.
- **Channel Length** - Length of the longest flow path for the subbasin that drains to the manhole. This variable is required during steady state simulation of



stormwater. Units are feet or meters.

- **Hyetograph** - A plot of rainfall intensity versus time. This variable is required during dynamic simulation of stormwater.
- **Hydrograph** - Refers to a unit hydrograph pattern, and is required is required during dynamic simulation of stormwater.
- **% Impervious** : Percentage of the subwatershed that is covered with impervious land uses. This parameter is used to estimate depression storage losses.
- **Pervious Retention** : Refers to depression storage of the pervious portion of the subwatershed. The value should be given in feet for US Customary units, and in mm for SI units.

- **Impervious Retention** : Refers to depression storage of the impervious portion of the subwatershed. The value should be given in feet for US Customary units, and in mm for SI units.
- **Initial Infiltration** : maximum or initial infiltration rate (at t = 0). The parameter is accepted in inches/hour for US Customary, and in mm/hour for SI units. The default value is value is 3 inches/hour (76.2 mm/hour).
- **Final Infiltration** : final infiltration rate. The value should be provided in inches/hour for US Customary, and in mm/hour for SI units. The default value is 0.5 inches/hour.
- **Decay Constant** : decay coefficient for Horton's infiltration equation, in per

second. The default value is 0.001/second.

- **Regeneration Constant** : Decay coefficient for the exponential recovery curve, in per second. It is generally considered to be less than decay constant, implying a longer drying curve than wetting curve.

- **Time of Concentration:** Time of concentration for the subcatchment that is just starting to contribute to the node. Minutes.

Information

- **Type** - The type of manhole. Type 0 is a normal manhole, type 1 is a chamber manhole while type 2 is an outlet. Click here to learn more.

- **Elevation** - The ground elevation of the manhole. This value may differ than the rim elevation.

Units are feet or meters.

- **Installation Year** - The year the manhole was installed.
- **Retirement Year** - The year the manhole is expected to be retired from service.
- **Zone** - The service are of the selected manhole.
- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to modeling a simulation.
- **(Other User Created Fields)** - Any fields added by the user will appear at the end of the Information tab. Click here to learn how to add a field.

Output

Output - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a manhole node output report.

Wet Well Data

Wet Well ID

- **ID** - The unique InfoSewer identification of the wet well.

- **Description** - A user defined description of the wet well.

Geometry

- **X** - The x-coordinate of the wet well.

- **Y** - The y-coordinate of the wet well.

Modeling

Type - The type of wet well to be used by InfoSewer. To change the type, merely click in the drop down box and select from the list. Click here to learn more about each type and the fields required.

Information

- **Installation Year** - The year the wet well was installed.
- **Retirement Year** - The year the wet well is expected to be retired from service.
- **Zone** - The service area zone of the selected wet well.
- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to modeling a simulation.
- **Cost ID** - A database field used to assign differing cost identifiers to differing facilities.
- **(Other User Created Fields)** - Any fields added by the user will appear at the end of the Information tab. Click here to learn how to add a field.

Output

Output - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a wet well output report.

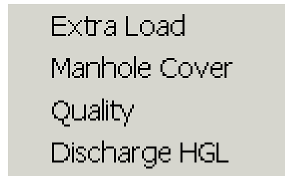
Other Related Topics - Tools menu

Tools Icon

Use this command to access certain advanced tools provided by InfoSewer to manipulate data related to the currently active element. Click on the Down Arrow icon  to launch the **Tools** menu. The tools menu changes depending on the type of element presently selected as shown below.

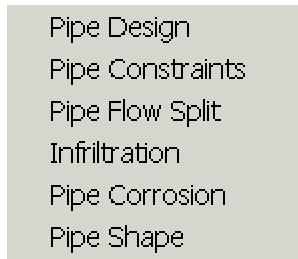
Click on any portion below to learn more:

Manhole

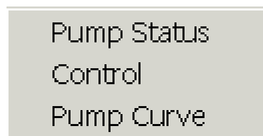


Wetwell

Pipe



Pump

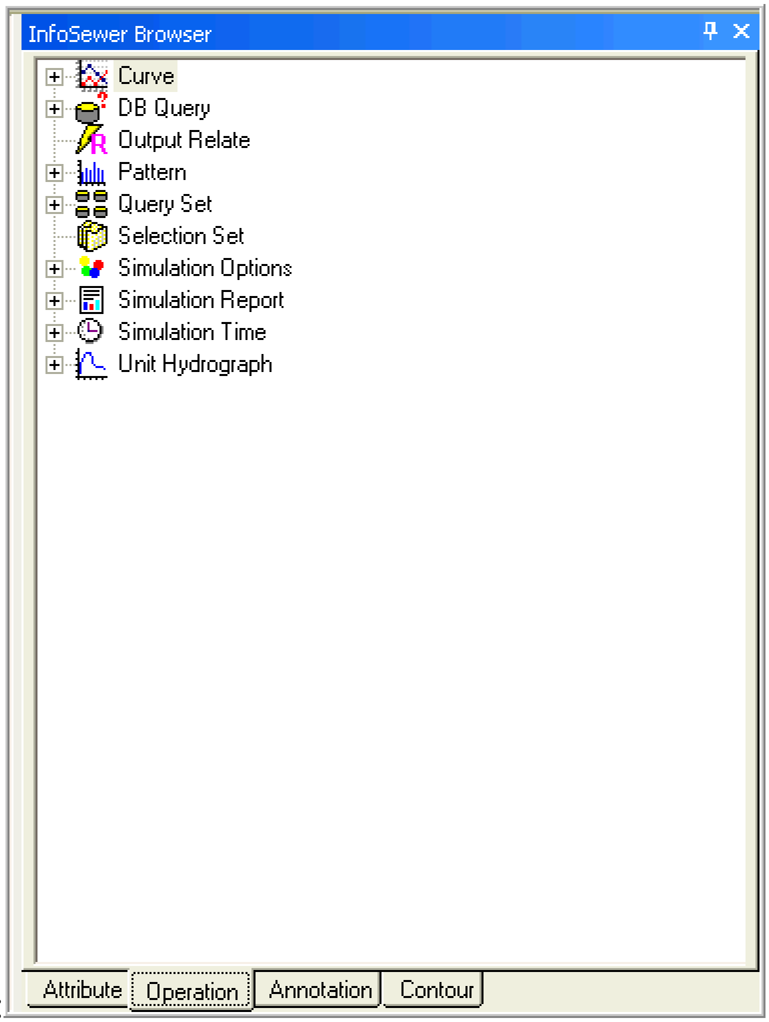


Other Related Topics - Attribute Browser

Operation Data

The Operation Tab section of the InfoSewer **Browser** provides you with tools to create, edit and operate on the different operational features of InfoSewer.

Click on any section below to learn more

:

Other Related Topics - Annotation Dialog Box, Annotation Methodology, Contour-Labelling, Contour-Level, Contour Dialog Box, Contour Methodology, Contour Options, InfoSewer Browser

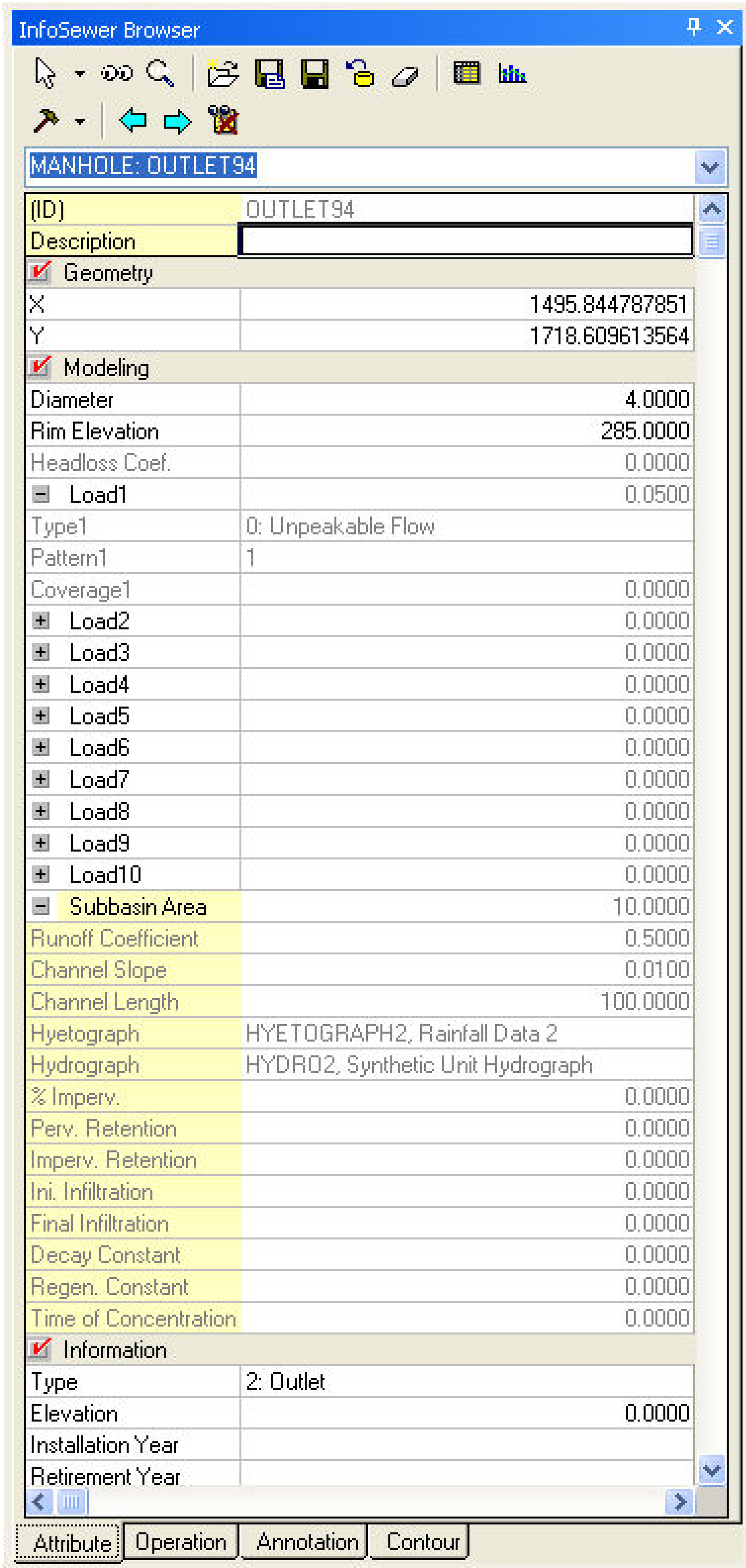
Browser Overview

The InfoSewer Browser provides a centralized location to access the various tools to create, edit, operate and analyze InfoSewer models. The Browser dialog box can be invoked by using the **Attribute Browser** command from the **InfoSewer> View** menu.

The InfoSewer Browser provides four major capabilities for the modeling and analysis of water distribution systems:

- **Attribute Browser:** The Attribute tab displays all of the element attribute tables for the selected data element. See Attribute Browser
- **Contour:** Use the contour tab to plot nodal input and output contours. To learn more about the different contouring options provided by InfoSewer click here.
- **Operation Data:** The Operation Data tab contains all of the operational features of InfoSewer. Click here to learn more about the Operation Data feature of InfoSewer.
- **Annotation:** The Annotation tab is used to color code pipes or nodes based on input and/or output results. Click here to learn more about the InfoSewer Annotation Manager.

Click on any section below to learn more:


InfoSewer Browser
MANHOLE: OUTLET94
(ID) OUTLET94
Description
Geometry
X 1495.844787851
Y 1718.609613564
Modeling
Diameter 4.0000
Rim Elevation 285.0000
Headloss Coef. 0.0000
Load1 0.0500
Type1 0: Unpeakable Flow
Pattern1 1
Coverage1 0.0000
Load2 0.0000
Load3 0.0000
Load4 0.0000
Load5 0.0000
Load6 0.0000
Load7 0.0000
Load8 0.0000
Load9 0.0000
Load10 0.0000
Subbasin Area 10.0000
Runoff Coefficient 0.5000
Channel Slope 0.0100
Channel Length 100.0000
Hyetograph HYETOGRAPH2, Rainfall Data 2
Hydrograph HYDRO2, Synthetic Unit Hydrograph
% Imperv. 0.0000
Perv. Retention 0.0000
Imperv. Retention 0.0000
Ini. Infiltration 0.0000
Final Infiltration 0.0000
Decay Constant 0.0000
Regen. Constant 0.0000
Time of Concentration 0.0000
Information
Type 2: Outlet
Elevation 0.0000
Installation Year
Retirement Year
Attribute
Operation
Annotation
Contour

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

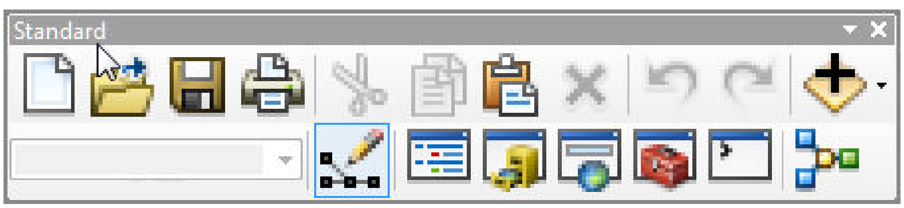
ArcMap Toolbars

InfoSewer is developed as a customized version of the ArcMap graphical user interface. Individual or logically grouped modeling and model data management functions are available as commands within ArcMap. The InfoSewer and the ArcMap toolbars are bundled together and may be accessed from the **View -> Toolbars** command. Additionally you may right click on any section of your ArcMap/InfoSewer user interface other than the Drawing Area to invoke any of the toolbars. Also you may use the Customize tool bar to choose the different tool bars that you want to invoke. The activated tool bars may be docked on any part of your graphical user interface.

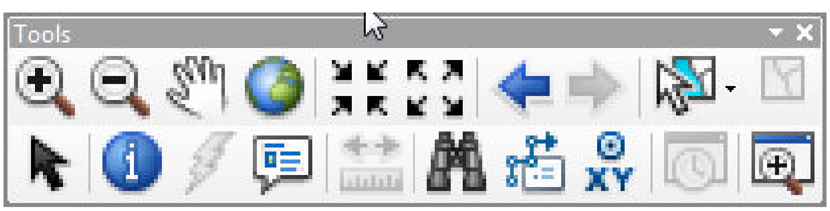
For more information on the ArcMap tool bars refer to your ArcMap help file. Detailed instructions on the usage and the capabilities of the InfoSewer toolbars are provided in the InfoSewer toolbars section.

The following are the different tool bars provided by ArcMap.

Standard

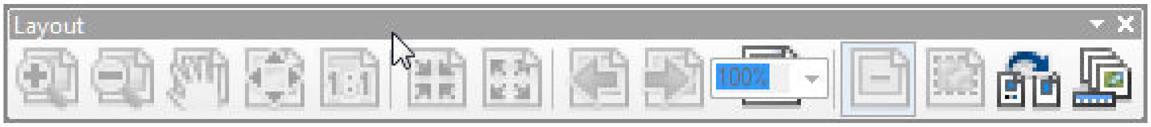


Tools

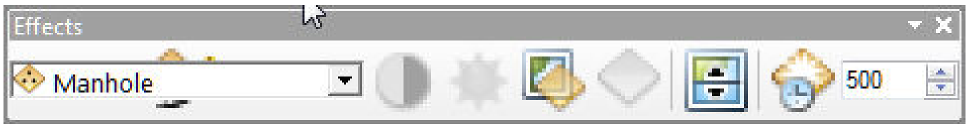


Draw

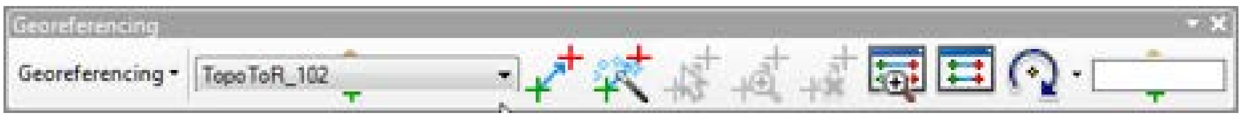
Layout



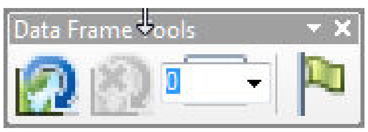
Effects



Georeferencing

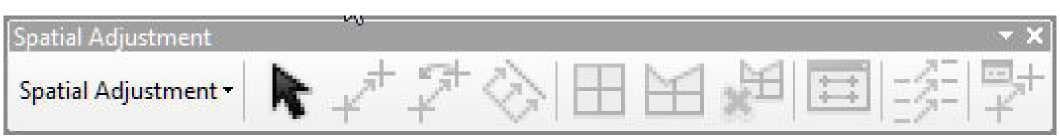


Data Frame Tools

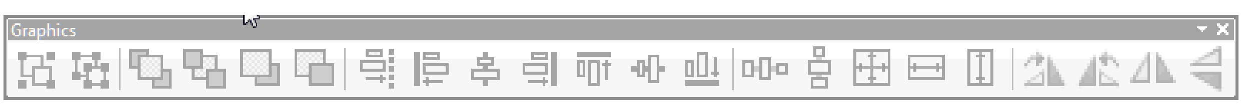


Utility Network Analyst

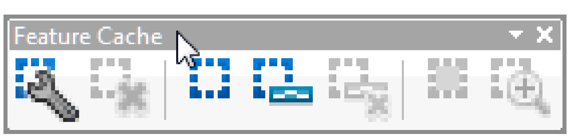
Spatial Adjustment



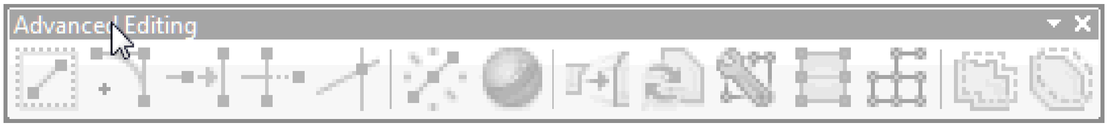
Graphics



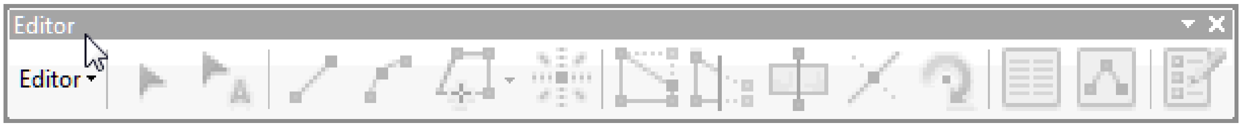
Edit Cache



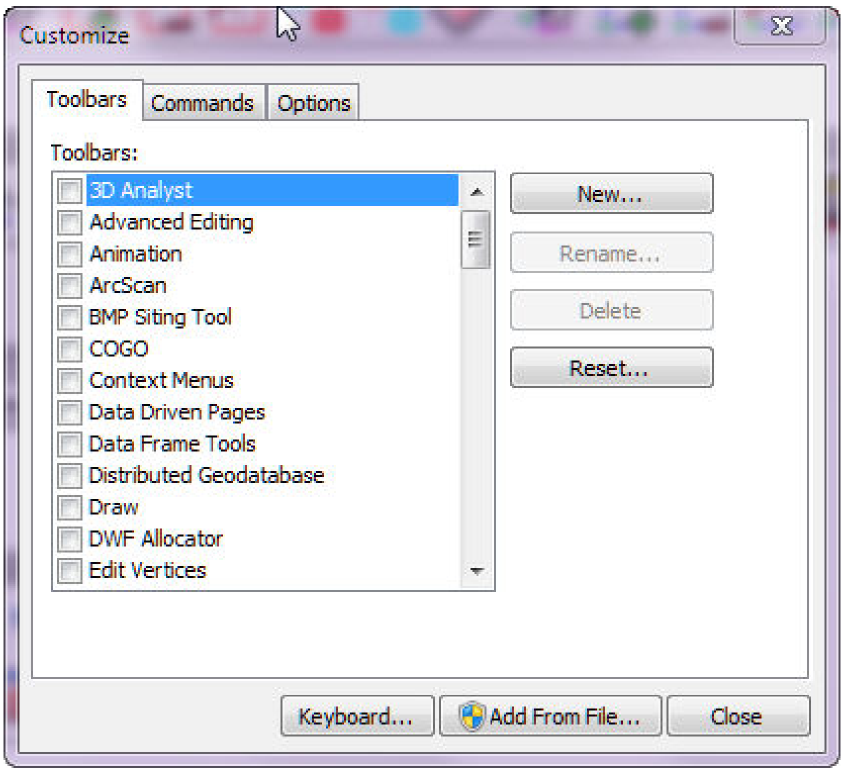
Advanced Editor



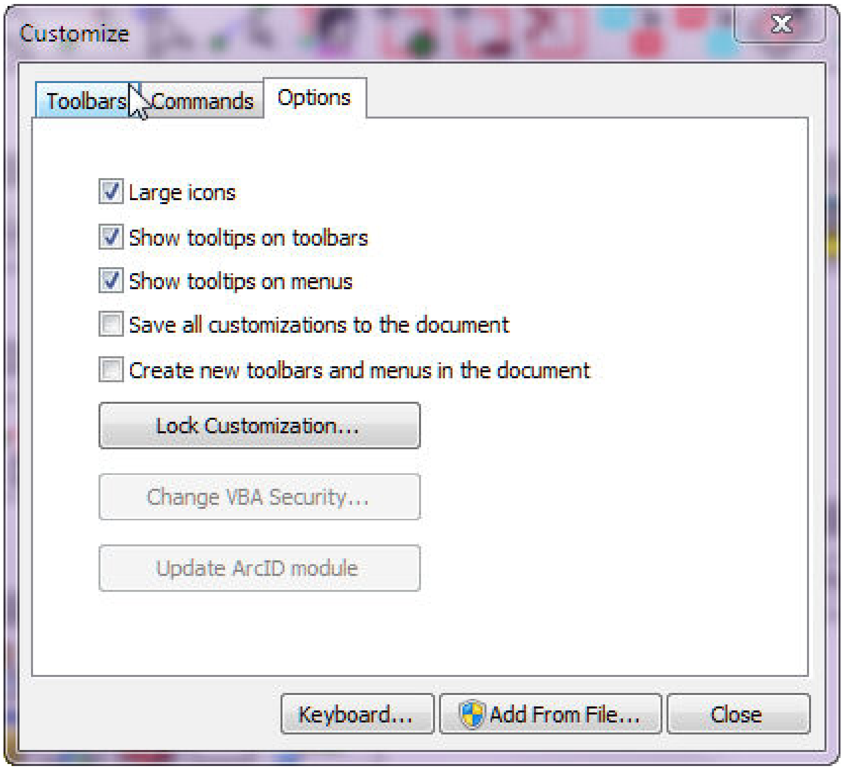
[Editor]



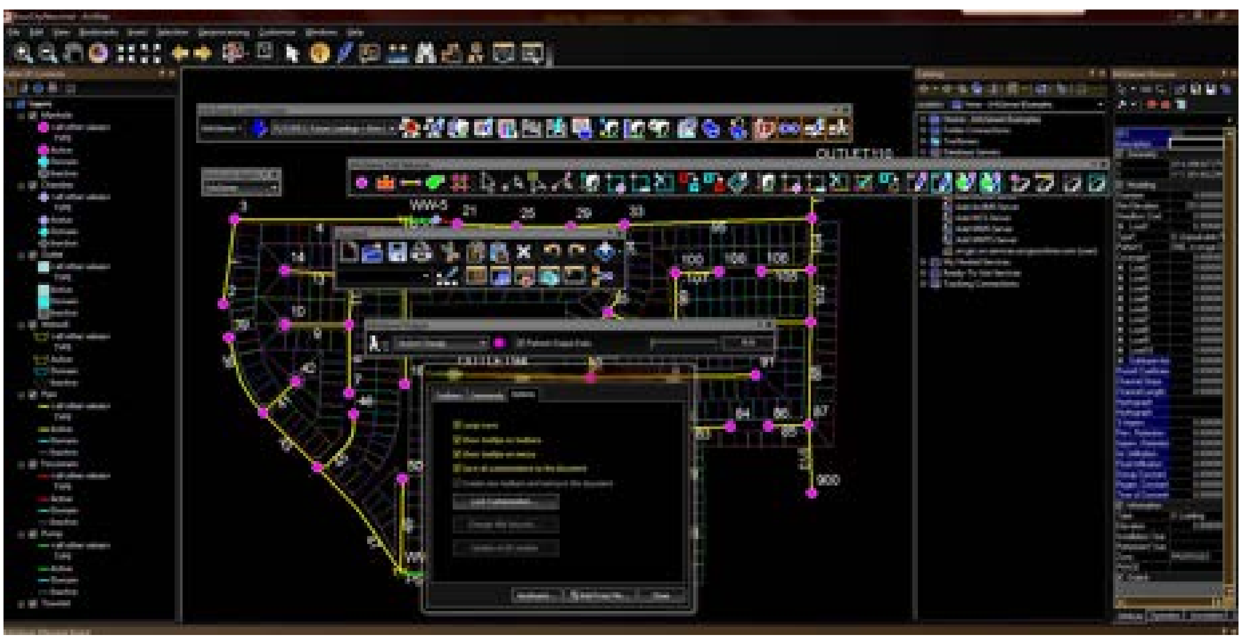
[Customize]



[Large Icon Options]



The large icons apply to the Arc Map and InfoSewer Icons, they can be saved permanently to the current MXD file

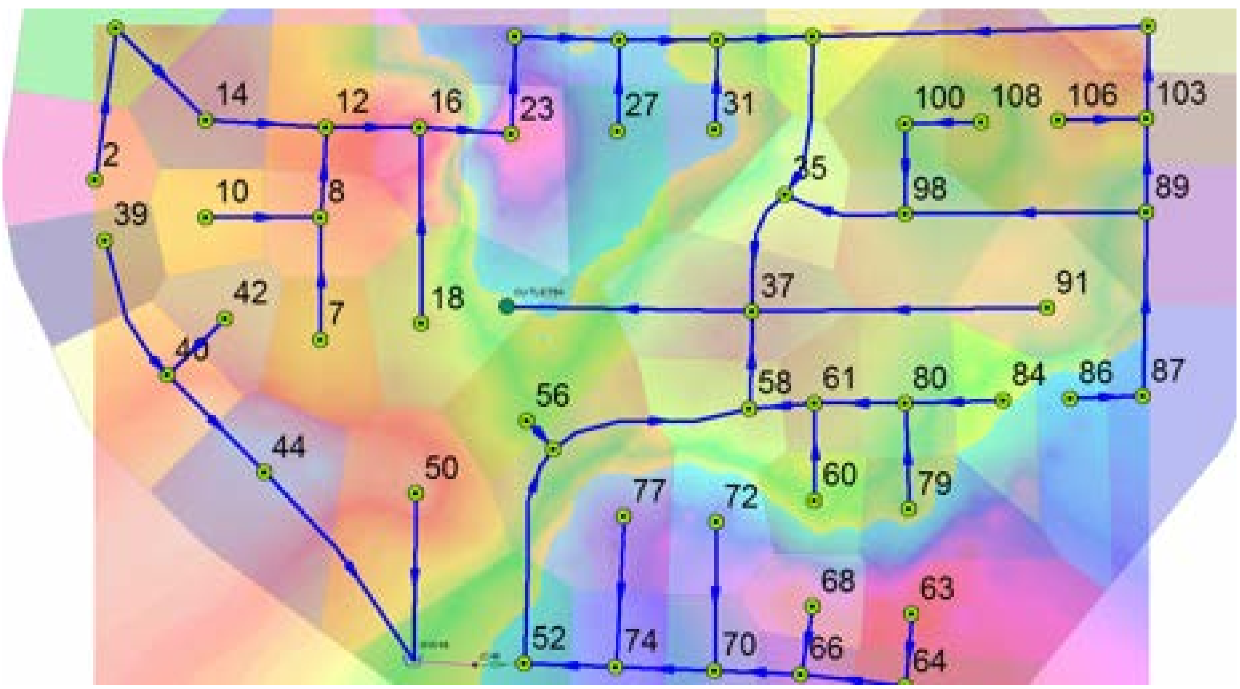


Other Related Topics - ArcMap Pull-Down Menus, Drawing Area

Drawing Area

The InfoSewer drawing area displays the actual network schematic. You may use a combination of InfoSewer and ArcMap commands to create, modify, edit and maintain your InfoSewer projects. You may bring in Base maps and/or other external drawings into ArcMap and use it to create your collection system.

Any of the InfoSewer elements such as pipes, pumps, wetwells and manholes may be created in the drawing area using the InfoSewer Create menu commands. Data may be associated with each of the elements in this drawing. These values will be stored in the InfoSewer database and may be edited at any time. Each element has a different symbol to identify them with. For any of the different available InfoSewer commands refer the InfoSewer help file.



Other Related Topics - ArcMap Pull-Down Menus, ArcMap Toolbars

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Innovyze Info Sewer User Guide

Introduction

In the interest of making more complete help systems for the users of InfoSewer we have combined the Help file and User Guide with Tutorials in this Combined Help File. The following sections are from the previously separate PDF User's Guide. The User Guide / UG was added to the Help file in 2014/2015.

A major capital investment made by a community is its sanitary and storm sewer collection system. These systems are essential to protecting the public health and welfare in all areas of concentrated population and development. Sanitary Sewers perform the vitally needed functions of collecting domestic, commercial, institutional, and industrial wastewaters and transporting them to locations of treatment and disposal. Municipal stormwater collection systems are used to store and safely transport stormwater runoff without causing flooding or undesirable overflows. More than 16,000 wastewater utilities now provide safe and reliable wastewater treatment services to seventy percent of the nation’s population and more than ninety percent of the nation’s urban population. These achievements represent public investment of some $500 billion.

However, many of our existing sewer systems are under considerable stress, either because of rapid growth or deterioration of an aging infrastructure. Growing communities continue to expand their sewer systems outward. Renewed interest in urban development and gradual land use changes have resulted in a drastic increase in loads on an older infrastructure that was not designed to sustain these additional flows. Through time the carrying capacity of these systems is eventually greatly reduced leading to sewer collection systems that are unable to carry their flows.

Rehabilitation, replacement, and expansion of sanitary sewer collection systems are sound remedial alternatives but can necessitate large expenditure which can often exceed available funds. Good engineering decisions based on sound analysis procedures will be required if the alterations and improvements to these systems are to be effective and economical. The effective management and rehabilitation of these systems is a multifarious task involving the use of a large amount of digital information with sound engineering analyses to accurately identify problem areas, restore needed capacity with the least amount of system improvement effort, and to efficiently administer the existing resources.

Today’s wastewater utilities and municipalities are realizing the benefits of geographic information system (GIS) technology in managing their sanitary sewer conveyance system infrastructures. This has led to a greater need for a seamless integration of GIS technology with advanced network modeling techniques to ensure that sound, cost-effective engineering solutions can be accomplished in the design, planning, maintenance, and operation of these systems. InfoSewer Pro was developed (by engineers for engineers) specifically to fully address this need by producing a common platform and interface for these synergistic applications.

Built with advanced object component technology, InfoSewer Pro combines a Relational Database with Spatial Analysis tools and mapping functions to provide a powerful decision-support application that integrates sophisticated and accurate network loading and modeling algorithms with complete infrastructure (asset) and business planning. It interfaces seamlessly with standard geographic information systems for spatial database management and analysis to give you an informative structured framework for sewer model construction, graphical editing, network simulation, results presentation, map generation, and enterprise-wide data sharing and exchange. You now have the reliable tool that lets you leverage your existing GIS data investments to help you identify the condition of your sanitary sewer and storm collection system, determine the most cost-effective way to obtain maximum reduction of your system overall's flows, restore/increase system capacity, optimize system rehabilitation and improvement options, satisfy the modeling requirements of the USEPA's Capacity, Management, Operations and Maintenance (CMOM) program, and improve public relations.

InfoSewer contains scores of cutting-edge performance features and mapping functions to help streamline and automate your sanitary and storm sewer network management activities. Within a true GIS environment, it performs both steady state and true dynamic simulations, utilizing a full-featured, state-of-the-art hydraulic computational engine, while providing complete dynamic mapping and information management functions. Through the use of extensive scenario/facility management functionality, InfoSewer Pro is also capable of analyzing existing or proposed sanitary and storm sewage collection systems.

InfoSewer Pro bridges the gap between network modeling and GIS software to support many types of applications in sewer collection system analyses, including:

Master Planning	**CMOM Program Planning**
Facility Sizing	**Wet-Well Design**
Facility Management	**Outlet Configuration**
Operations Study	**Pump Cycling**
Real-time Simulation	**Infrastructure Rehabilitation/Replacement**
Peak Flow Effects	**Flow Capacity Evaluation**
Overflow Management	**Infiltration/Inflow Modeling**
System Deficiency Identification	**System Improvements**
BOD Modeling	**Odor Control**
Source Tracing	**Pollutant Contribution Analysis**
Discharge Permit Compliance	**Corrosion Control**
Treatment Process Efficiency Evaluation	**Sewage Age Improvement**
ediment Modeling and Impact Analysis	**Sensitivity Analysis**
Alternative Analyses	**Dynamic Loading Comparison**
Flow Monitoring Design	**Model Calibration**
Operator Training	**Capital Improvement Program Development**
Capital Budgeting	**Impact Fees Determination**
System Expansion	**New System Design**
System Layout Mapping	**Dynamic Profiling**
Project Management	**Stormwater Modeling**

These comprehensive capabilities will greatly assist you in making informed decisions to ensure the most reliable and cost efficient sanitary and storm sewer conveyance system and keep your system operating properly into the future.

Our high-level, state-of-the-art research and development effort in GIS-based network analysis is continuing at a rapid pace and we intend to update and refine InfoSewerPro to reflect this progress. We are pleased to be at the forefront of this computer technology and to continue to advance it to an unprecedented level of reliability and performance.

InfoSewer - Innovyze Sewer GIS, DB, Hydraulics and Hydrology Components

---

Colby T. Manwaring, P.E.

Chief Executive Officer, Innovyze Inc.

Portland, Oregon USA March 31, 2021

Conventions

This guide uses the following typographic conventions:

ITALIC CAPS InfoSewer Pro menu titles, menu choices, and commands:

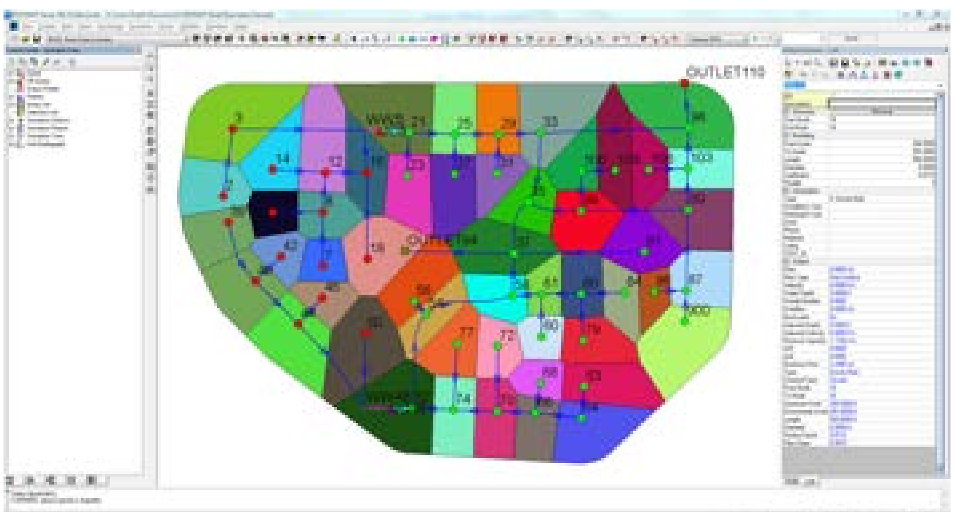
Select the *ADD PIPE* command from the *CREATE* menu.

Dialog box and window titles, and specific areas within a dialog box or window:

Choose "Gallon/Minute" from the *FLOW UNIT* drop-down list on the *SIMULATION OPTIONS* dialog box.

Bold Names of InfoSewer Pro projects:

The tutorial makes use of a sample InfoSewer Pro project named "**Yourcity**".



2.0 InfoSewer Capabilities

In the past few years, advances in infrastructure management technology have been occurring at an accelerated pace. This technology is now being realized within the context of sanitary and storm sewer modeling. It has allowed sewer collection system modeling applications to seamlessly integrate with GIS productivity software for comprehensive spatial data management and analysis in a shared management and decision making environment. You can now realize synergistic relationships between maps and the assets you manage to effectively sustain the flow of information to help you support many organizational programs and policies.

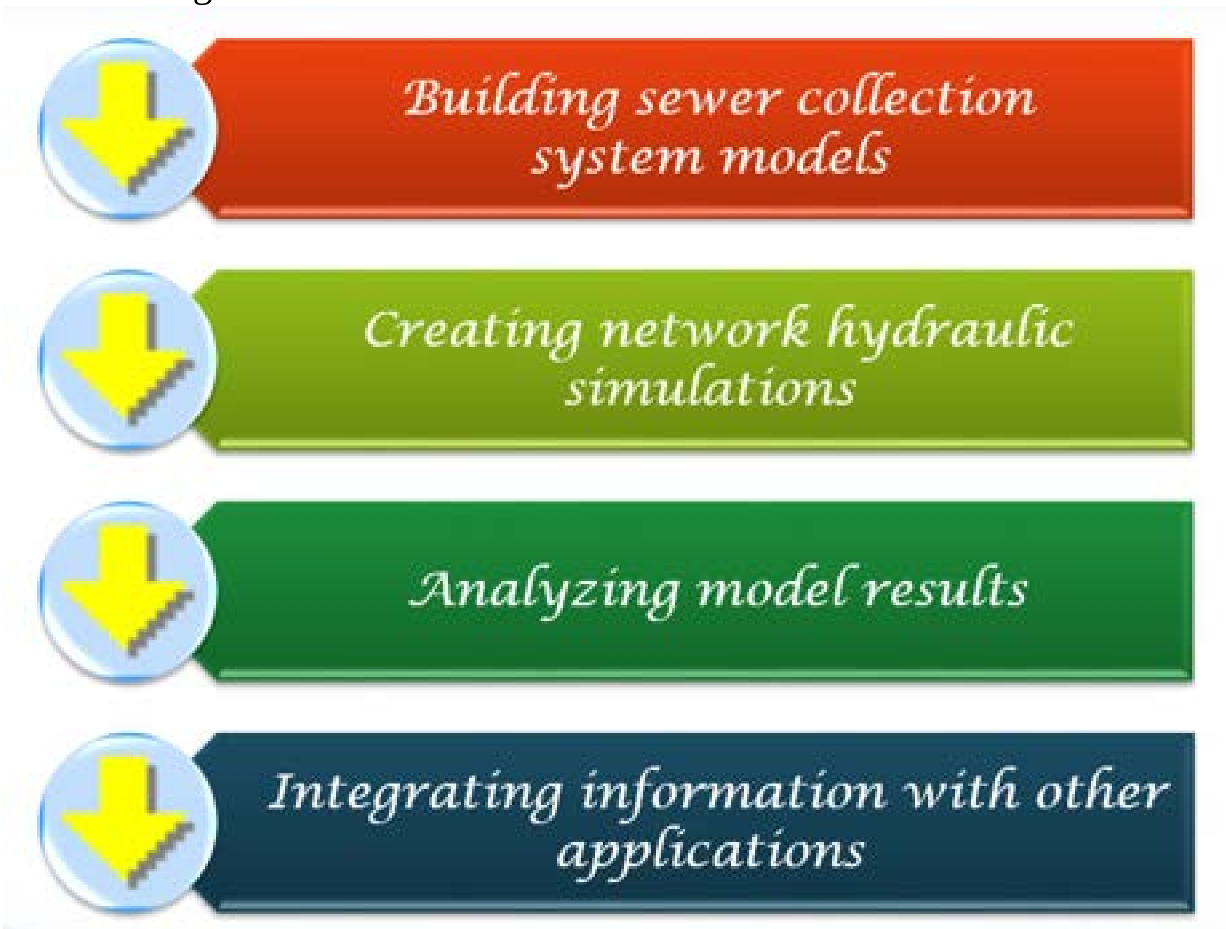
InfoSewer provides four major capabilities for the modeling and analysis of sewer conveyance systems:

- Building sewer collection system models - InfoSewer was designed to greatly facilitate and automate many of the data compilation and management duties in network modeling. Through its powerful and intuitive menu-driven interface, the user can construct an accurate representation of the distribution system, including a map of the system, attributes required for analysis, and other user-defined attributes.

- Creating network hydraulic simulations - Upon building an appropriate representation of the conveyance system, the user may perform several different types of steady state and extended period hydraulic simulations.

- Analyzing model results - Upon completion of each simulation, the user can view, query, and display modeling results using sophisticated graphical presentation tools including dynamic map annotation/labeling, color-coding, graphing, contouring, profiling, customizable tabular reporting, and vivid VCR-style animation to produce truly compelling results.

- Integrating information with other applications - Built with object-component technology and supporting numerous database, CAD, GIS, and data formats, InfoSewer offers many powerful GIS and mapping functions. Its unique open-architecture framework allows information to be easily exchanged with other applications including CAD and presentation applications, spreadsheets, and relational databases, customer information systems, work management systems and any other information system your organization uses.



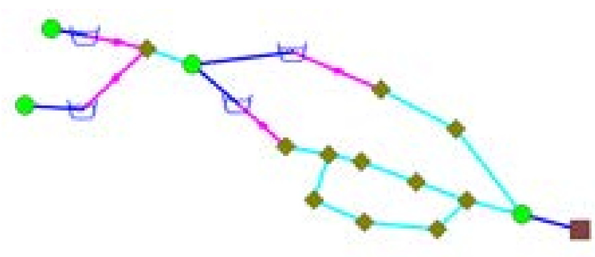
InfoSewer is developed to run entirely within a stand-alone (i.e., no additional CADD or GIS software required), true GIS-based environment.

InfoSewer tools are accessed using customized object-component commands, menus, and toolboxes, and the graphical user interfaces for these tools are based exclusively on high level Windows interface standards.

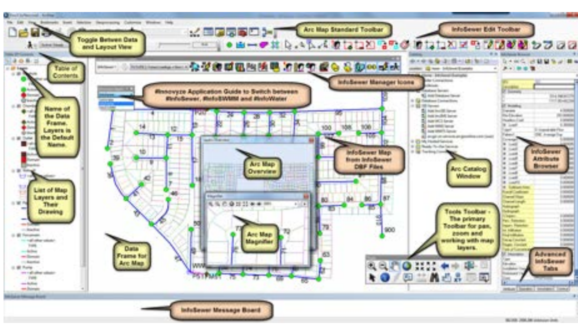
2.1 Building Sewer Collection System models

InfoSewer is developed to take full advantage of the capabilities of GIS and relational database management systems (RDBMS) technology. By embedding GIS mapping functions and utilizing proven and efficient tools for information exchange and graphical data entry, maintenance and display, InfoSewer provides the user with an efficient easy-to-use vehicle for network model construction and management.

In InfoSewer, a sanitary and storm sewer collection system model consists of two major components:

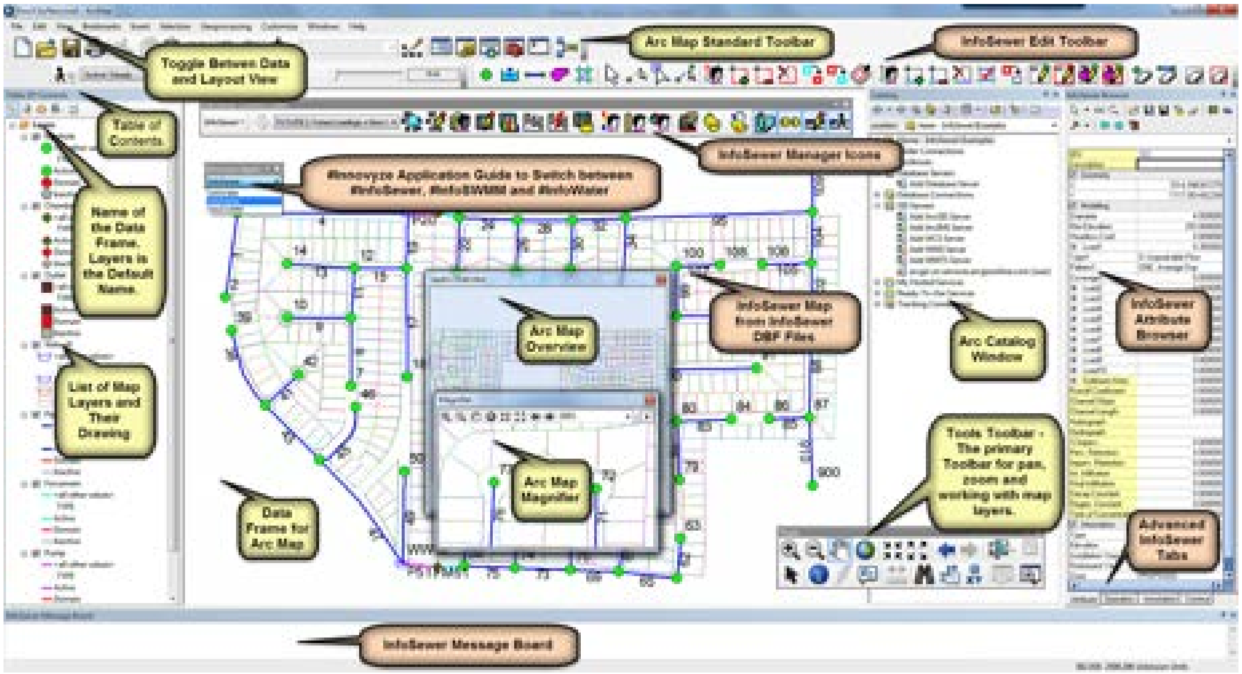


Network schematic - The network schematic provides a critical role as a model input, providing information on the connectivity, length, and shape of network components and it also plays an important role in the display and analysis of model results, acting as the vehicle for results presentation.



Modeling attributes - Information associated with network elements act as the primary input to the hydraulic network simulator. This information includes a set of required attributes describing critical components of system operations and a set of user-defined attributes for integration with other infrastructure management applications.

InfoSewer stores the network schematic and modeling attributes in GIS Shapefile and DBF format. It provides the following tools to build and maintain sewer collection system models.



Importing Existing Networks

InfoSewer supports multiple mapping layers which can be imported from one of many data sources including Computer-Aided Design (CAD) drawings (e.g., DXF, DWG, DGN), CAD world files, standard GIS formats (Shapefiles, Generate files, MID/MIF files, and ArcInfo coverages), Vector Product Format (VPF) files, Spatial Database Engine (ArcSDE) Layers, attribute tables, grid data, image files, and Comma Separated/Delimited Text (CSV) files. H_2OMAP Sewer/Pro also supports the new geodatabase standard of ArcGIS through ArcSDE connection. This comprehensive range of data format support readily allows the user to import existing models into InfoSewer from other sewer network modeling software.

Because it accesses GIS data directly in native format, InfoSewer can become an integral part of an enterprise-wide GIS, compatible with any GIS application.

Data Entry

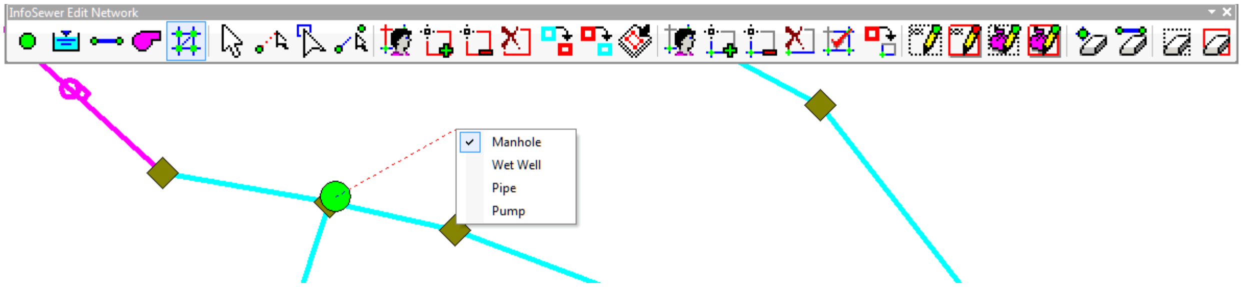
InfoSewer allows you to create network components including gravity mains, force mains, pumps, outlets, wet-wells and manholes and to simultaneously add the modeling attributes associated with these components. Gravity mains (free surface, partial flow) and force (full flow) mains can be of any shape and their lengths are automatically computed to the scaled drawing length using standard GIS measurement techniques. Multiple map layers can be easily displayed showing streets, parcels and boundaries. InfoSewer's unique open-architecture environment enables the user to build the model using other software packages. These applications can be used to populate the database for the various network elements. This data can then be retrieved into H_2OMAP Sewer/Pro for further manipulation and customization.

DATA MAINTENANCE

The user can move, add, insert, merge, split, or delete network component graphically at any time. Additionally, the user may modify any of the attributes associated with existing network components. Updates to the network map and database records are immediate and automatic.

DATABASE CUSTOMIZATION

Although InfoSewer requires a prescribed database structure to support the network simulator, the open-architecture environment enables the user to create and manipulate any number of additional attributes. The database tables for gravity mains, force mains, pumps, manholes, and wet-wells can be customized to better organize and manage system data and to support other infrastructure management applications.



InfoSewer Edit Network
Manhole
Wet Well
Pipe
Pump

2.2 Simulation Capabilities

InfoSewer provides unparalleled network analysis and simulation capabilities for performing a wide range of essential modeling tasks. With InfoSewer, the entire sewer collection system or any selected sub-system may be analyzed under steady state and extended period simulations. Please note the stormwater capabilities described here are only available for InfoSewer licenses. It utilizes a full-featured, state-of-the-art hydraulic computational engine that includes the following capabilities:

- places no limit on the size of the network and number of components that can be analyzed (unlimited link version)
- supports both English and metric (Standard International) units
- analyzes steady-state, extended period simulations (EPS), and dynamic simulations
- uses the Hazen-Williams (pressure) and Manning (open channel) friction formulas
- analyzes both pressurized (force mains) and partial (free surface) flow conditions
- accommodates multiple outlets
- supports any number of loops and parallel pipes
- models any number of separate sanitary and storm sewer networks in a single project
- models overflows and flow-splitting diversions
- allows multiple pumps and lift-stations to be modeled using capacity flow (pump or wet-well capacity), two-point design flow exponential curve, or multiple (3 points) head-flow pump characteristic curve
- models constant and variable speed pumps
- controls on-off status of pumps based on wet well levels, wet well volumes, discharged flows, time or incoming flows
- allows for both constant diameter or variable area wet-wells

- conserves pumped flows and load components
- calculates the age of sewage (time of concentration) throughout a network
- tracks the percent of sewage flow from any given manhole reaching all other pipes and manholes over time
- computes true flow/hydrograph attenuation throughout the collection system using the advanced Muskingum-Cunge explicit diffusion wave model
- tracks advective movement of pollutants flowing through the network over time
- accounts for local headlosses for manholes and junctions
- provides automated adverse slope correction
- considers multiple loading categories at any manholes, each with its own pattern of time variation (e.g. hydrographs)
- supports various methods of loading conditions including contributing population, service area, peakable or unpeakable flows, or any other user-defined loading types
- accounts for infiltration/inflow effects
- automatically designs the entire network based on user specified system performance criteria (e.g., depth-of-flow to diameter ratio)
- calculates sanitary and storm sewer network replacement/improvement costs
- generates HGL profiles
- models circular and non-circular pipes/channels

calculates the age of sewage (time of concentration) throughout a network

tracks the percent of sewage flow from any given manhole reaching all other pipes and manholes over time

computes true flow/hydrograph attenuation throughout the collection system using

the advanced Muskingum-Cunge explicit diffusion wave model

tracks advective movement of pollutants flowing through the network over time

accounts for local headlosses for manholes and junctions

provides automated adverse slope correction

considers multiple loading categories at any manholes, each with its own pattern

of time variation (e.g. hydrographs)

supports various methods of loading conditions including contributing population,

service area, peakable or unpeakable flows, or any other user-defined loading types

accounts for infiltration/inflow effects

automatically designs the entire network based on user specified system performance criteria (e.g., depth-of-flow to diameter ratio)

calculates sanitary and storm sewer network replacement/improvement costs

generates HGL profiles

models circular and non-circular pipes/channels

places no limit on the size of the network and number of components that can be

analyzed (unlimited link version)

supports both English and metric (Standard International) units

analyzes steady-state, extended period simulations (EPS), and dynamic simulations

uses the Hazen-Williams (pressure) and Manning (open channel) friction formulas

analyzes both pressurized (force mains) and partial (free surface) flow conditions

accommodates multiple outlets

supports any number of loops and parallel pipes

models any number of separate sanitary and storm sewer networks in a single project

models overflows and flow-splitting diversions

allows multiple pumps and lift-stations to be modeled using capacity flow (pump

or wet-well capacity), two-point design flow exponential curve, or ultiple (3 points)

head-flow pump characteristic curve

models constant and variable speed pumps

controls on-off status of pumps based on wet well levels, wet well volumes, discharged flows, time or incoming flows

allows for both constant diameter or variable area wet-wells

conserves pumped flows and load components

Multi-Level Inheritance Scenario Manager

InfoSewer provides a comprehensive tree-type SCENARIO EXPLORER with complete project inheritance. Each project "child" scenario inherits the information from its "parent" scenario. You can now maintain a single model of your sewer system and quickly develop and evaluate numerous modeling alternatives to support your capital improvement decisions. Change a pipe diameter, a manhole loading, a peaking factor or any other parameter and the changes will cascade through the entire set of projects in an easy-to-use tree-like structure. You can easily switch between scenarios and compare modeling results instantly. You can even directly extract (cut) one or more individual pressure (or boundary) zones from the main simulation model as well as merge (paste) together any number of models for detailed analyses.

Components of an InfoSewer scenario

Each of the three components of a scenario can be further defined as follows:

- **Simulation set** – Created through the General tab, they define the simulation options (durations, timesteps, analysis parameters, etc.) associated with the scenario. There are three different option set types, each storing a logical grouping of simulation options. Click here for more information on simulation sets.
- **Facility set** – Defines the network facilities (components such as pipes, pumps, wet wells and manholes) to be used in a simulation. Only one facility set can be active at a time (facility sets are created through the Facility tab). Click here for more information on facility sets.
- **Data set** – Stores modeling data (pipe diameter, invert elevations, manhole loadings, etc.) associated with each facility in a separate external database. There are nine different data set types (as seen above), each storing its own unique logical grouping of modeling data. Click here for more information on data sets.

When you define a scenario, you pick the facility, data, and option sets that comprise that scenario. When picking data sets for inclusion in a scenario, you may either specify that a data set associated with a given scenario is included in that scenario independent from other scenarios or alternately may specify that the given data set inherits its contents – properties – from a "parent" scenario. Click here to learn more about Parent-Child relationships and how they are built.

Once you have configured and created a scenario, you can activate that scenario at any time. Once a scenario is activated, any modifications made to any of the databases related to InfoSewer facilities will be changed, but only for the data sets that are related to, and dependent upon, the active scenario. Click here to learn how to activate a scenario.

2.3 Analyzing Model Results

InfoSewer provides a set of customized mapping functions to greatly accelerate and simplify the presentation of modeling results. With the many data visualization tools available within InfoSewer Pro, the user can browse and analyze model results without the laborious and time-consuming process of working with formidable amounts of data and comprehending large volumes of tabular results.

Using standard GIS thematic mapping functionality, the entire network, or user-specified portions of the network, can be color-coded according to any network variable: gravity main size, full flow, design flow, analysis flow, total flow, full flow excess, design flow excess, analysis flow excess, water depth, critical depth, velocity; force main flow, velocity and headloss; manhole load, elevation; pump flow and head; and any other database attribute. Numerical ranges for color coding classifications and color selections are user-specified.

Map Labeling/Annotation

InfoSewer Pro provides a set of customized mapping functions to greatly accelerate and simplify the presentation of modeling results. With the many data visualization tools available within InfoSewer Pro, the user can browse and analyze model results without the laborious and time-consuming process of working with formidable amounts of data and comprehending large volumes of tabular results.

InfoSewer Pro provides “what you see is what you get” annotation. You can annotate and view simultaneously modeling data and results for network attributes and for any hydraulic or quality conditions directly on the network map. These dynamic labels automatically update as you switch between hydraulic time steps and scenarios. Annotation of the various element attributes can be directly customized and flow directional arrows can be displayed along with text information.

Graphing

InfoSewer provides a set of customized mapping functions to greatly accelerate and simplify the presentation of modeling results. With the many data visualization tools available within InfoSewer, the user can browse and analyze model results without the laborious and time-consuming process of working with formidable amounts of data and comprehending large volumes of tabular results.

Time-series graphs can be generated for network components and can be used to visualize how user-specified variables change throughout an extended period dynamic simulation. Variables such as force main flow rate, velocity, and headloss; gravity main design flow rate, actual flow rate, velocity, water depth, critical depth; time of concentration, percent source contribution; nodal elevation, hydraulic grade, and loading; pump operations, and others can be viewed on time-series graphs. Group graphs can be generated for any group of selected elements (e.g., series pipes) to depict the effect of flow attenuation over the specified length of pipes. Frequency distribution graphs for any variable can also be generated for any given time period depicting the fraction of manholes and links not exceeding specified values. Graphs can be fully customized by the user. The user can superimpose multiple graphs for comparison or display purposes, switch between 2D and 3D display (including several options for each type), change data markers, graph scales, axes, and titles. Observed field data can also be linked to a graph. This proves especially useful for model calibration.

Animation

InfoSewer Pro provides a set of customized mapping functions to greatly accelerate and simplify the presentation of modeling results. With the many data visualization tools available within H_2OMAP Sewer/Pro, the user can browse and analyze model results without the laborious and time-consuming process of working with formidable amounts of data and comprehending large volumes of tabular results.

InfoSewer Pro offers a very useful VCR-style animation feature that helps you step through an extended period dynamic simulation and instantly visualize and comprehend the dynamic performance of your sewer system such as temporal variations in water depth for the length of the simulation period.

See Also: Animation

Contouring

InfoSewer Pro provides a set of customized mapping functions to greatly accelerate and simplify the presentation of modeling results.

For any given simulation run, InfoSewer Pro allows you to efficiently generate accurate, smooth (TIN) contours for any numerical database attributes including manhole load, hydraulic grade, and elevation. Beautiful contours can be generated directly on the map and for the entire network or any user-selected portions of the network. Each contour map can be assigned to any specific map layer thereby allowing multiple parameter contours to be overlaid and displayed on a single drawing. Color grading and coding and numerical interval ranges as well as various useful graphical presentation options are user-specified. Contours generated in InfoSewer Pro are saved as GIS Shapefiles, therefore fostering easy distribution to others in your organization.

DATABASE BROWSING

With InfoSewer Pro, the user can browse model input data and analysis results in a tabular format. The data appears in a spreadsheet format on-screen, accessed directly from the project database tables. The user can customize the database display and may generate database reports using either standard or customized formats. You can also instantly locate network elements and view modeling results in a "point and click" mode for any time period of a dynamic simulation.

See Also: Contouring

Database Queries

InfoSewer Pro provides a set of customized mapping functions to greatly accelerate and simplify the presentation of modeling results. With the many data visualization tools available within InfoSewer Pro, the user can browse and analyze model results without the laborious and time-consuming process of working with formidable amounts of data and comprehending large volumes of tabular results.

Sophisticated query tools let you build complex intelligent queries on any database and modeling attribute (both input and output data) to meet a single criterion or multiple criteria simultaneously. You can isolate those network components that possess a desired property or those that may indicate a specific problem. Such capabilities can greatly assist you in the decision making process for network asset inventory, design and rehabilitation requirements, and financial planning. Query results can be displayed graphically, in tabular format, printed, or exported to a standard spreadsheet or word processor.

See Also: Mapping

Comprehensive Report Generation

InfoSewer Pro provides a set of customized mapping functions to greatly accelerate and simplify the presentation of modeling results. With the many data visualization tools available within InfoSewer Pro, the user can browse and analyze model results without the laborious and time-consuming process of working with formidable amounts of data and comprehending large volumes of tabular results.

Comprehensive report generation tools are provided for simultaneous display of input and output variables. Report tables can be completely customized by selecting the variables to display, the order in which they appear along with the desired units, display precision, column width, and format, and the corresponding hydraulic time period. You can include the entire network, any portions or selected elements. Each table can be sorted according to any variable and filtered based on any search criteria. Maximum, minimum and average values for all variables are automatically calculated and reported for EPS runs.

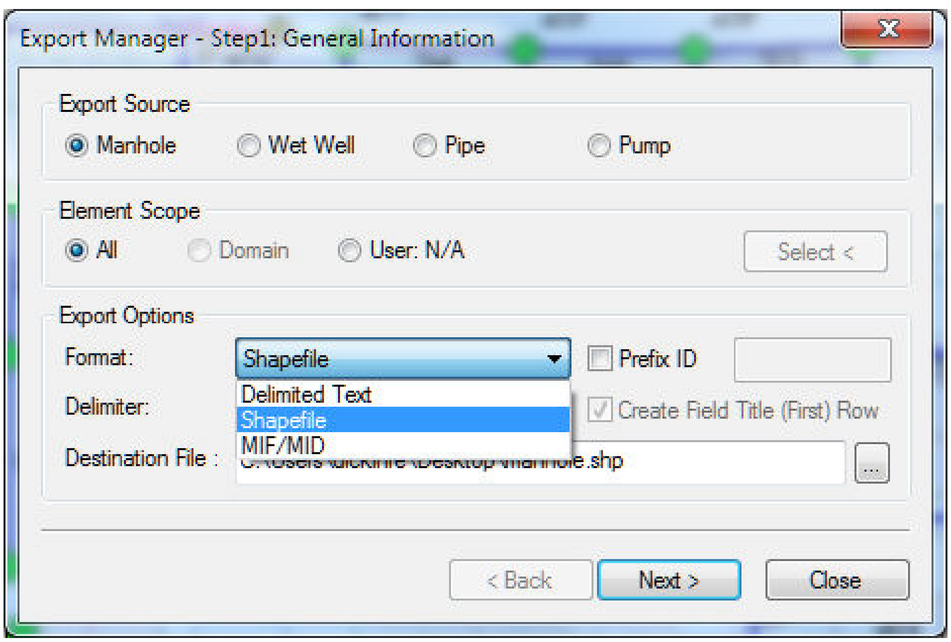
2.4 Exporting To Sanitary Sewer conveyance System Models

InfoSewer /Pro can write the contents of a network model created in InfoSewer Pro to standard GIS formats (e.g., Shapefiles) for use in other applications.

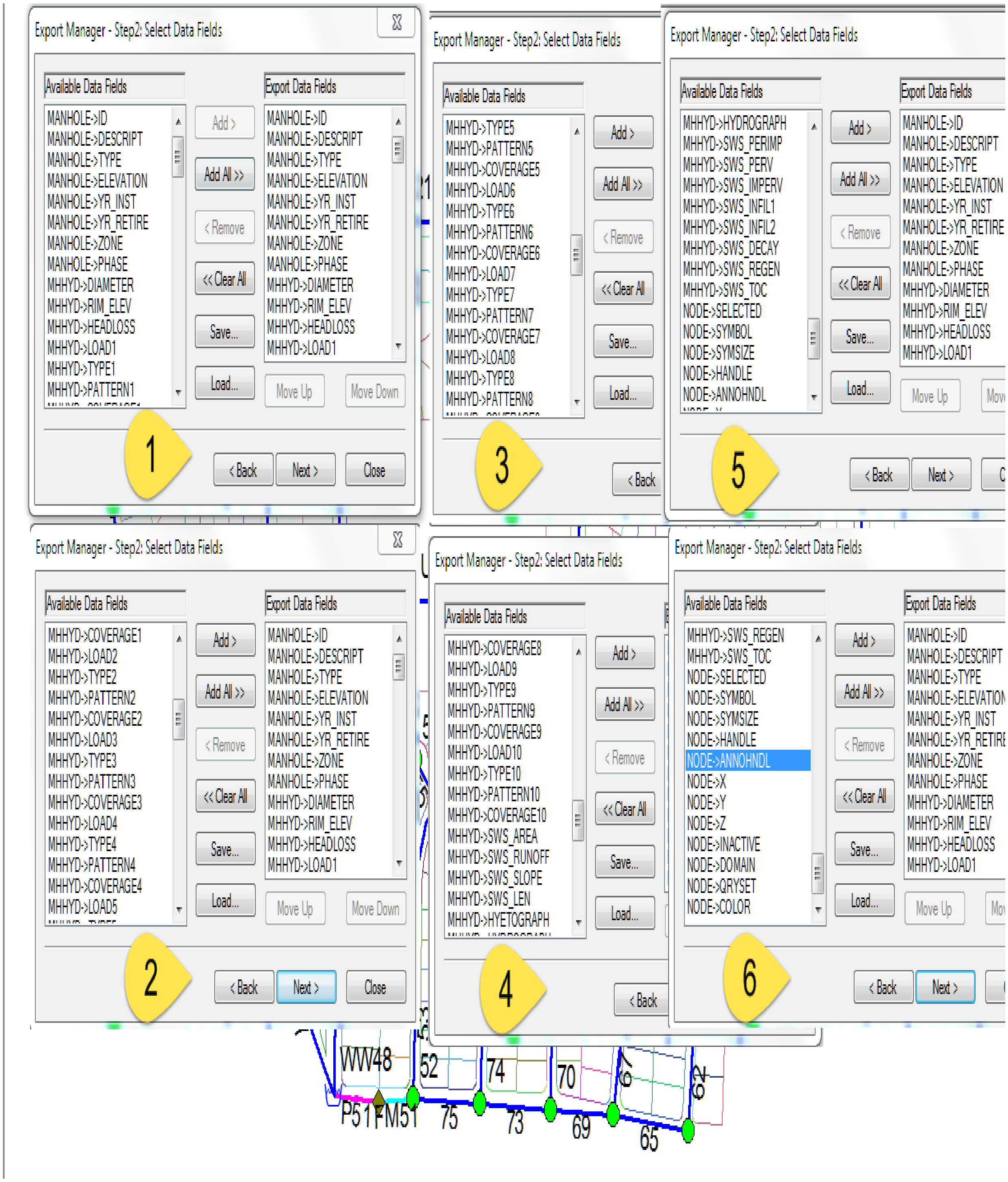
This will prove useful for sharing your modeling efforts with your organization via enterprise or web (Intranet or Internet) applications.

Export Options are:

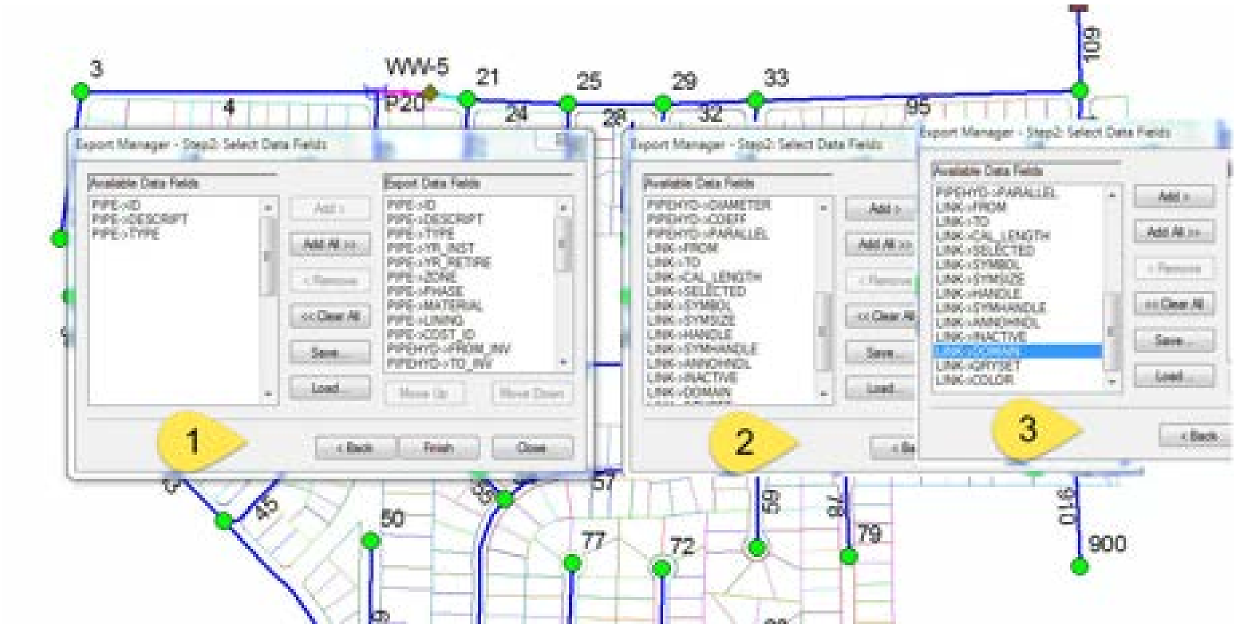
- CSV Files
- Shapefiles
- MIIF/MID
- for Pumps, PIpes, Wet Wells and Manholes



[Manhole Export and Import](#)

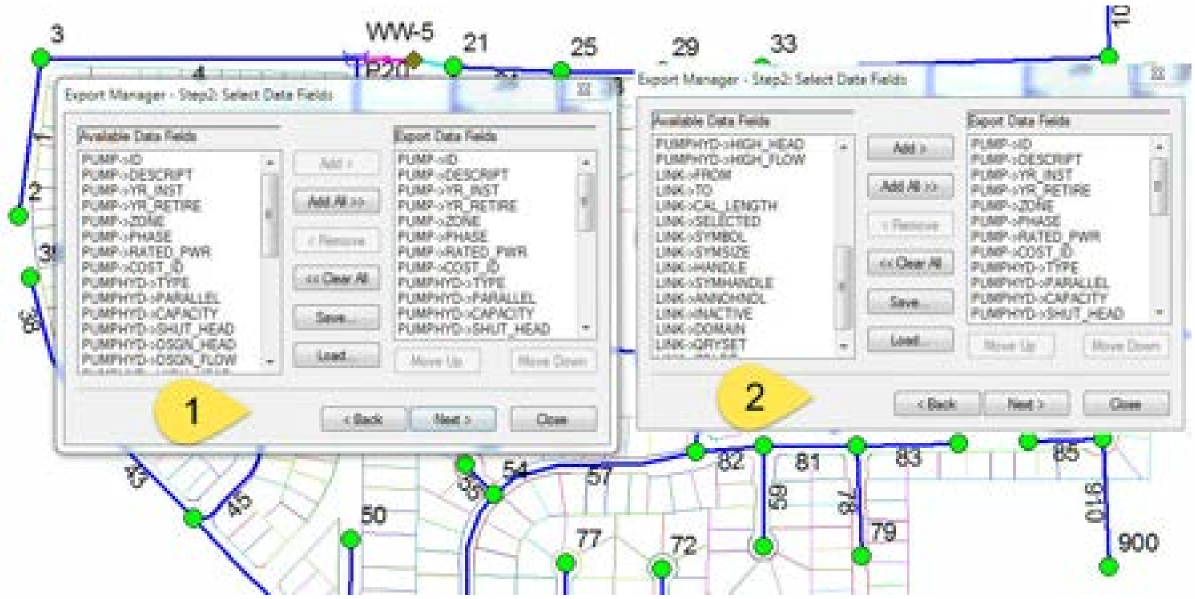


Pipe Export and Import



[Wet Well Export and Import]

[Pump Export and Import]



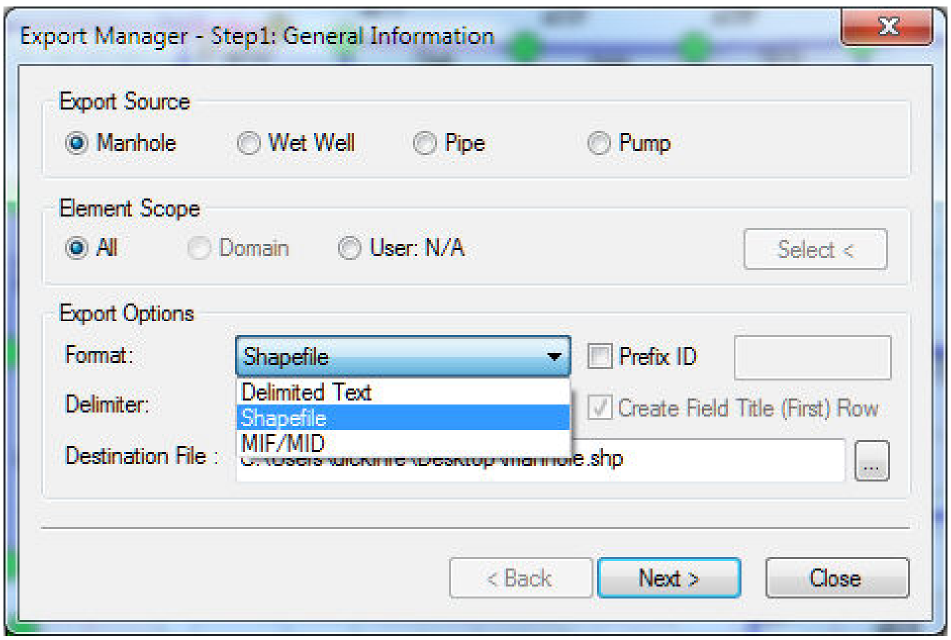
Export Manager - Step2: Select Data Fields
Available Data Fields
PUMP->ID
PUMP->DESCRIPT
PUMP->YR_INST
PUMP->YR_RETIRE
PUMP->ZONE
PUMP->PHASE
PUMP->RATED_PWR
PUMP->COST_ID
PUMPHYD->TYPE
PUMPHYD->PARALLEL
PUMPHYD->CAPACITY
PUMPHYD->SHUT_HEAD
PUMPHYD->DSGN_HEAD
PUMPHYD->DSGN_FLOW
Export Data Fields
Add >
Add All >>
< Remove
<< Clear All
Save...
Load...
Move Up
Move Down
< Back
Next >
Close
1
Export Manager - Step2: Select Data Fields
Available Data Fields
PUMPHYD->HIGH_HEAD
PUMPHYD->HIGH_FLOW
LINK->FROM
LINK->TO
LINK->CAL_LENGTH
LINK->SELECTED
LINK->SYMBOL
LINK->SYMSIZE
LINK->HANDLE
LINK->SYMHANDLE
LINK->ANNOHNDL
LINK->INACTIVE
LINK->DOMAIN
LINK->QRYSET
Export Data Fields
2
WW-5
900

EXPORTING NETWORK

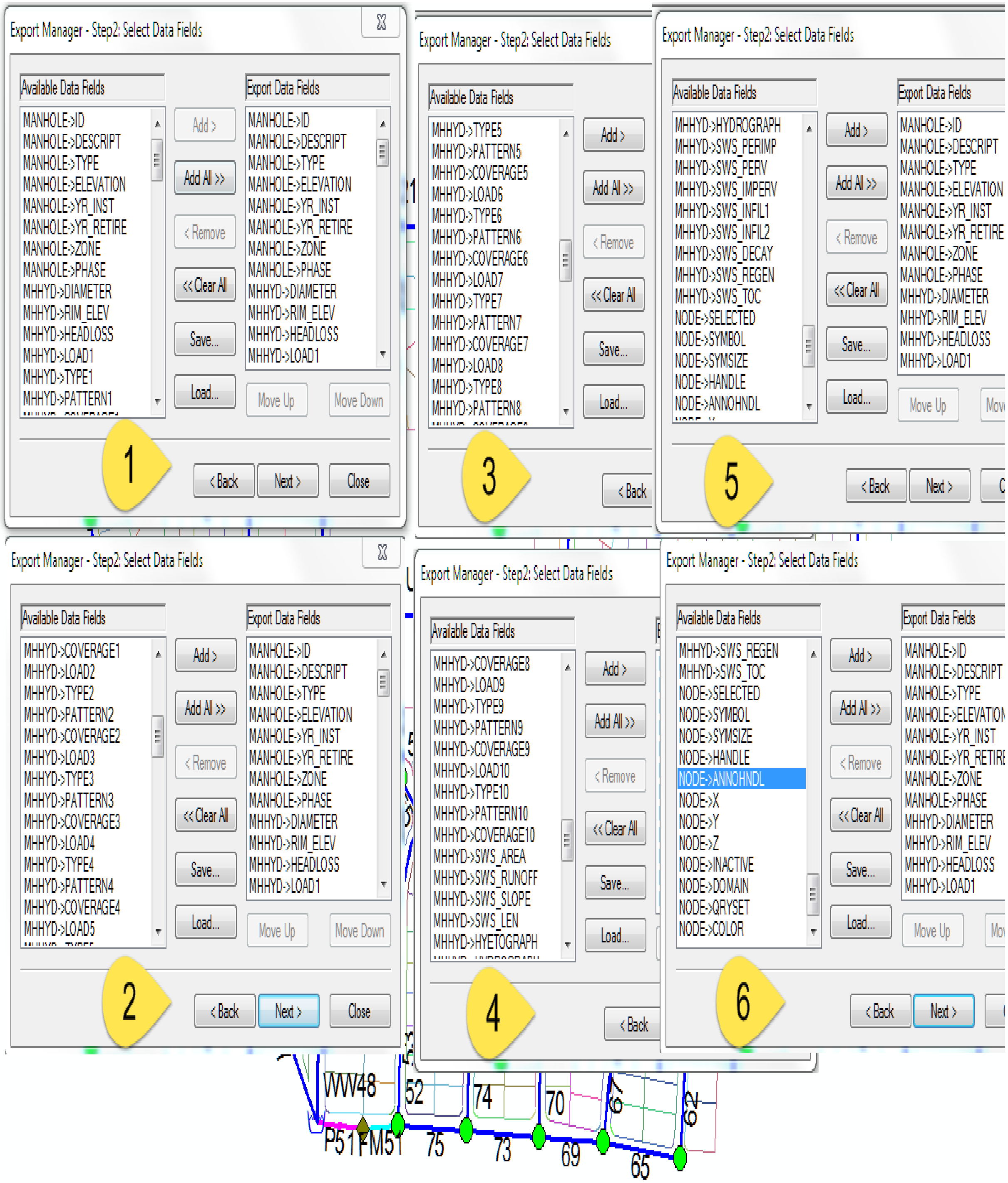
Just as an existing network map can be imported from another GIS application, a network map created with InfoSewer Pro can be easily exported to other GIS applications.

Export Options are:

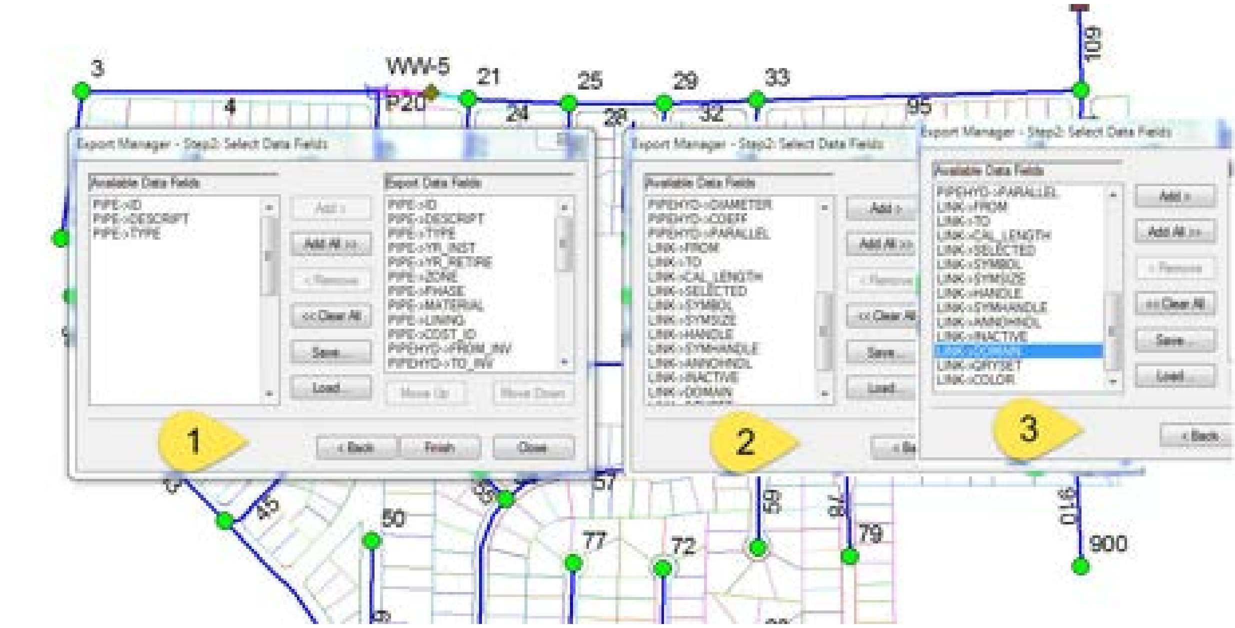
- CSV Files
- Shapefiles
- MIIF/MID
- for Pumps, PIpes, Wet Wells and Manholes



[Manhole Export and Import](#)

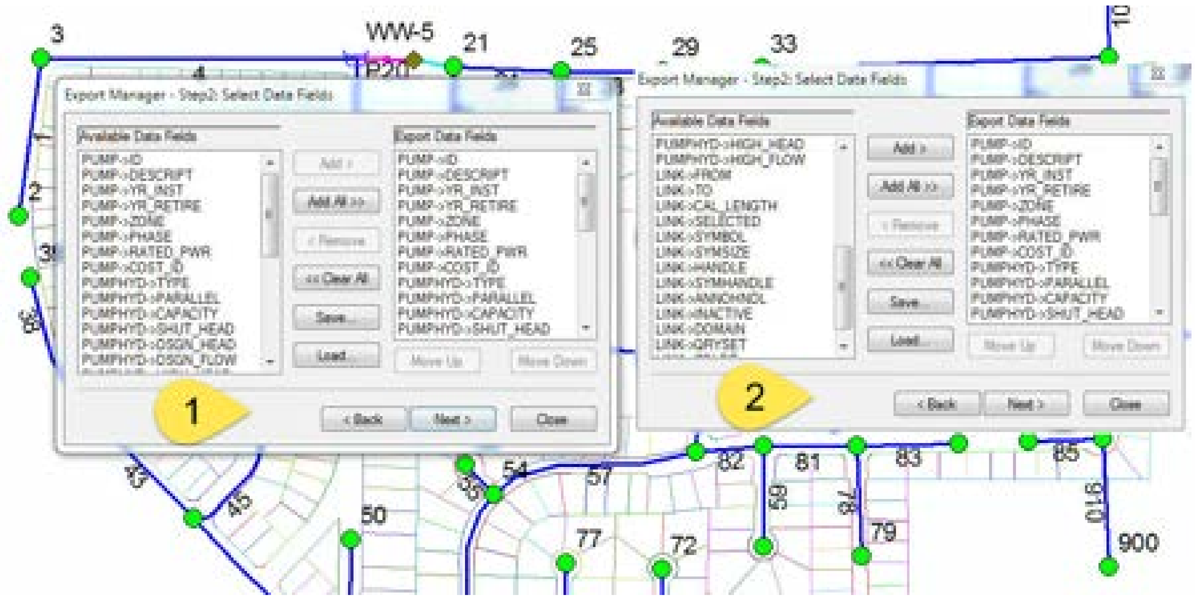


[Pipe Export and Import]



[Wet Well Export and Import]

[Pump Export and Import]

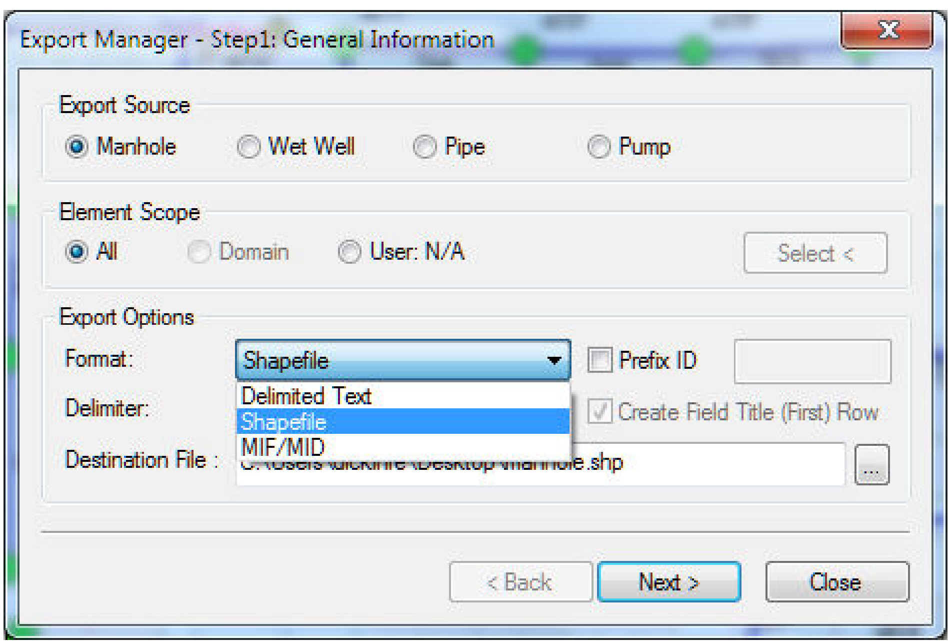

Export Manager - Step2: Select Data Fields
Available Data Fields
PUMP->ID
PUMP->DESCRIPT
PUMP->YR_INST
PUMP->YR_RETIRE
PUMP->ZONE
PUMP->PHASE
PUMP->RATED_PWR
PUMP->COST_ID
PUMPHYD->TYPE
PUMPHYD->PARALLEL
PUMPHYD->CAPACITY
PUMPHYD->SHUT_HEAD
PUMPHYD->DSGN_HEAD
PUMPHYD->DSGN_FLOW
Add >
Add All >>
< Remove
<< Clear All
Save...
Load...
Export Data Fields
PUMP->ID
PUMP->DESCRIPT
PUMP->YR_INST
PUMP->YR_RETIRE
PUMP->ZONE
PUMP->PHASE
PUMP->RATED_PWR
PUMP->COST_ID
PUMPHYD->TYPE
PUMPHYD->PARALLEL
PUMPHYD->CAPACITY
PUMPHYD->SHUT_HEAD
Move Up
Move Down
1
< Back
Next >
Close
Export Manager - Step2: Select Data Fields
Available Data Fields
PUMPHYD->HIGH_HEAD
PUMPHYD->HIGH_FLOW
LINK->FROM
LINK->TO
LINK->CAL_LENGTH
LINK->SELECTED
LINK->SYMBOL
LINK->SYMSIZE
LINK->HANDLE
LINK->SYMHANDLE
LINK->ANNOHNDL
LINK->INACTIVE
LINK->DOMAIN
LINK->QRYSET
Add >
Add All >>
< Remove
<< Clear All
Save...
Load...
Export Data Fields
PUMP->ID
PUMP->DESCRIPT
PUMP->YR_INST
PUMP->YR_RETIRE
PUMP->ZONE
PUMP->PHASE
PUMP->RATED_PWR
PUMP->COST_ID
PUMPHYD->TYPE
PUMPHYD->PARALLEL
PUMPHYD->CAPACITY
PUMPHYD->SHUT_HEAD
Move Up
Move Down
2
< Back
Next >
Close
WW-5
3
21
25
29
33
50
54
57
77
72
79
81
82
83
85
900

Exportng Databases

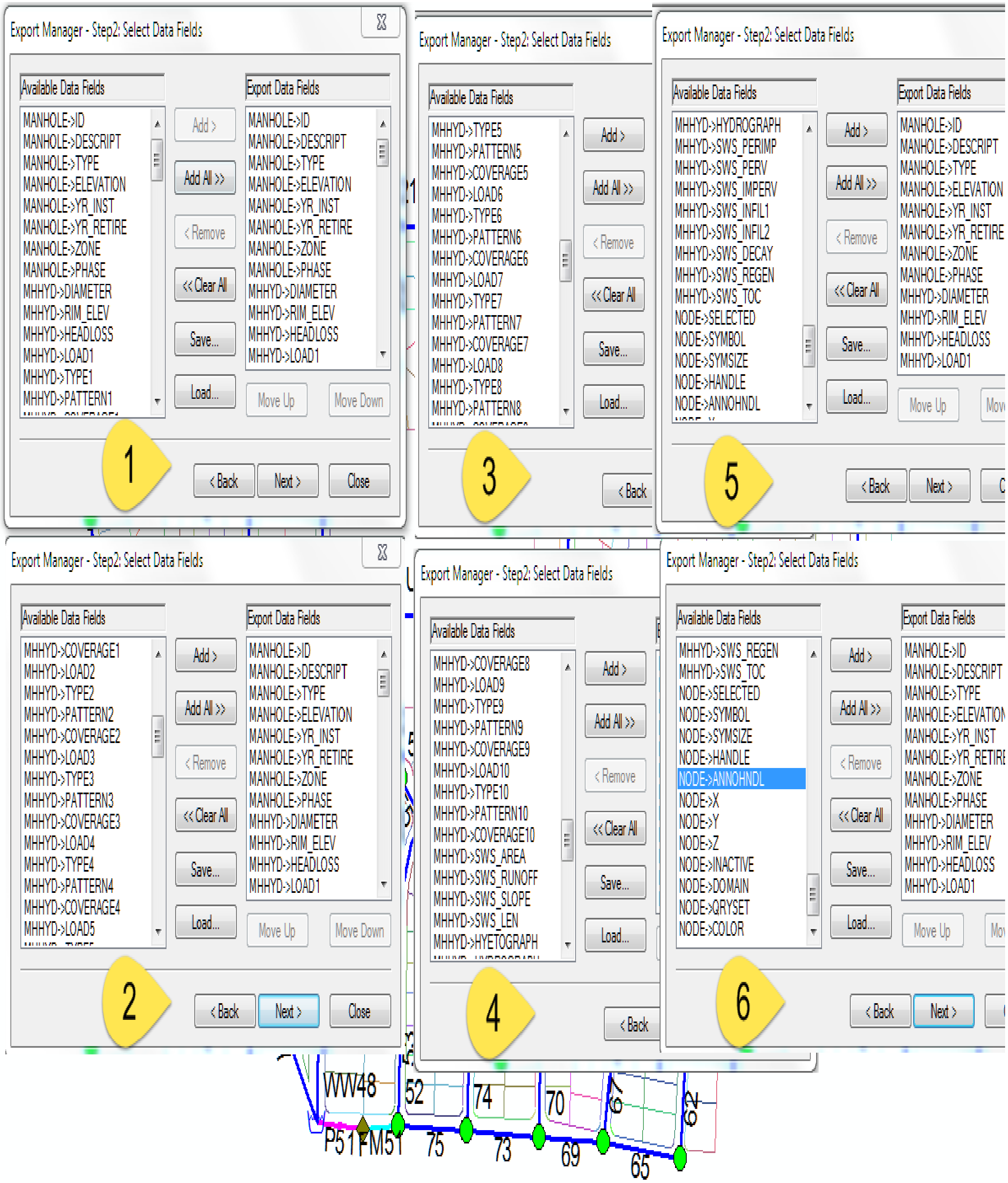
InfoSewer model inputs and outputs can be exchanged with other applications in four fashions. First, InfoSewer map and attribute data is stored in Shapefiles and DBF format and is therefore immediately compatible with any other application that can read or write from or to that format. Second, selected database rows or columns can be *copied* to the Windows Clipboard and then *pasted* into other Windows-compatible applications. Third, InfoSewer allows database tables to be exported (and imported) in a Windows comma delimited text (CSV) format. Finally, data can be directly accessed or shared through complete ODBC support.

Export Options are:

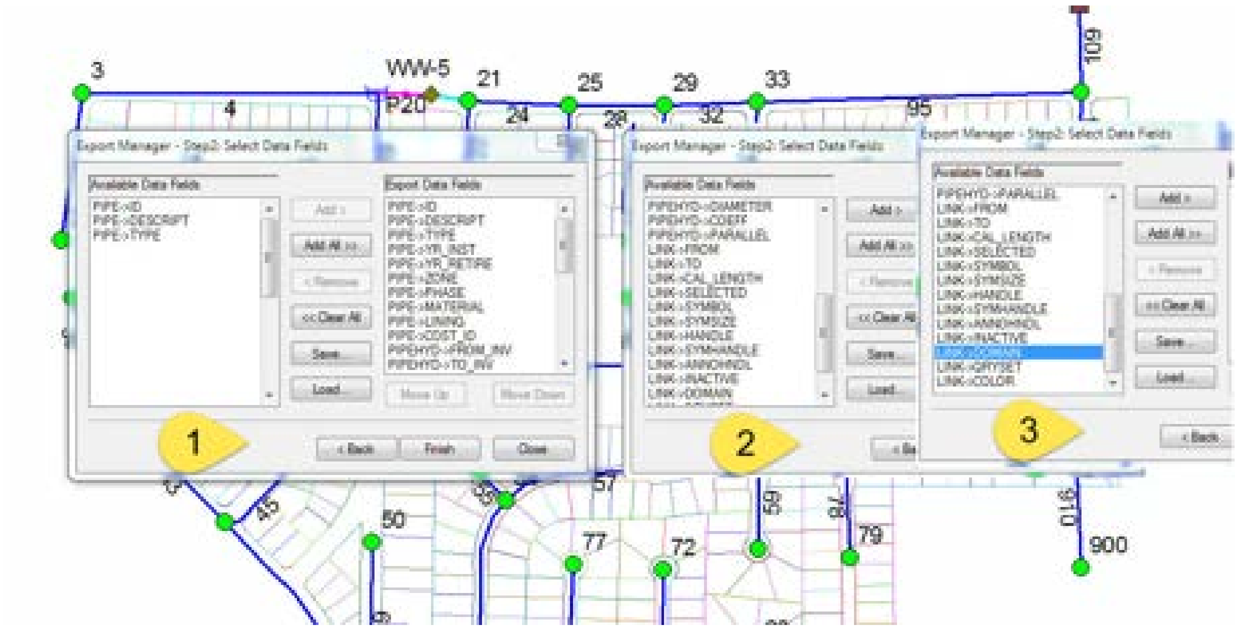
- CSV Files
- Shapefiles
- MIIF/MID
- for Pumps, PIpes, Wet Wells and Manholes



Manhole Export and Import

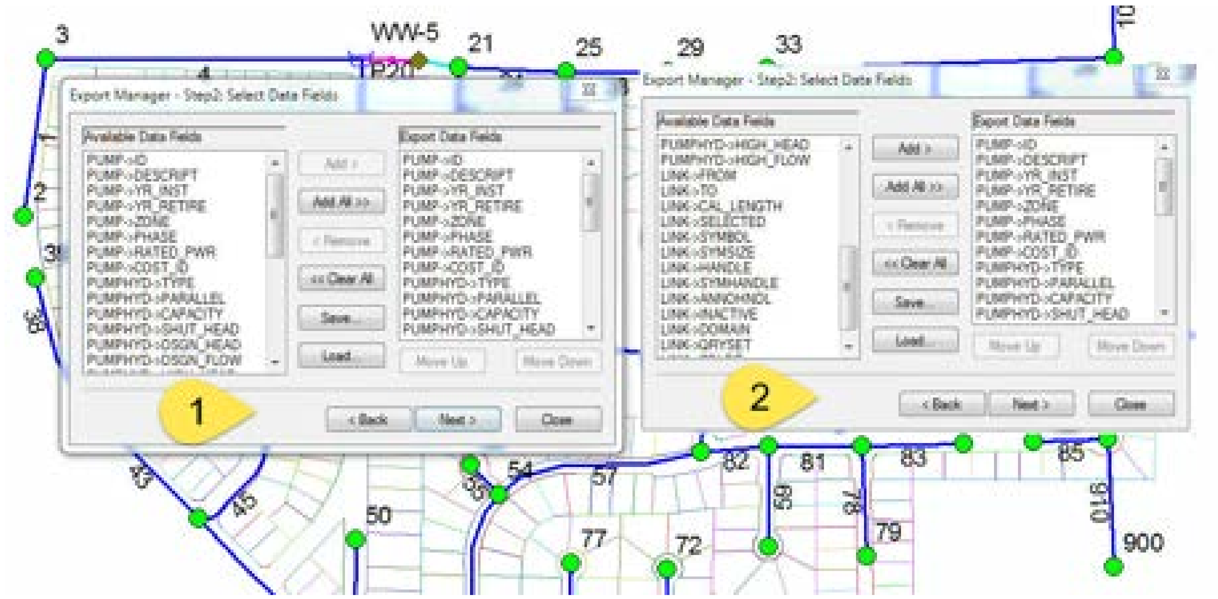


[Pipe Export and Import]()



[Wet Well Export and Import]

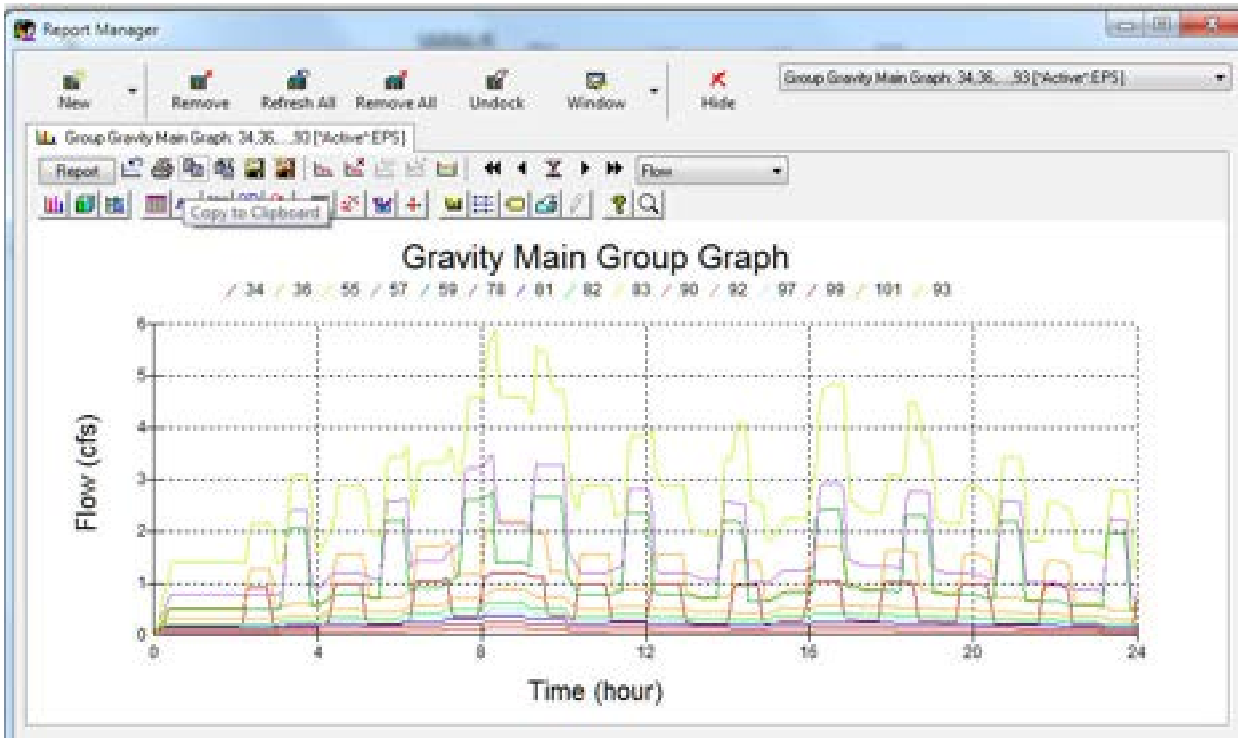
[Pump Export and Import]



Export Manager - Step2: Select Data Fields
Available Data Fields
Export Data Fields
Add >
Add All >>
< Remove
<< Clear All
Save...
Load...
Move Up
Move Down
< Back
Next >
Close
1
2

Exporting Graphs

InfoSewer Pro graphs can be exported to other applications via Windows *copy* and *paste* tools. After bringing a graph to the screen and optionally customizing the graph, the user can save the graph to a Windows Metafile (WMF) or Bitmap (BMP) format or export the graph display to word processors, presentation packages, or other Windows-based software applications.



More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Step 1: Open the Sample Project for InfoSewer

The first step is to open the InfoSewer project.

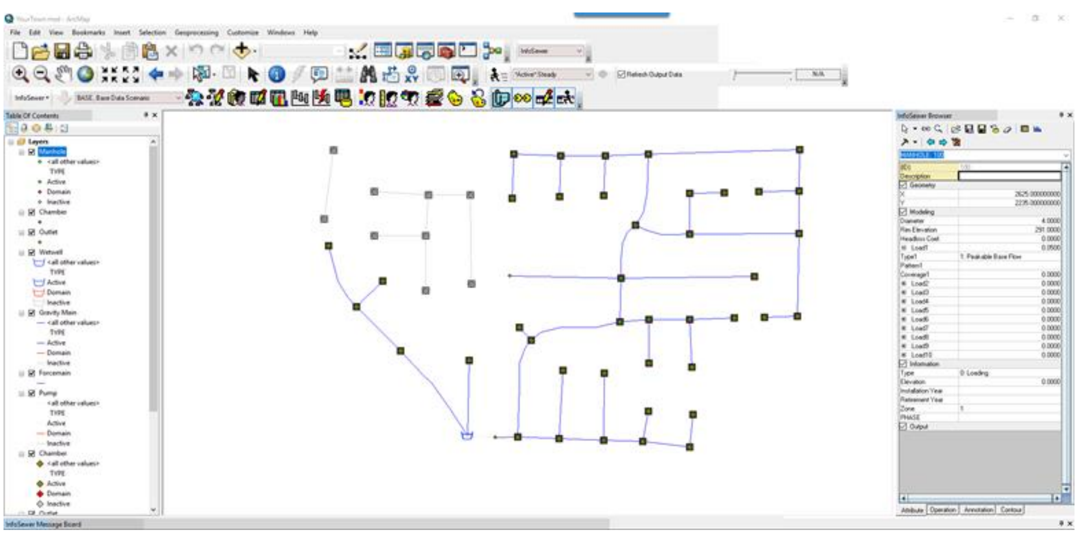
1. Choose the “Start” menu, select *PROGRAMS*, choose the InfoSewer *version X* program group, and then choose InfoSewer.

2. Choose *OPEN* from the *FILE* menu. On the *OPEN* dialog box, navigate to the directory containing the “YourTown” project and choose that file.

   C:\Users\Public\Documents\InfoSewer\Examples\YourTown.mxd

   (the path may be different for custom installations)

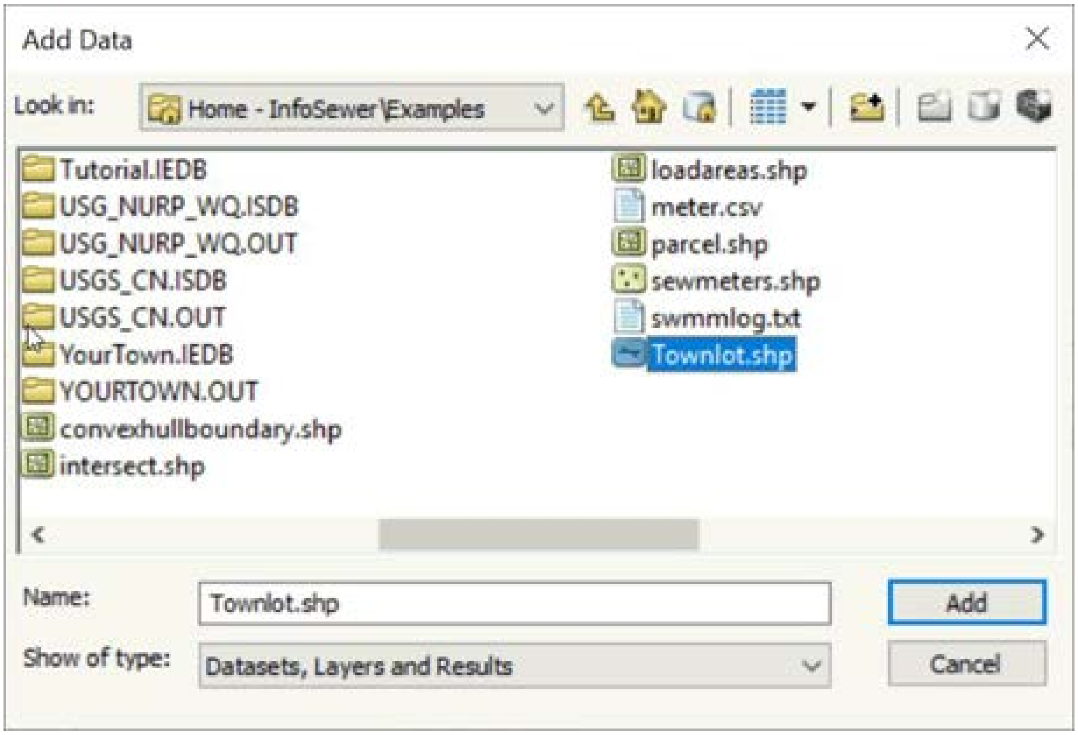
3. At this point the network map will be drawn on the screen. You will note that the drawing is missing several elements when compared to the illustration on the previous page. You will be adding those components during this tutorial.



4. **Add** “Townlot.shp” (Files of type: ESRI shapefiles *.shp) and select that file.

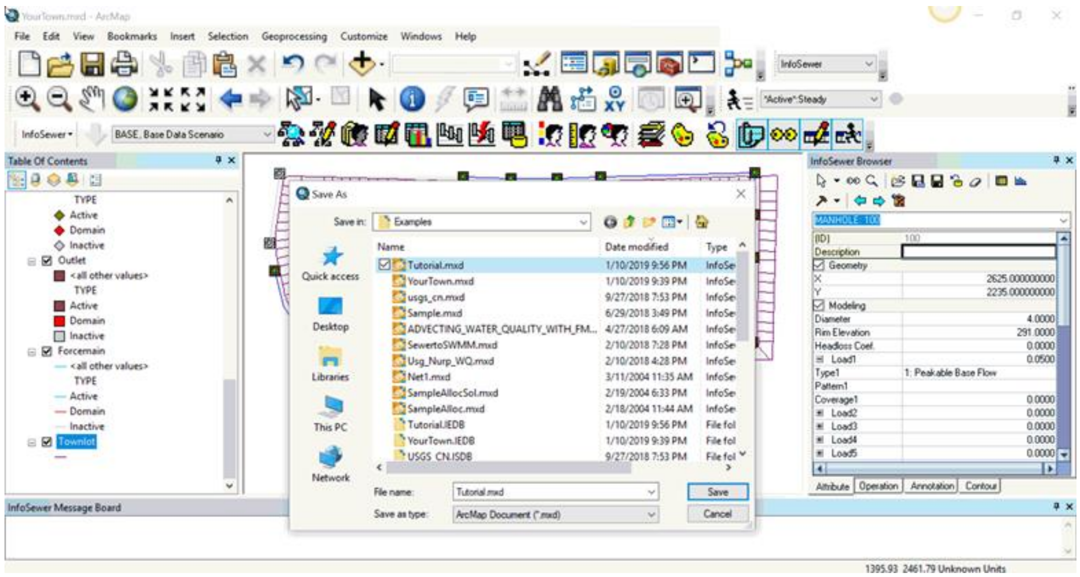
   C:\Users\Public\Documents\InfoSewer\Examples\Townlot.shp

(the path may be different for custom installations)



The parcel and the street layout will be added to the map as shown below.

5. Before continuing, save the “YourTown” project to a new project. If you wish to restart the tutorial, the original project will be available. Choose the *SAVE AS* command from the *FILE* menu. On the dialog box enter the new project name “Tutorial”. This becomes the active project.



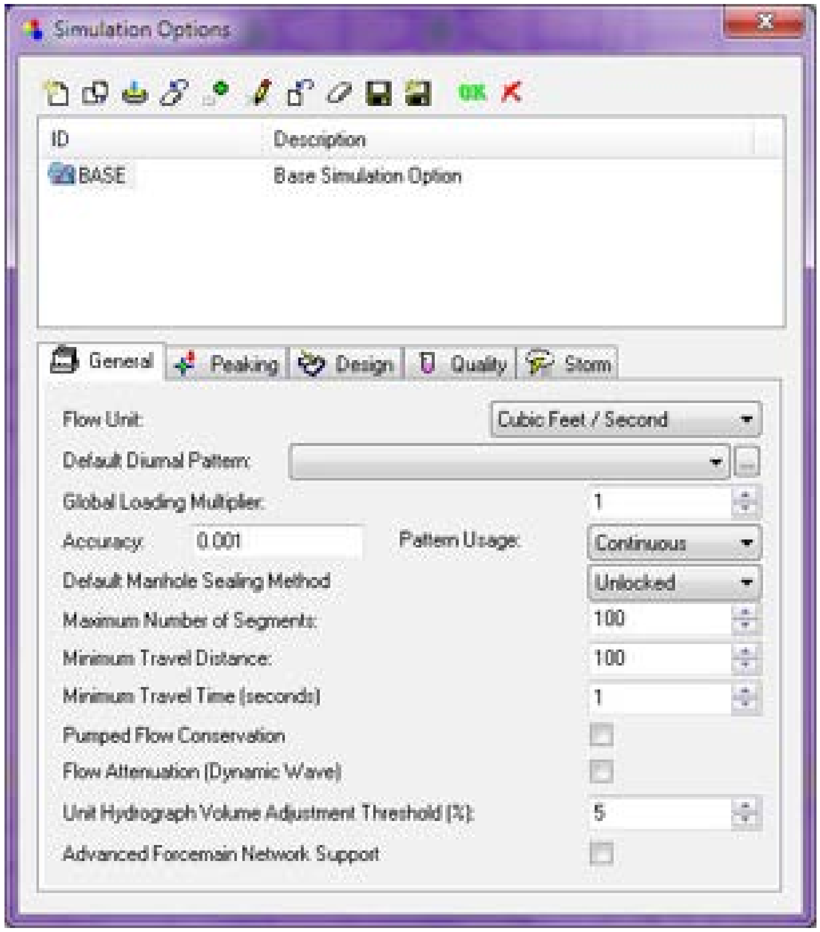
YourTown.mxd - ArcMap
File Edit View Bookmarks Insert Selection Geoprocessing Customize Windows Help
Table Of Contents
InfoSewer Browser
Save As
Save in: Examples
Name
Date modified
Type
Tutorial.mxd
YourTown.mxd
usgs_cn.mxd
Sample.mxd
SewertoSWMM.mxd
Usg_Nurp_WQ.mxd
Net1.mxd
SampleAllocSol.mxd
SampleAlloc.mxd
Tutorial.IEDB
YourTown.IEDB
File name: Tutorial.mxd
Save as type: ArcMap Document (*.mxd)
Save
Cancel
InfoSewer Message Board

Step 2: Define the Basic Options for the New Project

Before adding new components (pipes, pumps, manholes and wet wells) to the model, you should identify basic project units and hydraulic properties. In InfoSewer, two types of static hydraulic simulations can be analyzed. The first is a steady-state hydraulic analysis which applies a peaking factor to the collection system and determines system flows. The second is a steady-state hydraulic design simulation which evaluates the system and recommends new facilities where deficiencies exist.

In this step, you will identify the flow units and the peaking equation to be used in a steady-state model. These options are saved as a permanent part of the project. Skip Step 2 whenever the "**Tutorial**" project is opened for later use.

1. Select the *OPERATION DATA* tab on the InfoSewer Attribute Browser select the *SIMULATION OPTIONS* ® *BASE SIMULATION OPTIONS* and double-click. When the *SIMULATION OPTIONS* dialog box appears choose the *General* tab.



2. Choose "Cubic Feet / Second" from the *FLOW UNIT* drop-down list on the *SIMULATION OPTIONS* dialog box. Flow units are applied to link flows and to manhole loadings and define the units of your input data. "Cubic Feet / Second" indicates that your input data (elevations, diameters, etc.) will be in US Customary units.

3. Now select the *Peaking* tab and specify a peaking factor to be applied during a steady state simulation. In the *Peaking Factor K* field enter a value of "2.4" while in the *Peaking Factor p* field enter a value of "0.89". These two values will be utilized during a steady state model to determine peak flows traveling through a pipe.

4. Choose the "OK" button  to close the *SIMULATION OPTIONS* dialog box.

Step 3: Create Network Components

Create Network Components

The next step is to complete the network by adding new components and entering modeling data for those new network components (facilities). In this step, you will add components to the network and enter the critical attributes required for modeling into the project database. You will add five pipes, three manholes, one wet well, and one pump.

Use the illustration on page 3-2 and the tables below as a guide when creating the network.

WET WELL ATTRIBUTES

WET WELL ID	TYPE	BOTTOM ELEV. (ft)	INITIAL LEVEL (ft)	MIN. LEVEL (ft)	MAX. LEVEL (ft)	DIAM. (ft)
WW5	Cylindrical	265.00	0.00	0.00	18.00	15.00

MANHOLE ATTRIBUTES

MANHOLE ID	TYPE	DIAM. (ft)	RIM ELEVATION (ft)	BASE LOAD (cfs)	LOAD TYPE
46	Normal	3.00	288.00	0.05	Type 1: Peakable Base Flow
OUTLET110	Outlet	3.00	283.00	NA	NA
JC5	Chamber	3.00	274.00	NA	NA

PUMP ATTRIBUTES

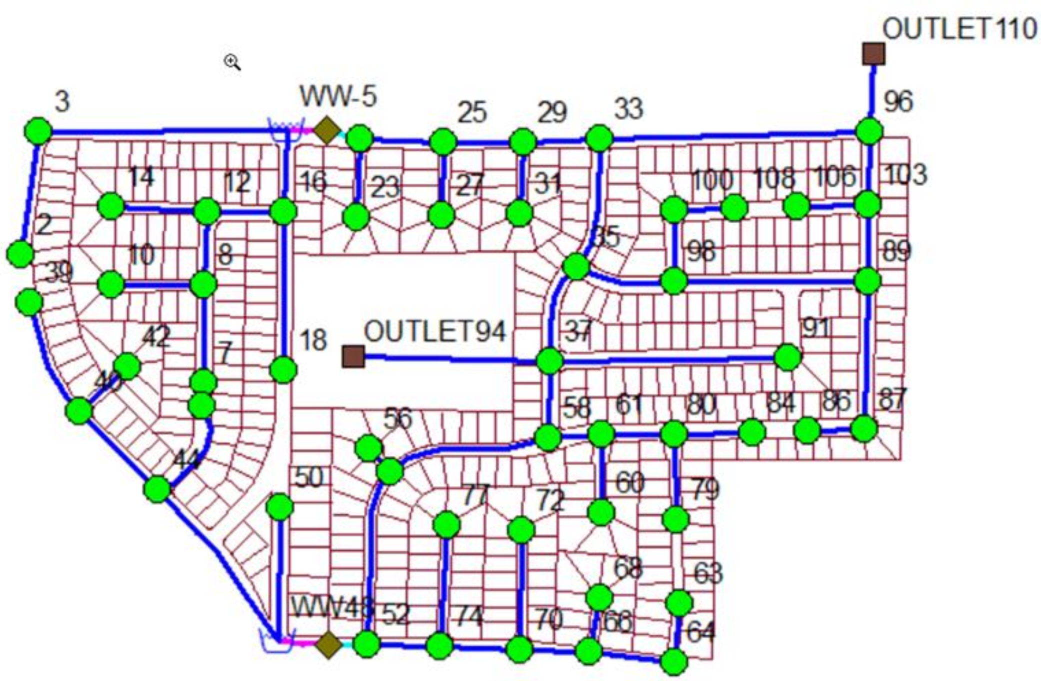
PUMP ID	FROM WW	TO MH	PUMP TYPE	CAPACITY (cfs)
P20	WW5	JC5	Type 0: Fixed Capacity	1.5

PIPE ATTRIBUTES

PIPE ID	FROM NODE	TO NODE	DIAM. (in)	LENGTH (ft)	FROM INVERT (ft)	TO INVERT (ft)	ROUGHNESS COEFFICIENT
4	3	WW5	6.00	870.00	283.00	280.00	0.013
19	16	WW5	10.00	265.00	281.00	280.00	0.013
FM20	JC5	21	6.00	240.00	278.00	290.00	130.00
45	46	44	6.00	200.00	283.00	280.00	0.013
109	96	110	12.00	170.00	281.00	278.00	0.013

NOTE: *Database fields for which no values are required are not illustrated here. Leave these fields empty when entering data.*

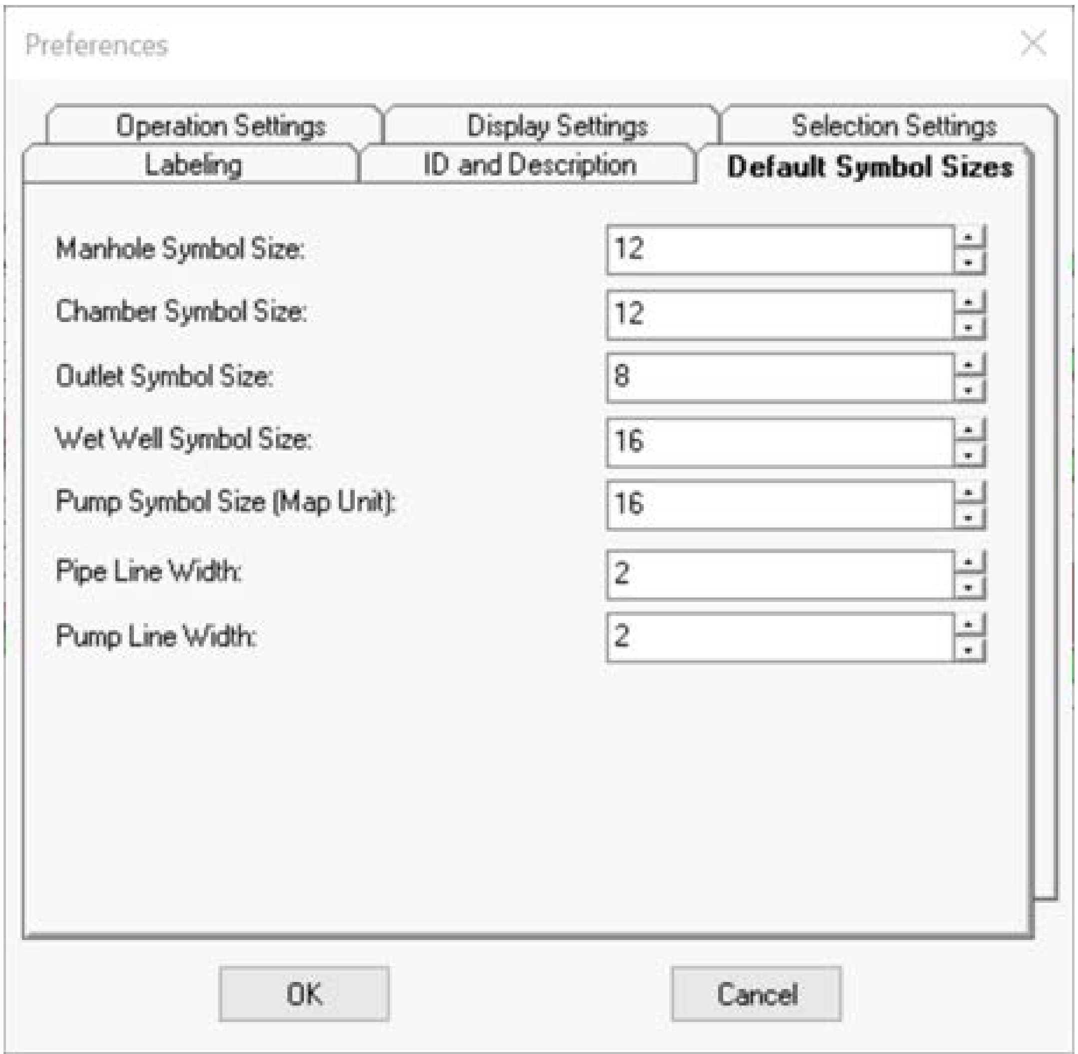
When creating the network, don't worry if it does not match the exact shape and dimensions of that illustrated on [page open project](); in tutorial Step 4 you will modify network geometry to better match the illustrated network representation.



Set Symbol Sizes for New Components

Before adding new components you should set the desired symbol sizes for those new components. InfoSewer stores and displays symbols in relative scale or in font size. To change your default symbol sizes, perform the following:

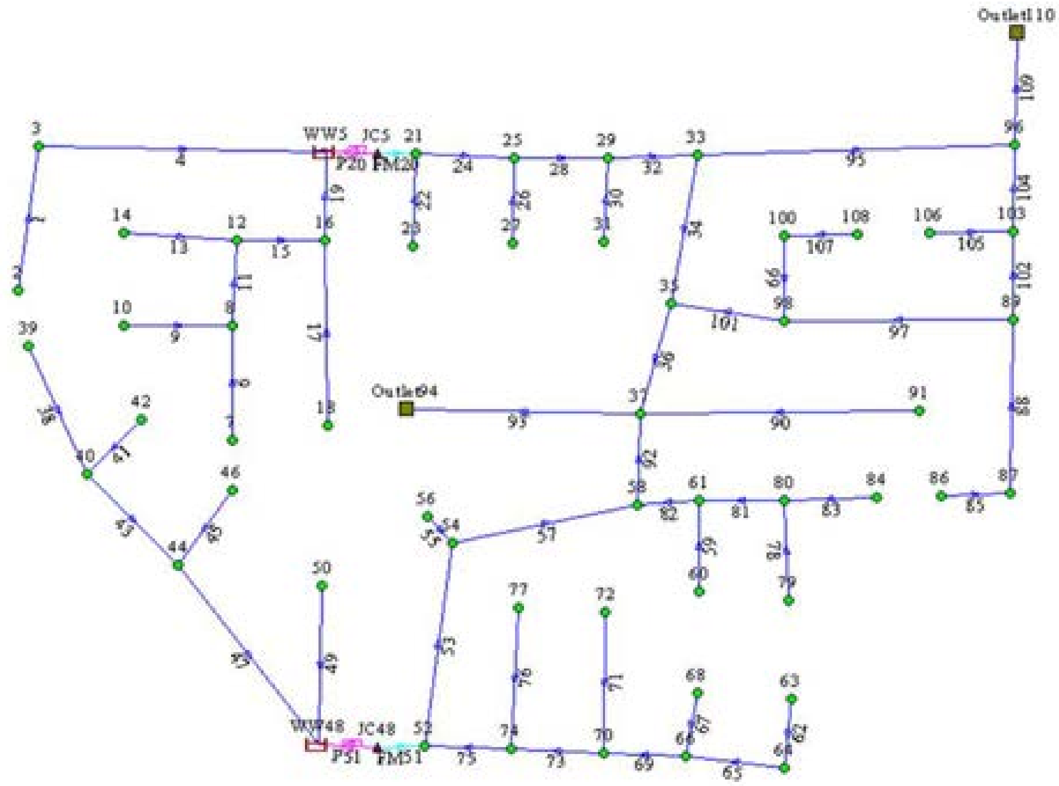
1. Select the *PREFERENCES* command from the *TOOLS* menu. When the *PREFERENCES* dialog box appears on the screen, select the *SYMBOLS* tab as shown below.



2. Set the above symbol sizes. Press the "Apply" button followed by the "OK" button to close the *PREFERENCES* dialog box. You can also choose a different color for each component by clicking on the corresponding color box.

Adding a Wet Well

You will start by digitizing the wet well WW5, located at the top middle of the model (use the following figure for graphical reference).



1. Select the *ADD WET WELL* command from the *CREATE* menu. Move the cursor to the desired location and press the left mouse button.

2. You are then prompted to enter the identifier and an optional description for the new wet well. Enter the *WET WELL ID* WW-5 as shown below, using a comma to separate the ID from the description.

3. Click on the "OK" button to save the information. The wet well should now appear on the screen as a brown tank symbol.

4. In the *ATTRIBUTE BROWSER-NODE* window on the right, perform the following:

· Choose “0: Cylindrical” in the *TYPE* field.

· Enter “265” in the *BOTTOM ELEVATION* field.

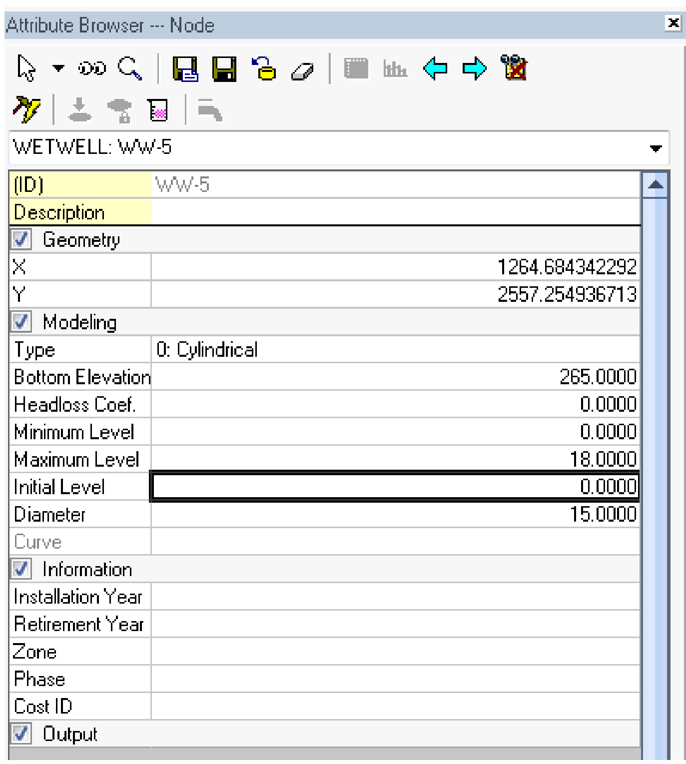
· Enter “0.00” in the *MINIMUM LEVEL* field.

· Enter “18.00” in the *MAXIMUM LEVEL* field.

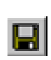
· Enter “0.00” in the *INITIAL LEVEL* field.

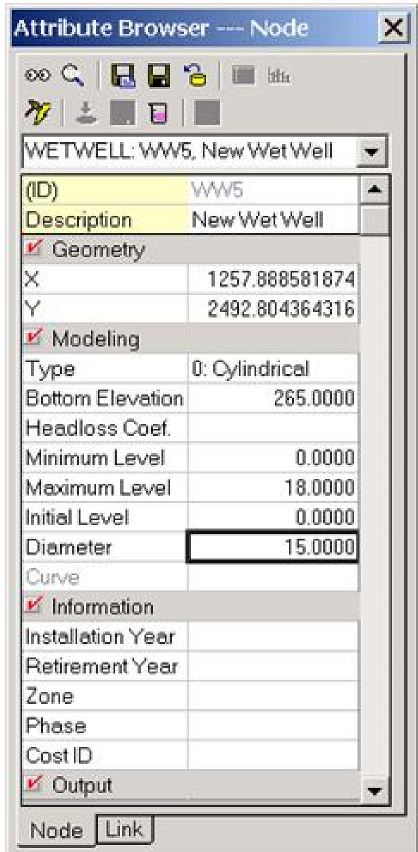
· Enter “15.00” in the *DIAMETER* field.

· Leave all other fields blank.



NOTE: You can use the Enter key, the Tab key or the down arrow key to enter data and move down one field. The Up arrow key can be used to move up one field.

5. Click once on the “Save” button  and the *ATTRIBUTE BROWSER-NODE* window should appear as shown below. Please note that you may have different values for the geometry coordinates.

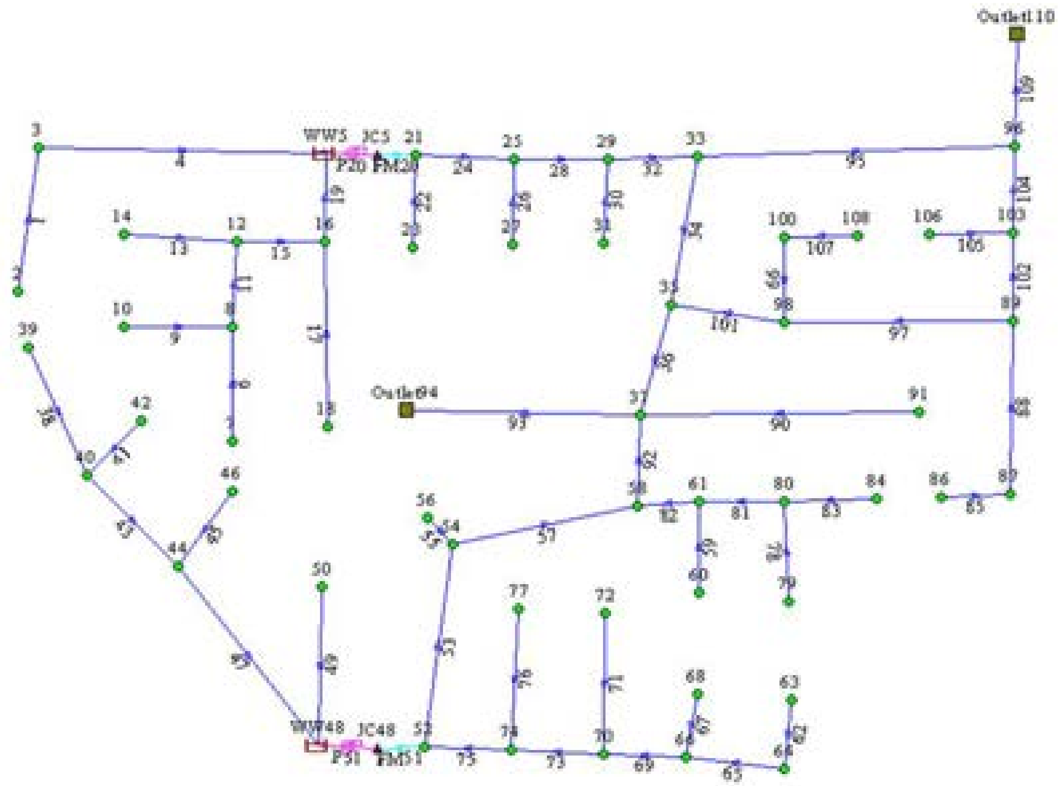


Refer to Data Elements à Wet Well topic in the on-line InfoSewer Help for more information on wet wells.

Adding a Manhole

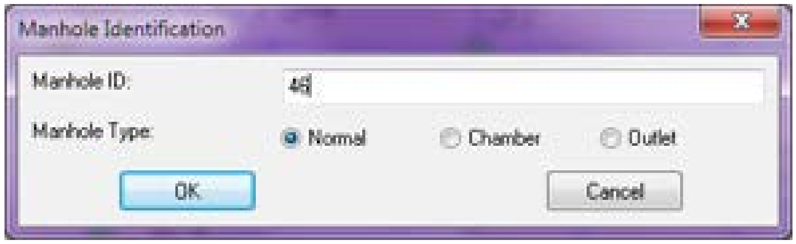
Adding A Normal Manhole

Now digitize manhole ID 46 in the lower left-hand corner of the network ((use the following figure for graphical reference).



1. Select the *ADD/INSERT MANHOLE* command from the *CREATE* menu. Move the *cursor to the lower left-hand corner of the network (just below of manhole ID 7) and* press the left mouse button.

2. You are then prompted to enter the identifier and an optional description for the new node. Enter the *MANHOLE ID* 46 as shown below, using a comma to separate the ID from the description. Select “Normal” as the manhole type.



3. Click on the "OK" button. Manhole 46 should now appear on the screen as a green circle.

4. In the *ATTRIBUTE BROWSER-NODE* window on the right, perform the following:

· Enter "3.00" in the *DIAMETER* field.

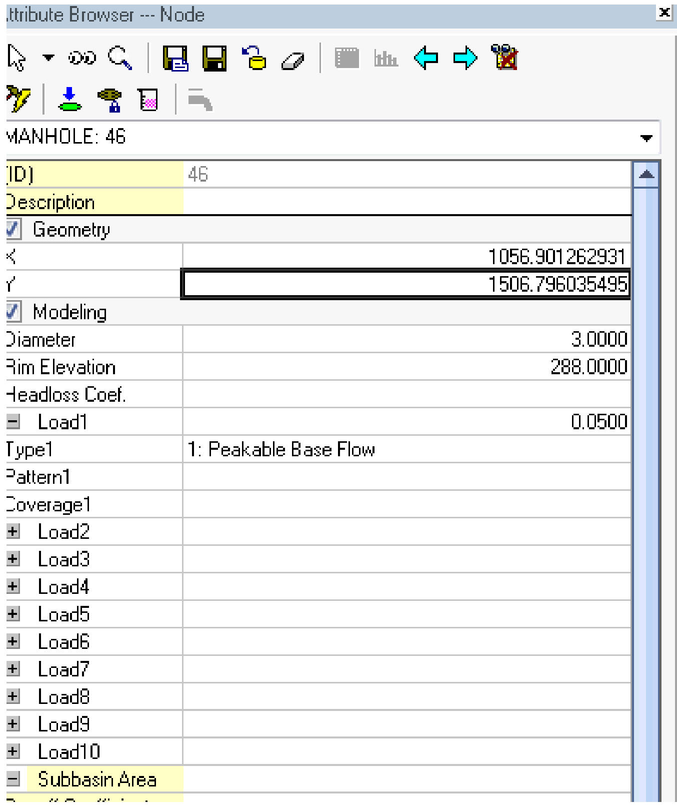
· Enter "288.00" in the *RIM ELEVATION* field.

· Enter "0.05" in the *LOAD1* field and enter "2" in the *ZONE* field.

· Select "1: Peakable Base Flow" in the *TYPE1* field.

· Leave all other fields blank.

5. Click once on the "Save" button  and the *ATTRIBUTE BROWSER-NODE* window should appear as shown below. Please note that you may have different values for the geometry coordinates.



NOTE: Under Tools->Preferences, if the “Auto Record Saving” option is enabled, then you do not need to explicitly click on the “Save” button.

Adding A Chamber Manhole

Chamber manholes represent artificial manholes that separate pumps and force mains. Digitize chamber manhole ID JC5 in the upper left-hand corner of the network (use the figure at top for graphical reference).

1. Select the *ADD/INSERT MANHOLE* command from the *CREATE* menu. Move the cursor to the right of wet well WW5 and press the left mouse button.

2. You are then prompted to enter the identifier and an optional description for the new node. Enter the *MANHOLE ID* as shown below, using a comma to separate the ID from the description. Select "Chamber" as the manhole type.

3. Click on the "OK" button. Chamber manhole JC5 should now appear on the screen as a mauve triangle.

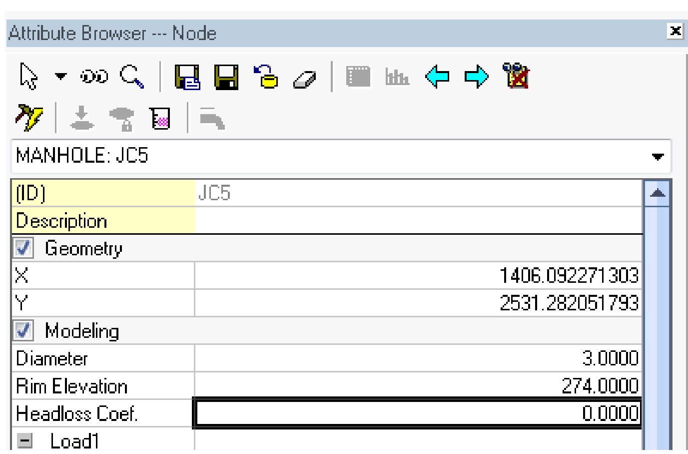
4. In the *ATTRIBUTE BROWSER-NODE* window on the right, perform the following:

· Enter "3.00" in the *DIAMETER* field.

· Enter "274.00" in the *RIM ELEVATION* field.

· Leave all other fields blank.

5. Click once on the "Save" button  on the *ATTRIBUTE BROWSER-NODE* window.



Adding An Outlet Manhole

Now digitize outlet manhole ID OUTLET110 in the upper right-hand of the network (use page 3-2 for graphical reference).

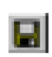
1. Select the *ADD/INSERT MANHOLE* command from the *CREATE* menu. Move the cursor to the upper right-hand corner of the network and press the left mouse button.
2. You are then prompted to enter the identifier and an optional description for the new node. Enter the *MANHOLE ID* as shown below, using a comma to separate the ID from the description. Select “Outlet” as the manhole type.

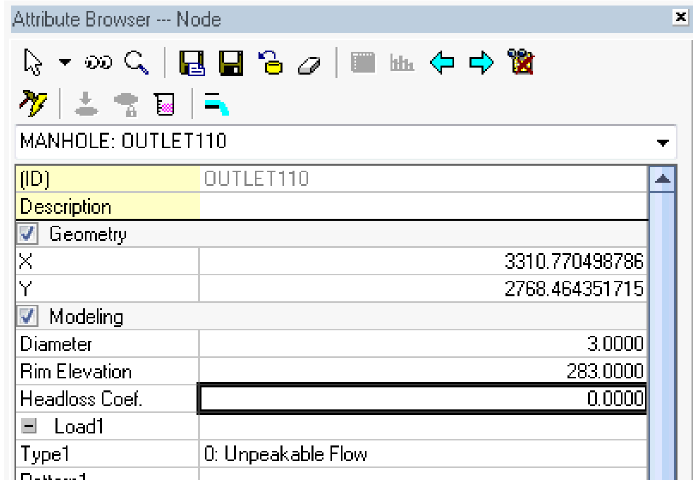
3. Click on the “OK” button. Outlet manhole OUTLET110 should now appear on the screen as a dark green square.
4. In the *ATTRIBUTE BROWSER-NODE* window on the right, perform the following:

· Enter “3.00” in the *DIAMETER* field.

· Enter “283.00” in the *RIM ELEVATION* field.

· Leave all other fields blank.

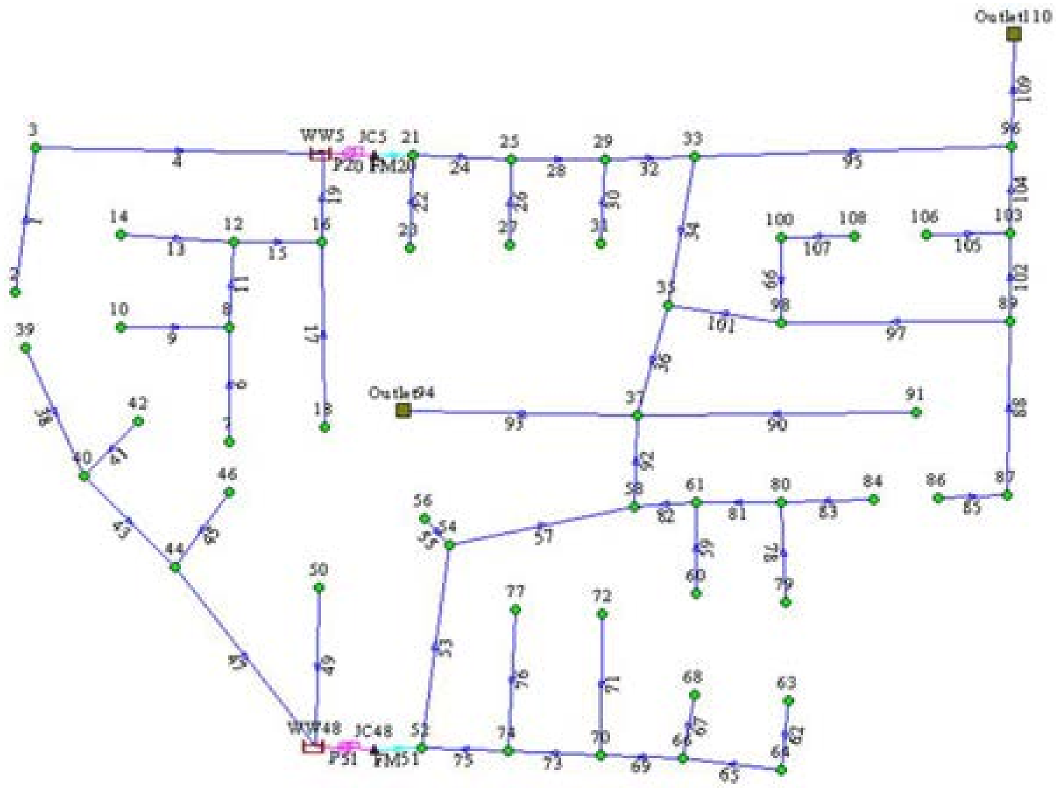
5. Click once on the “Save” button  on the *ATTRIBUTE BROWSER-NODE* window.



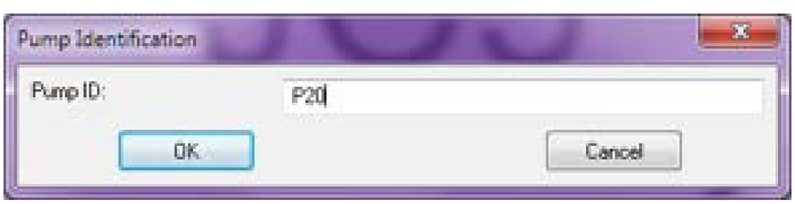
Refer to the Data Elements à Manholes topic in the on-line InfoSewerHelp for more information on manholes.

Adding a Pump

You will now add pump P20 between wet well WW5 and chamber manhole JC5((use the following figure for graphical reference) . The order that you pick the nodes is critical; flow can only move through pumps from a wet well to a chamber manhole.



1. Select the *ADD PUMP* command from the *CREATE* menu. Digitize the pump geometry as shown in the above figure. Using your mouse, left click on the upstream wet well WW5 then double-click on the chamber manhole JC5 to create the geometry for pump P20.



2. You are then prompted to enter the pump identifier and an optional description for the new pump. Enter “P20, New Pump” in the *PUMP ID* field.

3. Click on the “OK” button to save the information. Pump P20 should now appear as a magenta line with a magenta pump symbol. The pump symbol is oriented in the direction of the pump.

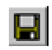
4. In the *ATTRIBUTE BROWSER-LINK* window on the right, perform the following:

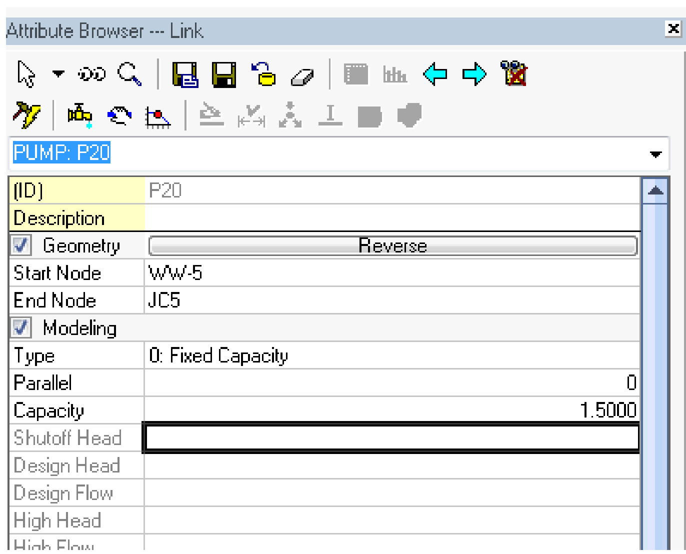
· Choose “0: Fixed Capacity” in the *TYPE* field.

· Enter “0.00” in the *PARALLEL* field.

· Enter “1.50 in the *CAPACITY* field.

· Leave all other fields blank.

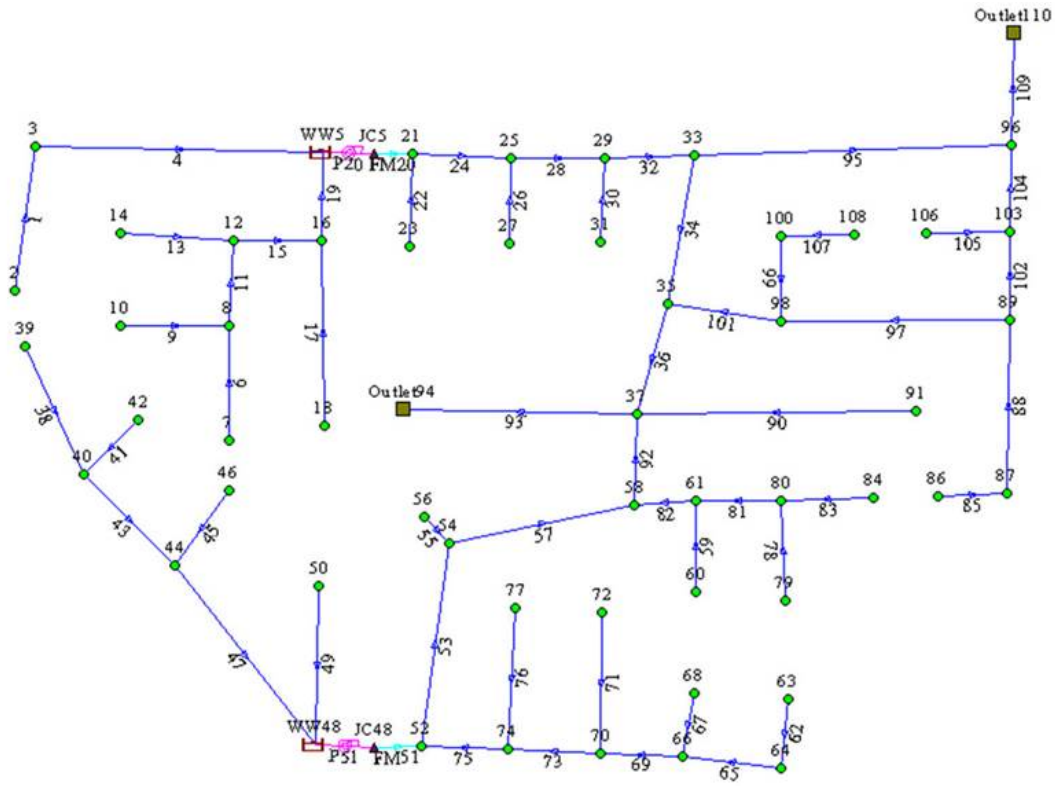
5. Click once on the “Save” button  and the *ATTRIBUTE BROWSER-LINK* window should appear as shown below.



Adding Pipes

Adding Gravity Pipes

Digitize pipes 4, 19, 45 and 109 starting with pipe 45 at the bottom left-hand corner of the network, adjacent to manhole 46 ((use the following figure for graphical reference).



1. Select the *ADD PIPE* command from the *CREATE* menu. Using your mouse, left click on manhole 46, add intermediate vertices as necessary, then double-click on manhole 44 to complete pipe 45.

2. You are then prompted to enter the pipe identifier and an optional description for the new pipe. Enter "45" in the *PIPE ID* field. No description is necessary.
3. Click on the "OK" button. Pipe 45 should now appear as a blue line connecting manholes 46 and 44.

4. In the *ATTRIBUTE BROWSER-LINK* window on the right, perform the following: (use Pipe Attribute Table reference)

· Enter “283.00” in the *FROM INVERT* field.

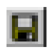
· Enter “280.00” in the *TO INVERT* field.

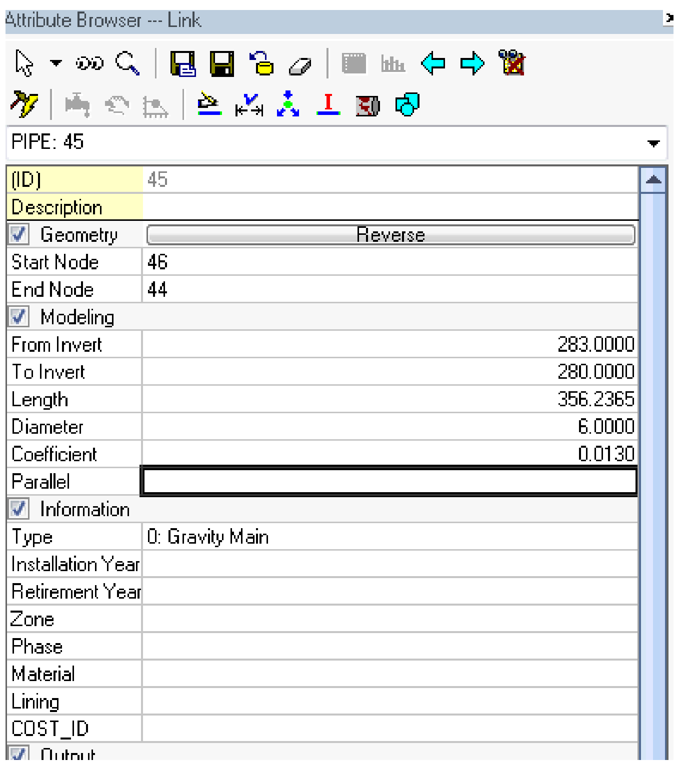
· Enter “6.00” in the *DIAMETER* field.

· Enter “0.013” in the *COEFFICIENT* field.

· Enter “0” in the *PARALLEL* field.

· Leave all other fields blank.

5. Leave the *LENGTH* field blank as you will edit it later in the tutorial. Click once on the “Save” button  and the *ATTRIBUTE BROWSER-LINK* window should appear as shown below.



Enter the location and attributes for pipes 4, 19, and 109 in a similar fashion as pipe 45. Populate the remaining *Modeling* section with the data provided in the Pipe Attribute Table reference). Note that all the pipes you added are circular types. InfoSewercan also model non-circular channels.

Adding a Force Main

Digitize force main ID FM20 between the chamber manhole and the node downstream of the lift station ((use the above figure for graphical reference).

1. Select the *ADD PIPE* command from the *CREATE* menu. Using your mouse, left click on chamber manhole JC5, add intermediate vertices as necessary, then double-click on manhole 21 to complete force main **FM20**.

2. You are then prompted to enter the pipe identifier and an optional description for the new pipe. Enter "FM20, Force Main" in the *NEW LINK ID* field.

3. Click on the "OK" button. Force main 20 should now appear as a cyan line connecting chamber manhole JC5 with manhole 21.

4. In the *ATTRIBUTE BROWSER-LINK* window on the right, perform the following: (use Pipe Attribute Table on page 3-6 for reference)

· Enter "278.00" in the *FROM INVERT* field.

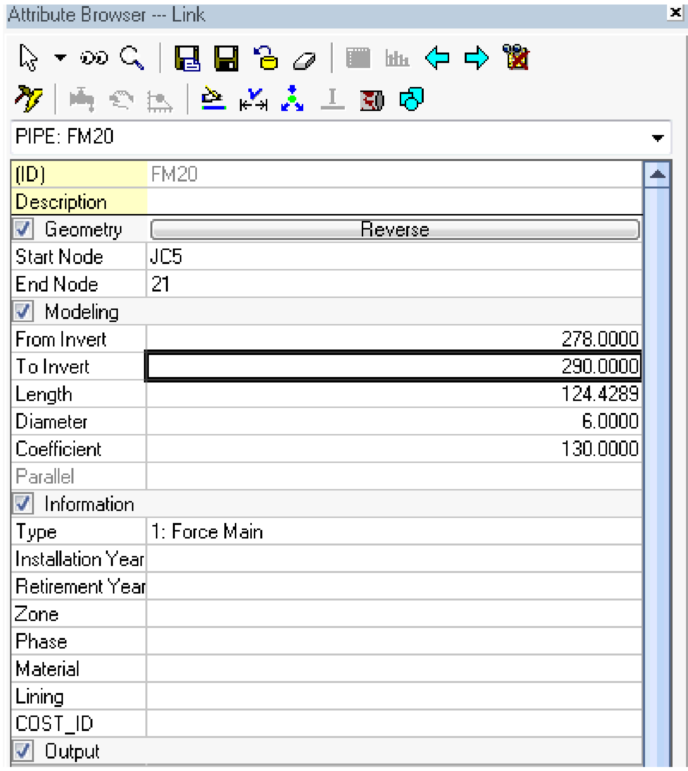
· Enter "290.00" in the *TO INVERT* field.

· Enter "6.00" in the *DIAMETER* field.

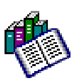
· Enter "130.00" in the *COEFFICIENT* field.

· Enter "0.00" in the *PARALLEL* field.

· Leave all other fields blank.



5. Leave the *LENGTH* field as you will edit it later in the tutorial. Click once on the “Save” button  on the *ATTRIBUTE BROWSER-LINK* window.



Refer to the Data Elements à Pipes topic in the on-line InfoSewer Help for more information on pipes and force mains.

You have probably noticed that nodes (manholes, wet wells, chambers and outlets) must be added to the project prior to adding the links connecting them. Choosing the ADD/INSERT MANHOLE and ADD WET WELL commands before adding pipes and pumps can therefore slow the network creation process when you have a large number of components to add.

InfoSewer provides a shortcut allowing you to create network component of all types at one time. You can use the *DIGITIZE NETWORK* command from the *CREATE* menu to digitize nodes and links in series, eliminating the need to keep making separate choices. *DIGITIZE NETWORK* provides a method for data entry where you are not required to first enter the nodes prior to entering links. Use of the *DIGITIZE NETWORK* command is entirely optional, but is recommended for bulk data entry.

To insert a manhole on an existing pipe (i.e. breaking that pipe in two), select first the pipe to be broken using the “Select” button  and then *ADD/INSERT MANHOLE* command from the *CREATE* menu.

To modify the attributes of any network component, use the *ATTRIBUTE BROWSER* window.

Step 4: Modify the Network Configuration

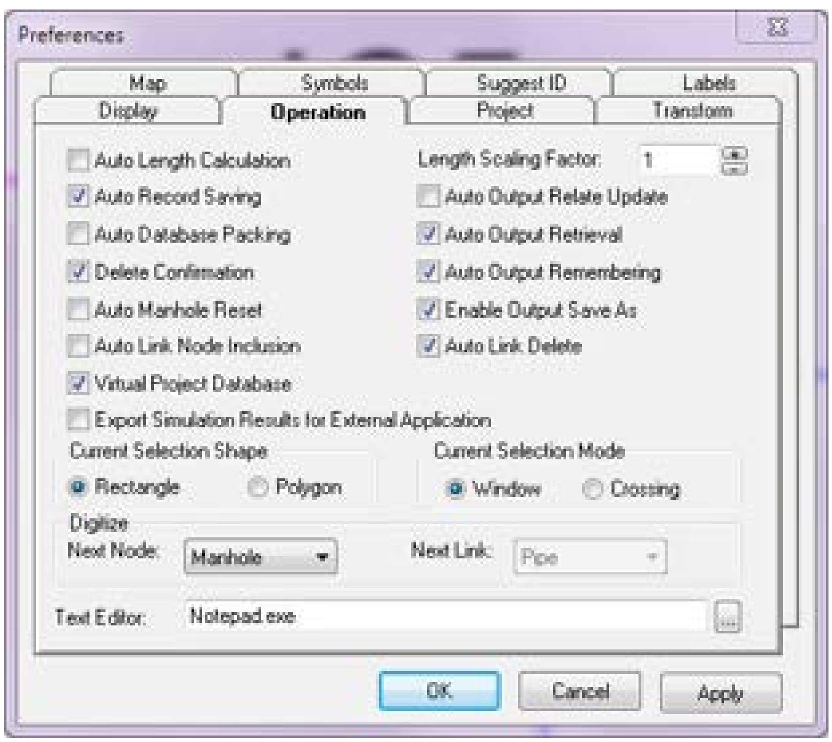
Turning off Auto Length Calculation

Before modifying the network, you should turn off the AUTO LENGTH CALCULATION preference. When this setting is active, InfoSewer automatically calculates pipe lengths based on the drawing scale when pipes are added or edited. When this preference is inactive, user-specified pipe lengths will be preserved regardless of the length of the pipe on the map display. To disable the AUTO LENGTH CALCULATION preference, perform the following:

1. Select the PREFERENCES command from the TOOLS menu.

2. Choose the OPERATION tab and disable the AUTO LENGTH CALCULATION preference by clicking on the check-box so that no check appears next to the prompt. Leave all other preferences in their current state.

3. Click on the "Apply" button and the "OK" button to close the window.

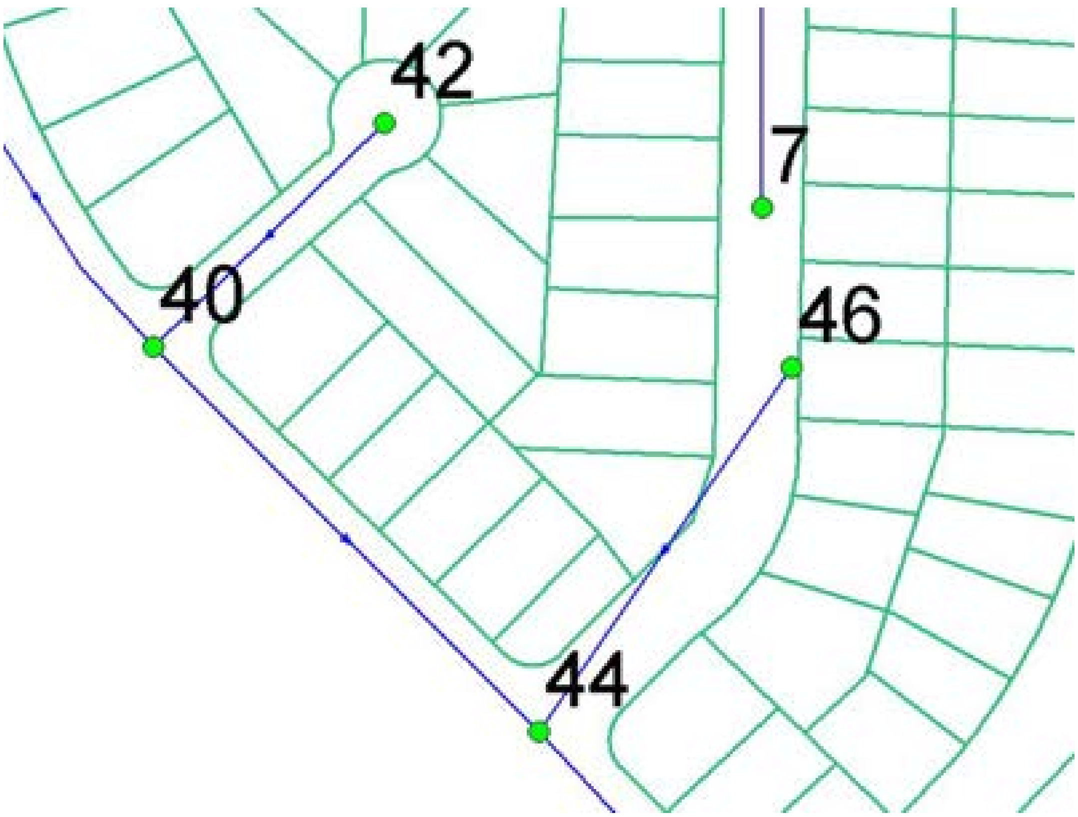


If you added any links in Step 3 above with a curvilinear representation, you can now adjust any intermediate shape-defining points for those links. You will now re-digitize pipe 45 to better match the shape of the street under which the pipe lies. First note the length of the pipe before the edit.

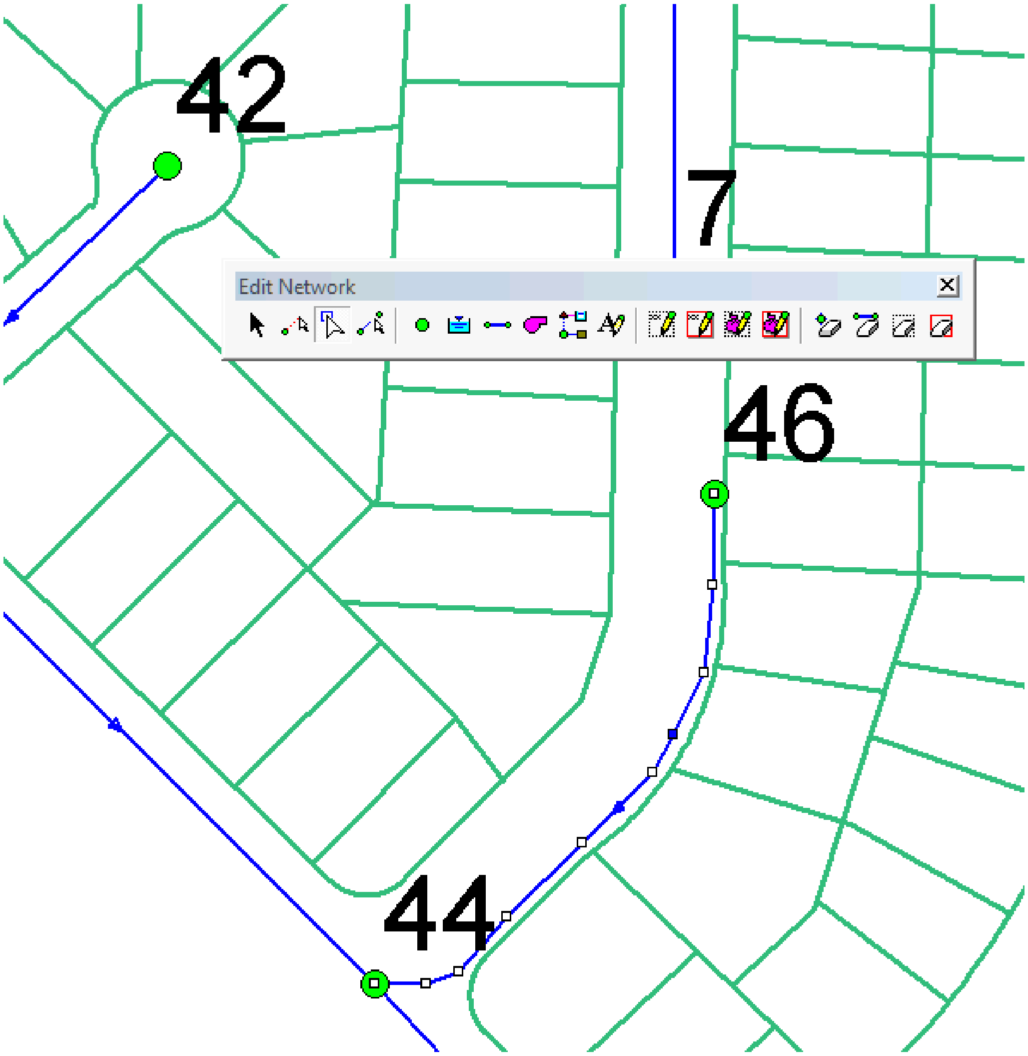
1. Select the *REDRAW LINK* command from the *EDIT* menu to redraw the pipe.
2. Using your mouse left click on the start node and then left click to define intermediate vertices. Finally, double-click on the end node to terminate the redrawing process.

From the *ATTRIBUTE BROWSER-LINK* window, note that the length of the pipe in the *Modeling* section remains unchanged as the *AUTO LENGTH CALCULATION* preference is OFF (disabled). Alternatively, you can edit the vertices by using the *EDIT LINK VERTEX* command from the *EDIT* menu.

Before



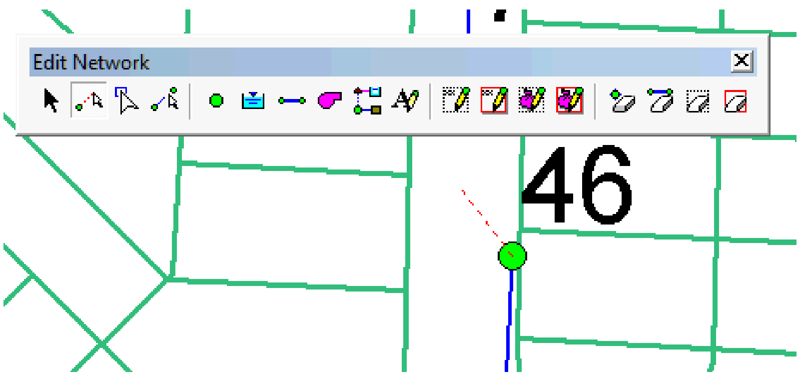
After Redrawing Link 45


42
7
Edit Network
46
44

Moving Manholes

Now you can adjust the location of one or more manholes so that your network more accurately matches the diagram illustrated on the previous pages of this tutorial. Note that when you move a manhole, InfoSewer will also stretch the links connected to that manhole. If the Auto Length Calculation preference is “ON” (it is not at this point in the tutorial), InfoSewer will re-calculate the lengths of all connecting pipes. To move a node, perform the following:

1. From the *EDIT* menu, select the *MOVE NODE* command and left-click to select the manhole to be moved. Place the mouse cursor on the desired manhole and then press and hold the left mouse button.
2. Drag the manhole to the desired location and release the left mouse button to indicate the new location.
3. Repeat the above two steps to continue to move the manhole as necessary. Moving Manholes

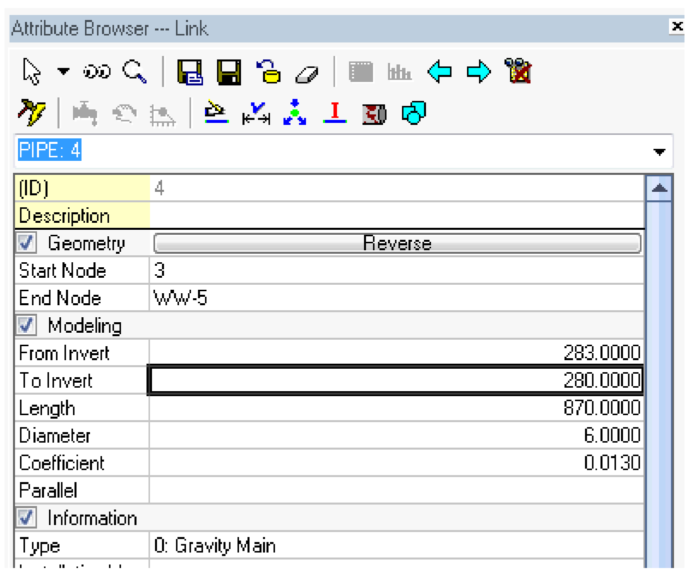


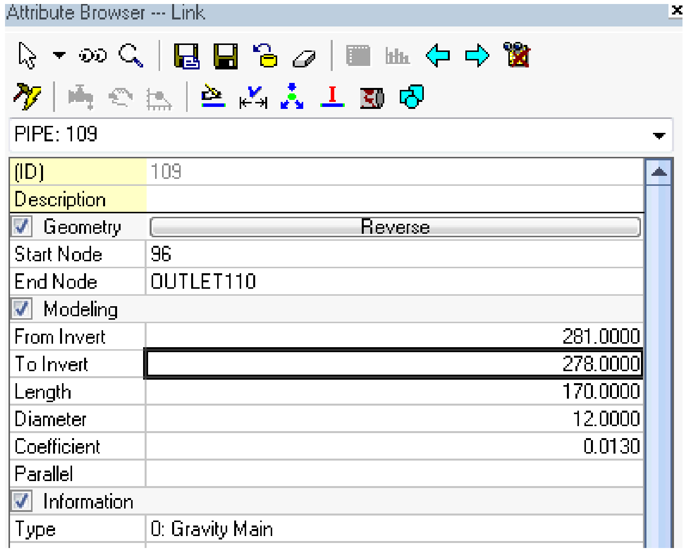
Step 5: Modify Component Attributes

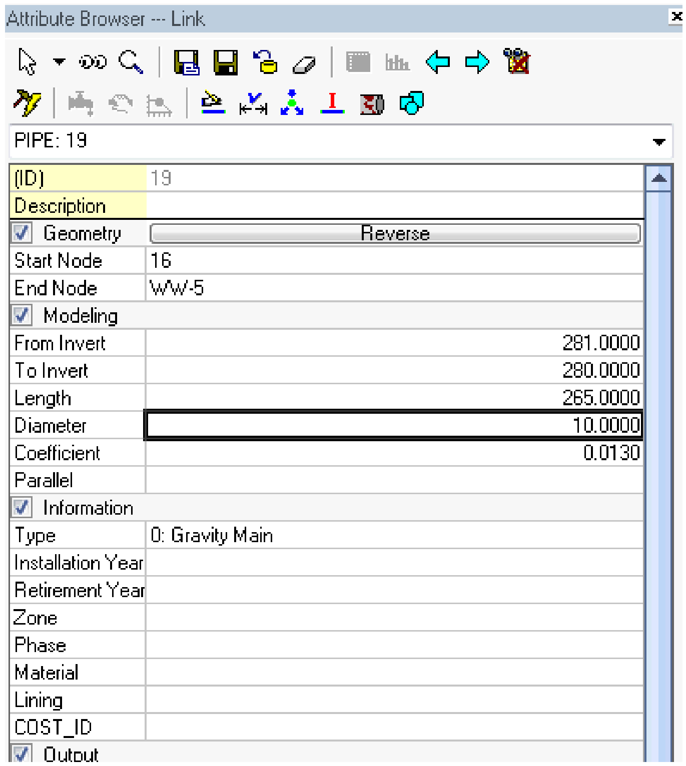
Modify Elements One at a Time

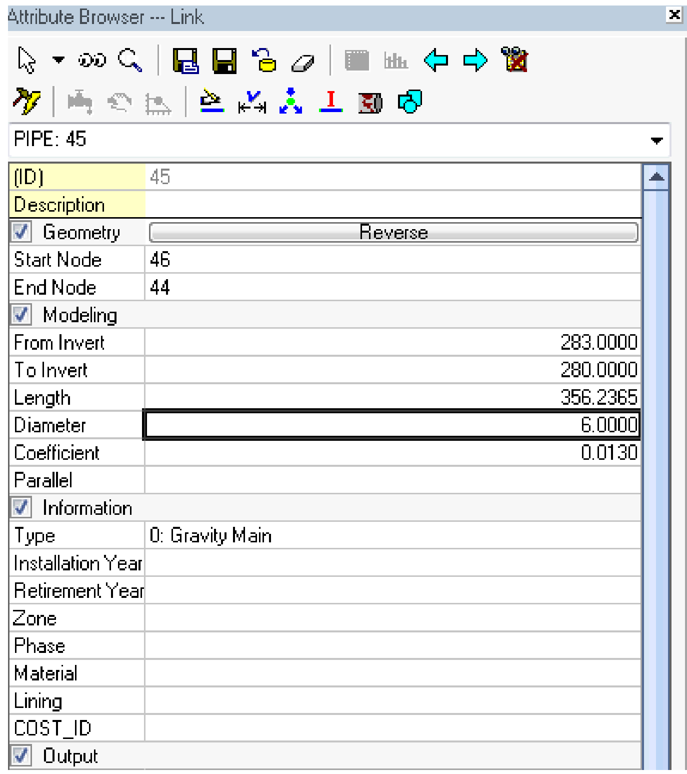
When entering pipes earlier in the tutorial, InfoSewer had automatically assigned their lengths. Now, using information from table enter pipe lengths for pipes 4, 19, FM20, 45 and 109. Perform the following for each pipe:

1. Click on the "Select" button, place the cursor on pipe with ID 4 and then press the left mouse button.
2. In the *Modeling* section of the *ATTRIBUTE BROWSER-LINK* window, change the length (as computed by InfoSewer) to 870 ft as indicated here.
3. Click once on the "Save" button and repeat the process for the four other pipes.




Attribute Browser --- Link
PIPE: 109
(ID) 109
Description
Geometry Reverse
Start Node 96
End Node OUTLET110
Modeling
From Invert 281.0000
To Invert 278.0000
Length 170.0000
Diameter 12.0000
Coefficient 0.0130
Parallel
Information
Type 0: Gravity Main


Attribute Browser --- Link
PIPE: 19
(ID) 19
Description
Geometry Reverse
Start Node 16
End Node WW-5
Modeling
From Invert 281.0000
To Invert 280.0000
Length 265.0000
Diameter 10.0000
Coefficient 0.0130
Parallel
Information
Type 0: Gravity Main
Installation Year
Retirement Year
Zone
Phase
Material
Lining
COST_ID
Output


Attribute Browser --- Link
PIPE: 45
(ID) 45
Description
Geometry
Reverse
Start Node 46
End Node 44
Modeling
From Invert 283.0000
To Invert 280.0000
Length 356.2365
Diameter 6.0000
Coefficient 0.0130
Parallel
Information
Type 0: Gravity Main
Installation Year
Retirement Year
Zone
Phase
Material
Lining
COST_ID
Output

Group Editing with the DB Editor

The *DB EDITOR* is used to maintain InfoSewer database tables. The *DB EDITOR* is also used to customize select tables by adding user-defined fields, modifying, and deleting table information in a spreadsheet fashion. Using the *DB EDITOR* you will globally adjust the friction coefficients for all pipes in the "**Tutorial**" project.

1. Select the *DB TABLES* command from the *EDIT* menu or click once on the "DB Editor" button . The *DB EDITOR* appears on the screen.

2. Under the *ELEMENT HYDRAULIC DATA* folder, select the *PIPE HYDRAULIC (MODELING) DATA* table. At the bottom of the dialog box, select *ENTIRE TABLE* in the *DISPLAY SCOPE* area (indicating you want the entire contents of the table to be displayed and available for editing), and then click the "OK" button. The *PIPE HYDRAULIC (MODELING) DATA* table appears on the screen.

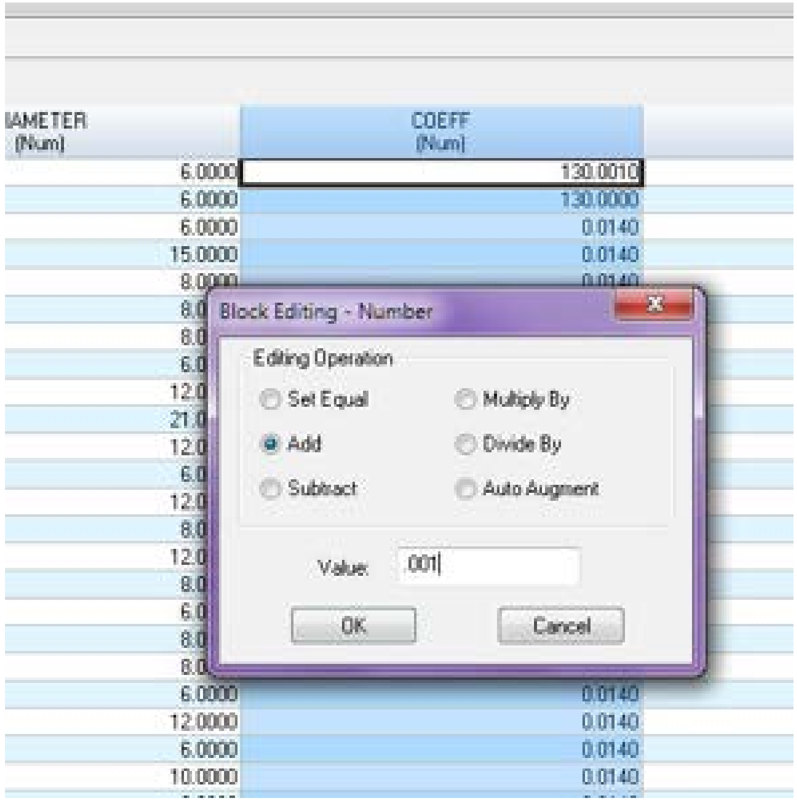
3. To adjust the contents of an entire field, in this case *COEFF*, click once on the column header with the mouse. The entire column should be highlighted. (Note: Force mains are also included in this table. In H_2OMAP Sewer/Pro, gravity pipes are evaluated using the Manning equation while pipes under pressure are evaluated using the Hazen-Williams equation.)

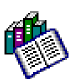
4. You will adjust all roughness coefficients by increasing all current coefficients by 0.001. Choose the "Block Editing" button while the *COEFF* field remains highlighted. When the *BLOCK EDITING* dialog box appears on the screen, choose the *ADD* operation, enter "0.001" for the value to be added, then choose "OK". All roughness coefficients should now be increased by 0.001. (Note: Force mains also increased their Hazen-Williams coefficient by 0.001).

5. Select the "Save" button and then choose the "Close" button at the top of the *DB EDITOR* window.

6. Close the *DB EDITOR* by selecting the "Exit" button at the top of the *DB EDITOR* window.

7. Save the project! This would be a good time to save your work. Choose the *SAVE* command from the *FILE* menu. This will save the current state of the InfoSewerproject data. It is recommended that you save the project after all subsequent tutorial steps.

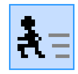


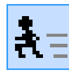
 Refer to the Database Management à DB Editor topic in the on-line InfoSewer Help for more information on modifying InfoSewer table structures and data.

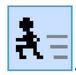
Step 6: Running a Steady State Analysis Simulation

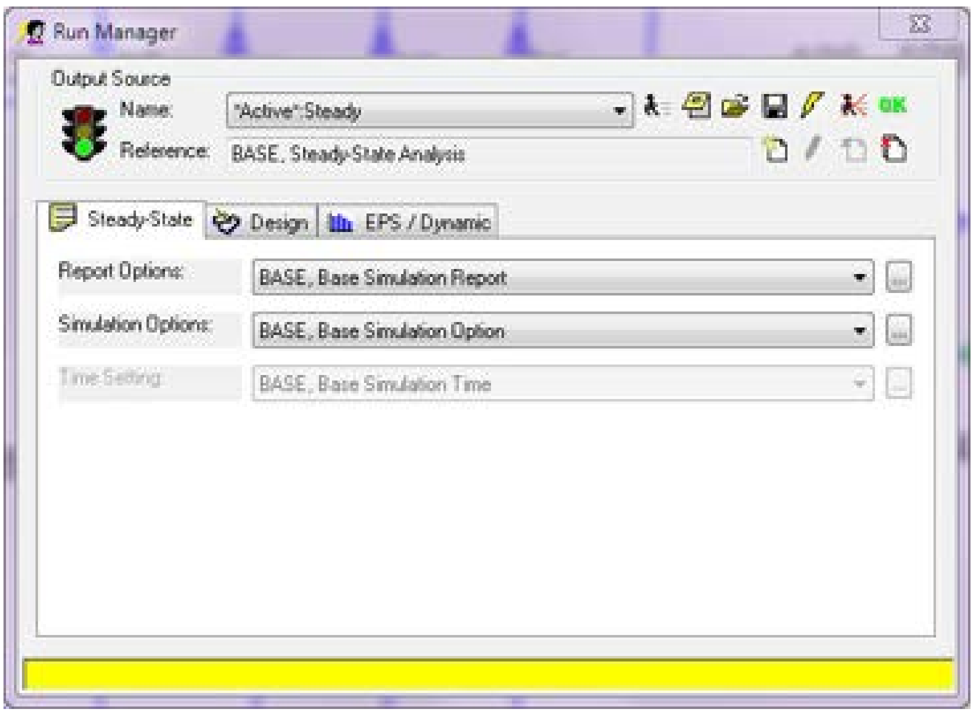
Run Steady State Simulation

To run a steady state simulation, perform the following:

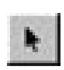
- **1.** Choose the *RUN MANAGER* command  from the *TOOLS* menu.

  The *RUN MANAGER* dialog box appears on the screen. Choose the *STEADY-STATE* tab.

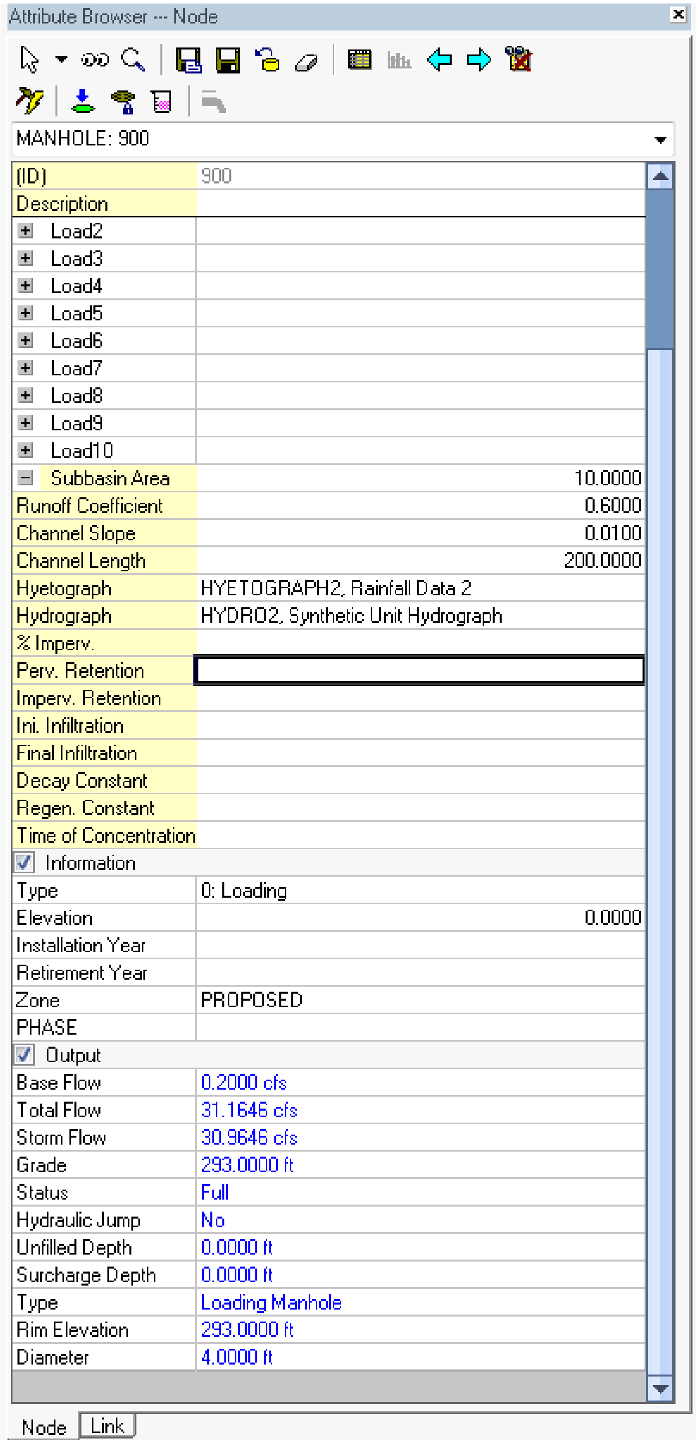
- **2.** Click on the “Run” button . Upon successful completion of the simulation, the Output Source Status Indicator (the stoplight) should show green.



Review the steady state analysis results using the *ATTRIBUTE BROWSER* window one network component at a time. To do this, perform the following:

- Click on the “Select” button  and then click on the desired node or link. Results for that node or link are shown in the *Output* section of the *ATTRIBUTE BROWSER* window.

The model reports storm flows for the pipes and the loading manholes. Run Steady State Simulation



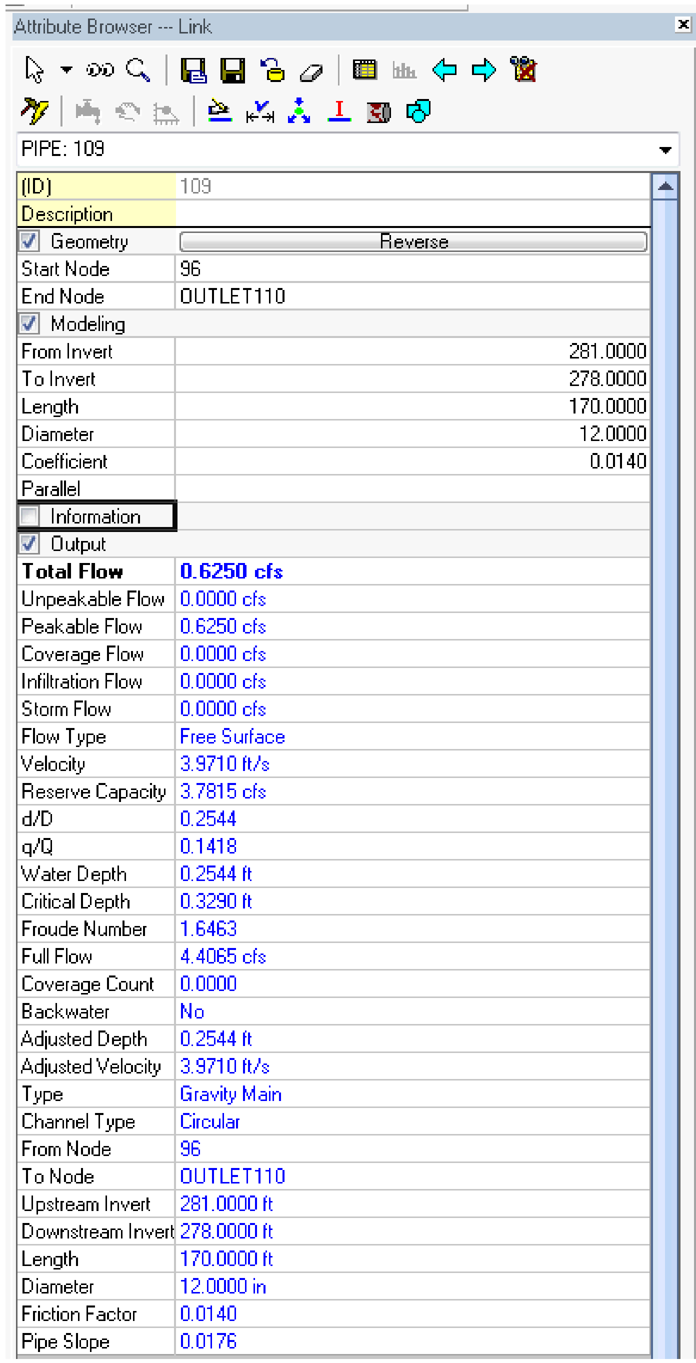
Attribute Browser --- Node
MANHOLE: 900
(ID) 900
Description
Load2
Load3
Load4
Load5
Load6
Load7
Load8
Load9
Load10
Subbasin Area 10.0000
Runoff Coefficient 0.6000
Channel Slope 0.0100
Channel Length 200.0000
Hyetograph HYETOGRAPH2, Rainfall Data 2
Hydrograph HYDRO2, Synthetic Unit Hydrograph
% Imperv.
Perv. Retention
Imperv. Retention
Ini. Infiltration
Final Infiltration
Decay Constant
Regen. Constant
Time of Concentration
Information
Type 0: Loading
Elevation 0.0000
Installation Year
Retirement Year
Zone PROPOSED
PHASE
Output
Base Flow 0.2000 cfs
Total Flow 31.1646 cfs
Storm Flow 30.9646 cfs
Grade 293.0000 ft
Status Full
Hydraulic Jump No
Unfilled Depth 0.0000 ft
Surcharge Depth 0.0000 ft
Type Loading Manhole
Rim Elevation 293.0000 ft
Diameter 4.0000 ft
Node
Link

Step 7: Review Model Results

Review Results with the Attribute Browser (AB)

First you will use the *ATTRIBUTE BROWSER* window to review model results one network component at a time. To do this, perform the following:

1. Click on the “Select” button  and then click on the desired node or link. Results for that node or link are shown in the *Output* section of the *Attribute Browser RESULTS WITH THE Attribute Browser* window.



Attribute Browser --- Link

PIPE: 109

(ID)	109
Description	
Geometry	Reverse
Start Node	96
End Node	OUTLET110
Modeling	
From Invert	281.0000
To Invert	278.0000
Length	170.0000
Diameter	12.0000
Coefficient	0.0140
Parallel	
Information	
Output	
Total Flow	**0.6250 cfs**
Unpeakable Flow	0.0000 cfs
Peakable Flow	0.6250 cfs
Coverage Flow	0.0000 cfs
Infiltration Flow	0.0000 cfs
Storm Flow	0.0000 cfs
Flow Type	Free Surface
Velocity	3.9710 ft/s
Reserve Capacity	3.7815 cfs
d/D	0.2544
q/Q	0.1418
Water Depth	0.2544 ft
Critical Depth	0.3290 ft
Froude Number	1.6463
Full Flow	4.4065 cfs
Coverage Count	0.0000
Backwater	No
Adjusted Depth	0.2544 ft
Adjusted Velocity	3.9710 ft/s
Type	Gravity Main
Channel Type	Circular
From Node	96
To Node	OUTLET110
Upstream Invert	281.0000 ft
Downstream Invert	278.0000 ft
Length	170.0000 ft
Diameter	12.0000 in
Friction Factor	0.0140
Pipe Slope	0.0176

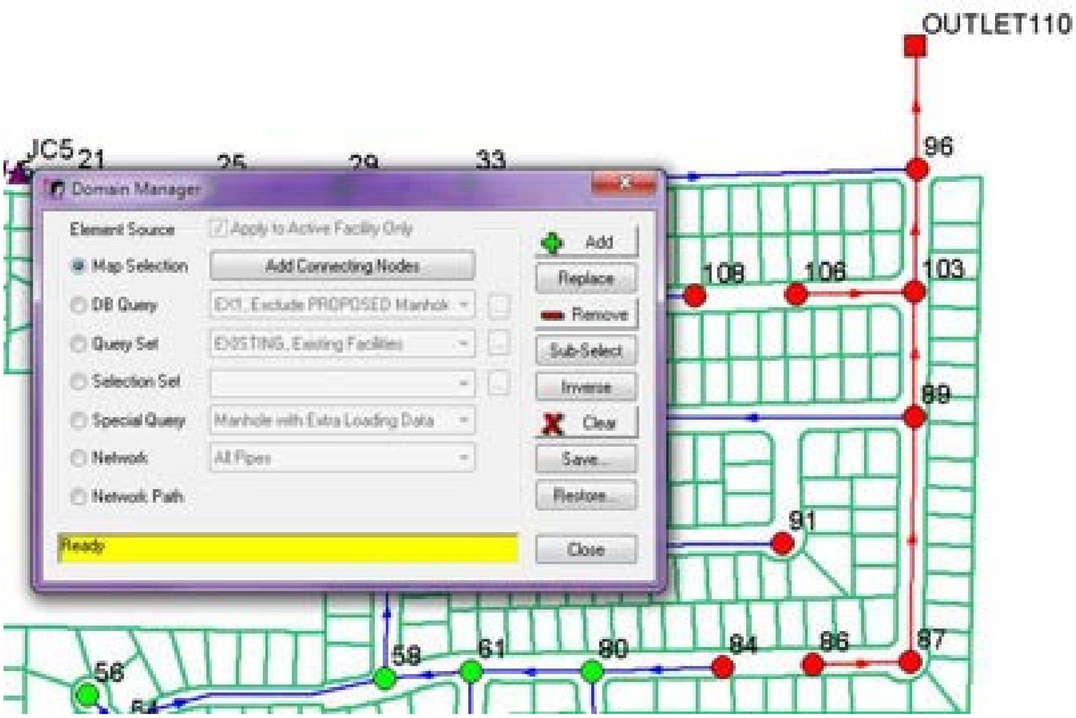
Opening a Graph

One of the best ways to see EPS model results is through time-series graphs. With InfoSewer you can graph any simulation result variable for links or nodes. You can also display pipe profile graphs. Here, you will graph the temporal variation in flow for a selected gravity main and create a pipe profile graph.

Create a Domain

A domain is a temporary selection of network components which is used to edit, view, or report on a portion of the network model. Components in the domain are highlighted for visual reference. The domain is useful in that many of InfoSewer data editing and mapping functions can be performed globally on components in the domain. There are several methods for including network components in the domain, including graphical selection and database query. First create a domain of the desired network components. You may pick the entire network or a smaller subset of the network at your discretion. Unselected components (those not included in the domain) will not be affected by the rest of this tutorial step.

1. Choose the *DOMAIN®DOMAIN MANAGER* command from the *TOOLS* menu or alternately click once on the “Domain Manager” button  on the InfoSewer toolbar. The *DOMAIN MANAGER* dialog box appears on the screen.

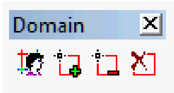


2. On the *DOMAIN MANAGER* dialog box, choose the *MAP* Selection in the *ELEMENT SOURCE* area and then click on the “Add” button.

3. InfoSewer will then prompt you to select an area of the network. Place the cursor anywhere in the network then press the left mouse button and drag the cursor to the desired selection window. Click on the right mouse button and choose “Enter” to confirm the selection window.

4. All the network components that fall entirely within the selection window should now appear highlighted in the color red and are now included in the domain. Close the *DOMAIN MANAGER* dialog box by selecting the “Close” button 

.

NOTE: You can alternatively use the “Enlarge Domain” button , or the “Reduce Domain” button , to graphically select an area for inclusion or removal from the domain. 

Specify Map Customization Options

You will customize the display for the pipes included in the domain as follows:

1. Choose the *MAP DISPLAY* tab from the InfoSewer View Menu.

2. Choose *OUTPUT DATA VIEW* as the *DESIRED MAP VIEW*. This output data source stores the results of your most recent simulation run.

3. Select *GRAVITY MAINS* as the *ELEMENT TYPE* from the *LINK* tab whose display will be customized and then select *PIPE FLOW* as the *DATA CATEGORY* that will be displayed.

4. Click on the "Display Settings" button of the *MAP DISPLAY* tab from the InfoSewer View Menu.

5. Now choose the colors and range breaks used to display the base flow. Define five classification range breaks in the *CLASSES* tab of the *LAYER PROPERTIES - LINK* window as defined as follows:

Classification	Range	Color
1	**Less than 0.50**	**Magenta**
2	**0.50-> 1.00**	**Cyan**
3	**1.00 -> 1.50**	**Green**
4	**1.50-> 2.00**	**Yellow**
5	**Greater than 2.00**	**Red**

· For this example, enter "0.5" in the *BREAK* field as the first break value and assign a magenta color to highlight all pipes with a total flow less than 0.5 cfs. Single-click on the corresponding color box, choose the desired color and then click on the "OK" button.

· Repeat the above step using the range breaks shown in the table above.

· Choose the *LABEL PROPERTIES* tab and check the *SHOW TEXT* check-box to see the labeling pipes with their flow values. You can also edit the font and the color of the text displayed by clicking on the "Font" and "Color" buttons respectively. Click the "OK" button to close the window.

6. Finally, click on the "Update Map" button Update Map of the *MAP DISPLAY* tab from the InfoSewer View Menu and the pipes in the current domain will be redrawn to reflect your choices.

NOTE: *To remove the map customization and revert to the default display, select MAP DISPLAY tab from the InfoSewer View Menu and choose DEFAULT VIEW as the DESIRED MAP VIEW.*

You can now open the *DOMAIN MANAGER* dialog box and choose the "Restore" button Restore... at any time. You can alternatively use the "Clear Domain" button .

Refer to the User Interface à Control Center topic in the on-line InfoSewerHelp for more information on InfoSewermap display customization using model inputs or results.

Step 8: Browse Model Results

Open the Output Report Manager

Another method for reviewing model results is to view results in tabular or spreadsheet format. InfoSewer provides several tools for browsing results in this fashion. The OUTPUT REPORT MANAGER is used to display simulation results in either tabular or graph form. You can fully customize the format, the content and the layout of display tables and graphs with the OUTPUT REPORT MANAGER.

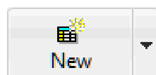
In this example you will open a table displaying model results for gravity pipes.

OPEN THE OUTPUT REPORT MANAGER

1. Select the *OUTPUT REPORT/GRAPH* command from the *TOOLS* menu or alternately choose the corresponding Output Report/Graph  icon. The *OUTPUT REPORT MANAGER* appears on the screen.

2. On the *OUTPUT REPORT MANAGER* choose the "New" button 

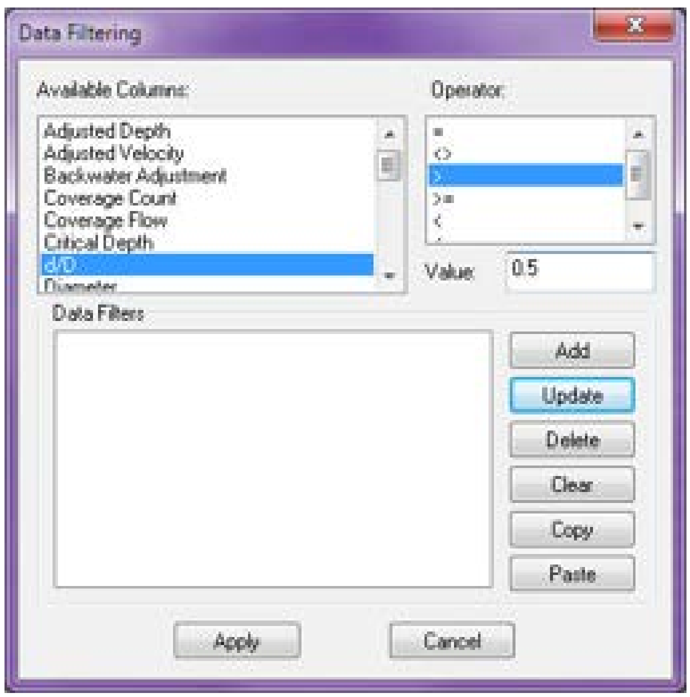
. When the *OUTPUT REPORT & GRAPH* dialog appears on the screen, select ***Active*:STEADY*** (containing results from your most recent model run) from the *ALL OUTPUT SOURCES* area. Choose *STATIC GRAVITY MAIN REPORT* from the *REPORT* tab, and then click on the "Open" button.

3. The results are displayed for steady state simulation. You can open additional output reports by choosing the "New" button 

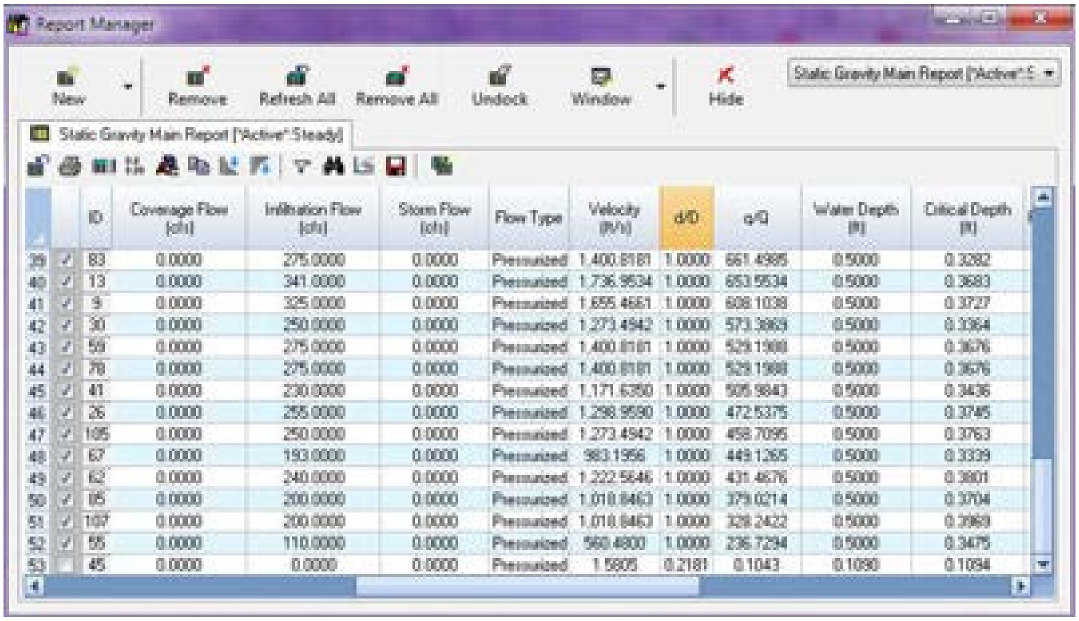
.

You may now restrict the display to only those components meeting a user-specified criterion. For instance, you could request to see only those pipes where the d/D ratio is greater than 0.50. To do this, perform the following:

1. Select the "Filtering" icon at the top of the Output Report Manager. The *DATA FILTERING* dialog box appears on the screen.
2. Click once on *d/D* in the *AVAILABLE COLUMNS* area.
3. Click once on > (greater than) in the *OPERATOR* area. Type "0.5" in the *VALUE* entry field.
4. Click on the "Add" button. The query "d/D > 0.50" should appear in the *DATA FILTERS* list. Click on the "Apply" button.



5. When the *DATA FILTERING* dialog box closes, an information dialog box appears on the screen defining the number of selected records. Moreover, the pipe records meeting your criterion will be identified with a check.



6. The selected records may be scattered throughout the table. Therefore you can sort the table on the d/D column such that the selected records are grouped together, by selecting the check mark column header and clicking on the sort descending button .

7. Click once on the “d/D” field header in the *OUTPUT REPORT MANAGER*. The entire column should be highlighted.

8. Choose the “Sort Descending” button . The Static Gravity Main Report table is now sorted from the highest d/D ratio to the lowest. The “Sort Ascending” button would sort the pipes in reverse (ascending) order.

Refer to the Reports and Graphing à Output Report Manager topic in the on-line Help for more information on displaying model results in tabular format.

Step 9: Running a Steady State Design Simulation

Domain and Group Editing

In this tutorial, you will create a domain containing all the pipes.

1. Choose the *DOMAIN®DOMAIN MANAGER* command from the *TOOLS* menu or alternately click once on the “Domain Manager” icon on the InfoSewertoolbar. The *DOMAIN MANAGER* dialog box appears on the screen.
2. To add all pipes to the domain, choose *NETWORK* in the *ELEMENT SOURCE* area, select *ALL PIPES* from the drop-down list and then press the “Add” button. All pipes should now appear highlighted in the color red. Press the “Close” button to close the *DOMAIN MANAGER* dialog box.

3. Choose the *GROUP EDIT ON DOMAIN* command from the *EDIT* menu or alternately choose the corresponding icon on the InfoSewertoolbar. The *GROUP EDITING* dialog box appears on the screen.
4. Choose the *PIPE CONSTRAINTS* tab on the *GROUP EDITING* dialog box. All components in the domain will be assigned a maximum velocity of 9.5 ft/s and a minimum velocity of 1.2 ft/s.

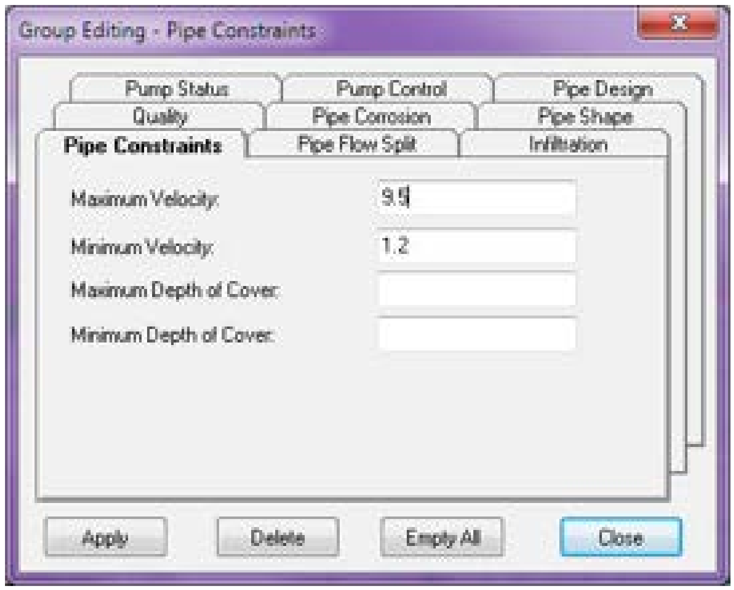
   · Enter “9.5” in the *MAXIMUM VELOCITY* field.

· Enter “1.2” in the *MINIMUM VELOCITY* field.

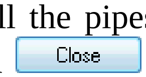
· Choose the “Apply” button.

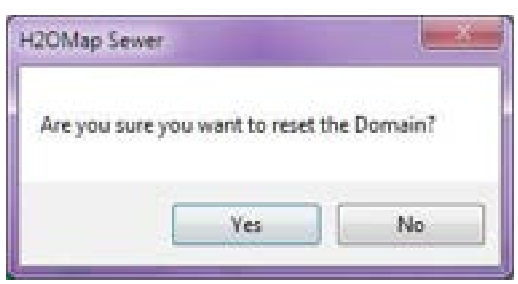
· Click “OK” when prompted to confirm applying changes and then close the *GROUP EDITING* dialog box.

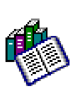
· Click on the “Close” button  to close the *GROUP EDITING* dialog box.



Any values set on the *GROUP EDITING* dialog box are applied to the components in the domain.

5. Finally, clear the domain by opening the *DOMAIN MANAGER* dialog box (see Step 1 above) and click once on the “Clear” button . All the pipes should now appear in their original color. Press the “Close” button  to close the *DOMAIN MANAGER* dialog box.





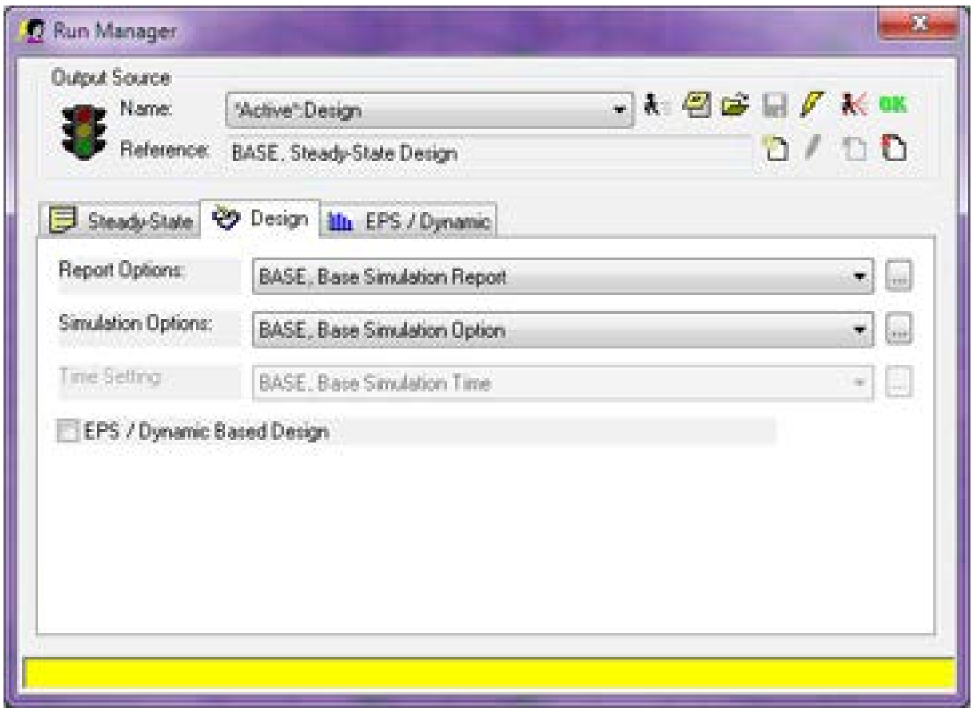
Refer to the Domains and Facilities à Domain Manager topic in the on-line InfoSewerHelp for more information on using the domain to edit values for multiple network components.

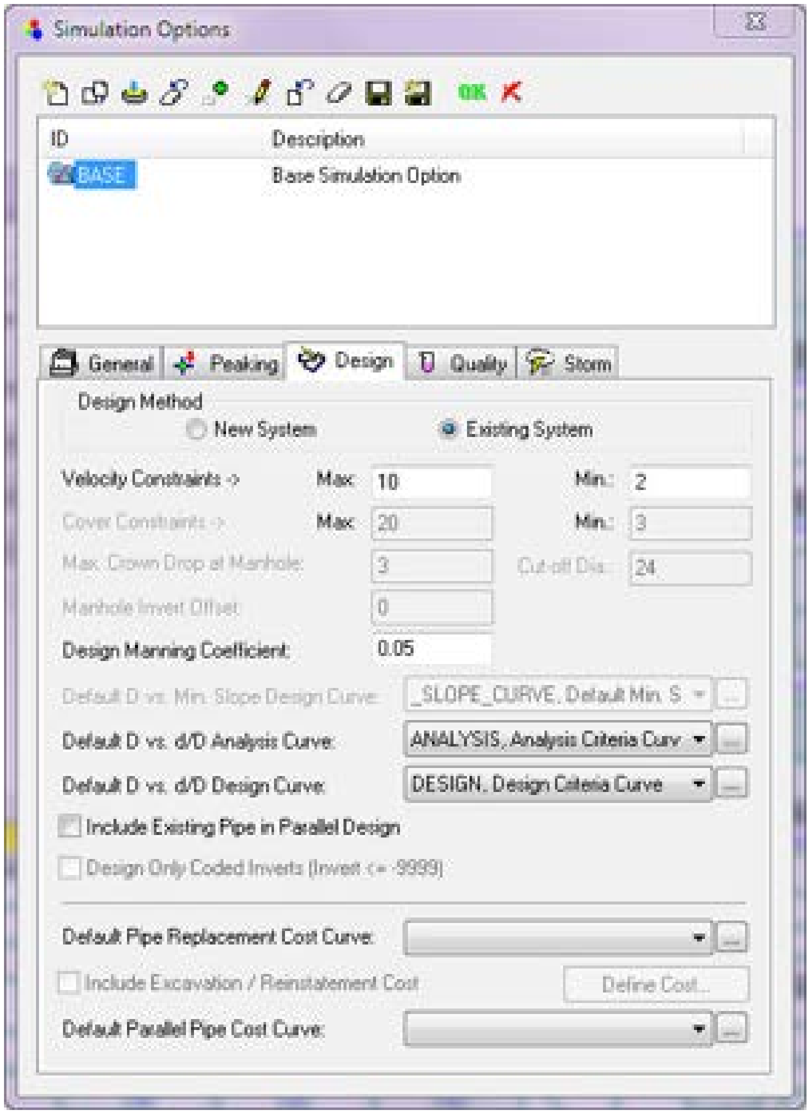
Now that you have completed the process for creating design analysis criteria, the next step is to run a steady state design simulation of the collection system. In step 2, we specified flow units and assigned parameters for a peaking equation to be applied during the simulation.

To run the simulation, perform the following:

1. From the *TOOLS* menu, select *RUN MANAGER* or alternately click the "Run Manager" icon  from the InfoSewer toolbar. The *RUN MANAGER* dialog box appears on the screen.

2. Choose the *DESIGN* tab. Before running the simulation you can specify various report and simulation options to be applied to the simulation.





3. Click on the “Run” button at the top of the *RUN MANAGER* dialog box. Choose the “OK” button to close the *RUN MANAGER* dialog box.

NOTE: *The model results are stored in an output source entitled:*

Active*:Design

When you want to access results from this run, you will choose the output source with that name.

Open the Output Report Manager

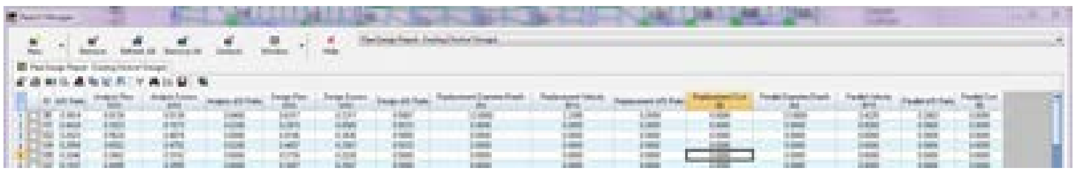
1. Select the OUTPUT REPORT/GRAPH command from the TOOLS menu or alternately choose the corresponding Output Report/Graph  icon. The OUTPUT REPORT MANAGER appears on the screen.

2. On the OUTPUT REPORT MANAGER choose the “New” button 

. When the OUTPUT REPORT & GRAPH dialog appears on the screen, select *Active*:Design (containing results from your most recent model run) from the ALL OUTPUT SOURCES area. Choose PIPE DESIGN REPORT from the REPORT tab, and then click on the “Open” button.

3. The results are displayed for a steady state design simulation. This report will show you which pipes need to be replaced or have parallel pipes installed due to d/D requirements specified in the previous exercises. In addition, InfoSewerchecks the flow velocity in the pipe against the minimum and maximum design velocities. If the velocity is out of the range, then the pipe is flagged as exceeding the design criteria for velocity.

4. When complete, choose the Hide button to close the Output Report/Graph dialog box.

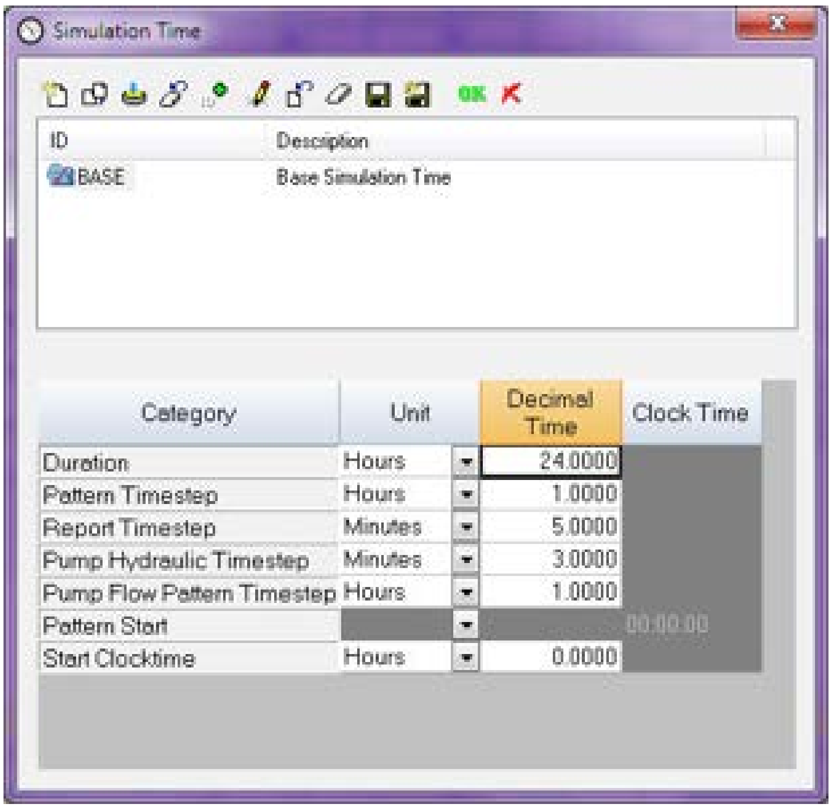


Step 10: Running an Extended Period Simulation

Set Duration of the Simulation

2. In the *DURATION* entry field, specify “hours” as the unit and enter “24” (indicating a length of the entire simulation of 24 time units), and press the *TAB* key to apply the change.

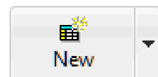
3. Change the *PATTERN TIMESTEP* to “1” hour. InfoSewerwill apply each sewer loading pattern multiplier (see below) for each timestep period. Therefore, 24 loading multipliers are required to model varying loadings over a 24-hour period. The *SIMULATION TIME* dialog box should appear as:



4. Choose the “OK” button  to close the *SIMULATION TIME* dialog box.

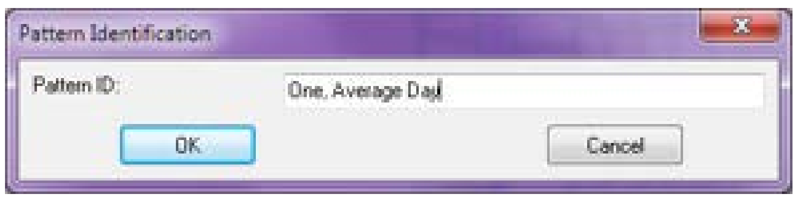
Build a Pattern to Define Variable Loadings

Build a time-series (stepwise) pattern defining a set of multipliers to be applied to the base loading of each manhole at each simulation timestep. Each pattern is identified by a unique alpha-numeric identifier (up to twenty characters with no embedded spaces or special characters) and an optional (recommended) description. At any timestep, the loading at a manhole is the base loading multiplied by the current pattern multiplier. You will enter 24 multipliers for an average day loading pattern and InfoSewerwill apply each multiplier for a 1-hour period during the simulation.

1. Select the *OPERATION DATA* tab  from the Attribute Browser select *PATTERN* and click on the "New" button 

in the upper left-hand corner of the *CONTROL CENTER* window.

2. You are then prompted to enter a new ID and description for the pattern. Enter "One**, Average Day**" in the *PATTERN ID* field and choose the "OK" button.



3. The *PATTERN* dialog box now appears on the screen. You will enter the following 24 pattern multipliers (representing a residential neighborhood).

Factor 1	**Factor 2**	**Factor 3**	**Factor 4**	**Factor 5**	**Factor 6**
0.65	0.65	0.65	0.75	0.90	1.2
Factor 7	**Factor 8**	**Factor 9**	**Factor 10**	**Factor 11**	**Factor 12**
1.45	1.8	1.5	1.1	1.0	1.1
Factor 13	**Factor 14**	**Factor 15**	**Factor 16**	**Factor 17**	**Factor 18**
1.0	0.9	0.85	1.05	1.2	1.1
Factor 19	**Factor 20**	**Factor 21**	**Factor 22**	**Factor 23**	**Factor 24**
1.05	1.0	0.90	0.85	0.75	0.60

0.6500

0.6500

0.6500

0.7500

0.9000

1.2000

1.4500

1.8000

1.5000

1.1000

1.0000

1.1000

1.0000

0.9000

0.8500

1.0500

1.2000

1.1000

1.0500

1.0000

0.9000

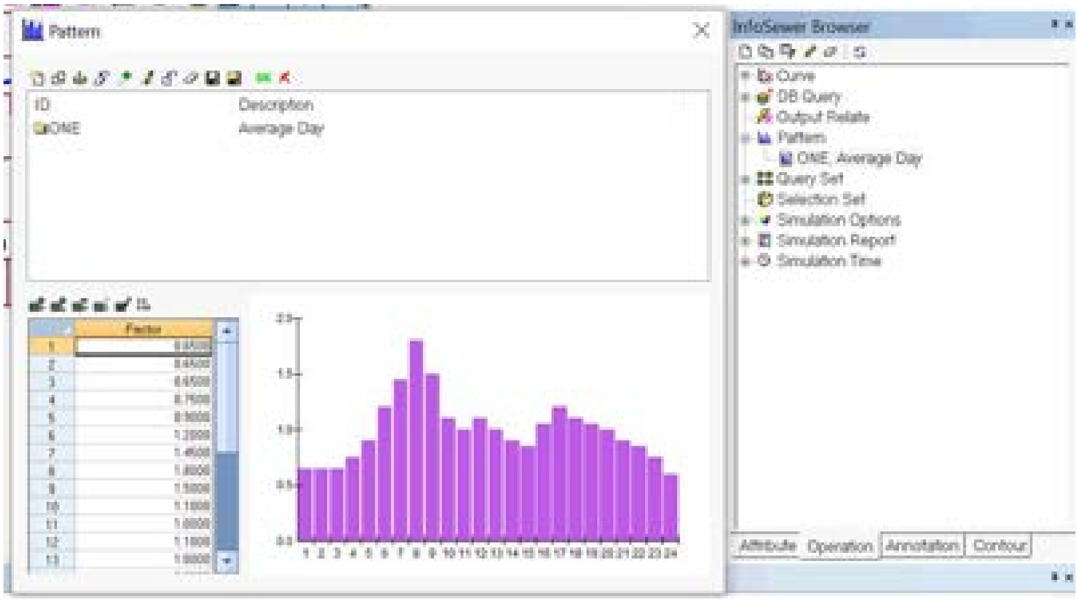
0.8500

0.7500

0.6000

4. To define the twenty-four multipliers for Pattern 1, perform the following:

- Choose the “Set Rows” button , enter “24” as the value, and choose “OK”.
- Enter the pattern factors as defined above and press the *ENTER* key to apply the change. As you add the multipliers the graph will be automatically updated on the dialog box.
- Click on the “Save” button at the top of the dialog box to apply the factors to pattern 1. The following picture illustrates how the pattern should appear on the dialog box.



5. Choose the “OK” button to close the *PATTERN* dialog box. Pattern 1 now contains 24 multipliers and is ready for use.

Refer to the Curves and Patterns à Patterns topic in the on-line InfoSewerHelp for more information on patterns and their use in InfoSewer Pro.

Now that a time-series pattern of loading multipliers has been defined, the pattern can be assigned to the manholes.

1. To assign the 24 loading multipliers to the manholes, click once on the "DB Editor" button . The *DB EDITOR* appears on the screen.
2. When the *OPEN TABLE* dialog box appears on the screen, choose the *MANHOLE HYDRAULIC (MODELING) DATA* table in the *ELEMENT HYDRAULIC DATA* section. Select *ENTIRE TABLE* in the *DISPLAY SCOPE* area (indicating you want the entire contents of the table to be displayed and available for editing), and click once on the "OK" button. The *MANHOLE HYDRAULIC (MODELING) DATA* table appears on the screen.
3. To adjust the contents of an entire field, in this case *PATTERN1*, click once on the column header with the mouse. The entire column should be highlighted.
4. Now allocate pattern ID "One" (Average Day) to each manhole. Choose the "Block Editing" button while the *PATTERN1* field remains highlighted. When the *BLOCK EDITING* dialog box appears on the screen, enter "One" in the *REPLACE WITH* field. Choose the "OK" button to close the dialog box. All manhole nodes now have a pattern ID of "One".
5. Select the "Save" button and then choose the "Close" button at the top of the *DB EDITOR* window.
6. Close the *DB EDITOR* by selecting the "Exit" button at the top of the *DB EDITOR* window.

Set controls for pump P20 so that the pump turns on when the water level in wet well WW5 rises above 16 feet (above the wet well's bottom elevation) and off when the water level in wet well WW5 falls below 2 feet at any time during the simulation.

1. Click on the "Select" button , place the cursor on pump P20 and then press the left mouse button.

2. To define the control rules to turn pump P20 on and off, choose the "Control" button of the *ATTRIBUTE BROWSE-LINK* window. The *CONTROL DATA* dialog box appears on the screen.

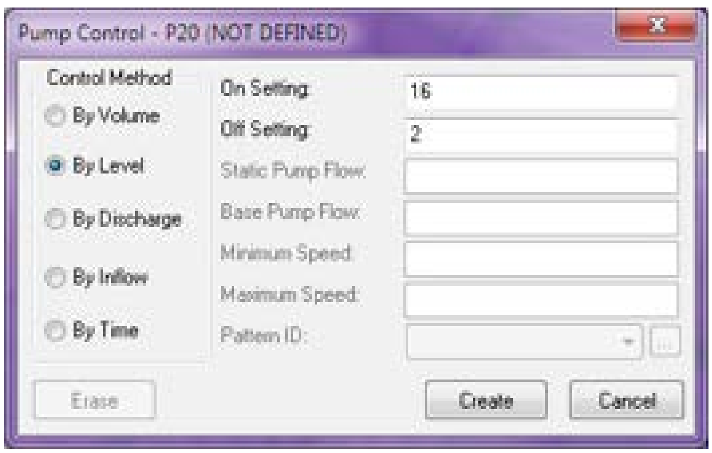
3. Define the following control settings for the pump:

· Choose *BY LEVEL* as the *CONTROL METHOD*.

· Enter "16" as the *ON SETTING*.

· Enter "2" as the *OFF SETTING*.

· Click on the "Create" to close the dialog box.



The following picture illustrates how the *PUMP CONTROL* dialog box should appear after defining the two control rules for pump P20.

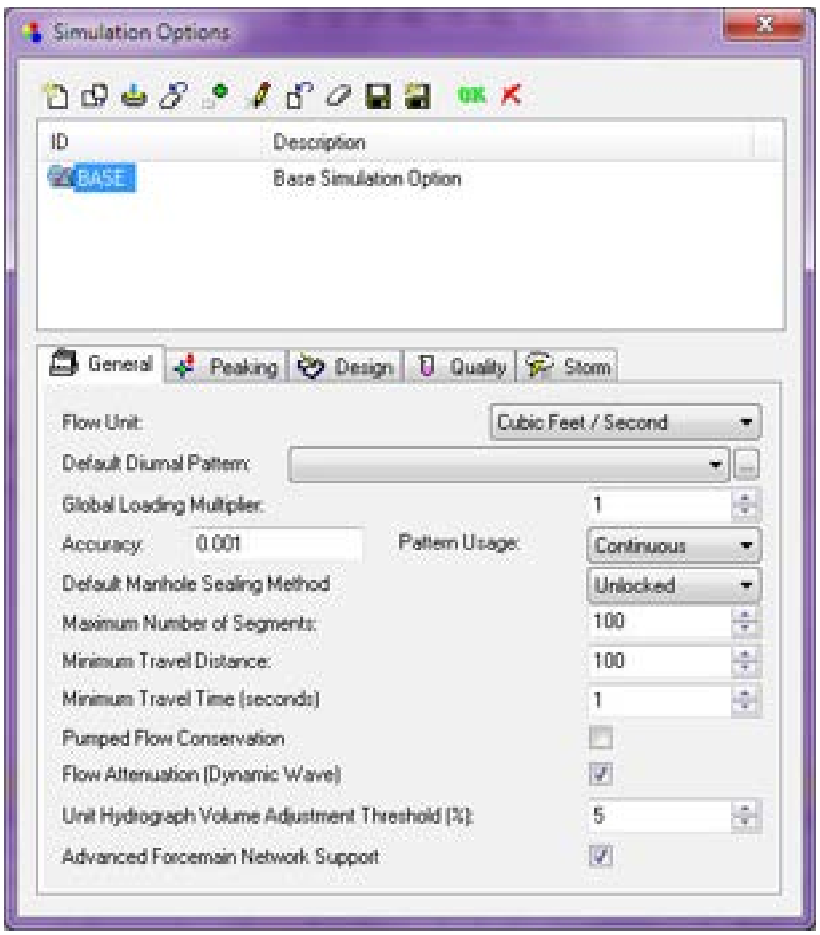
4. Click once on the "Save" button of the *ATTRIBUTE BROWSER–LINK* window.

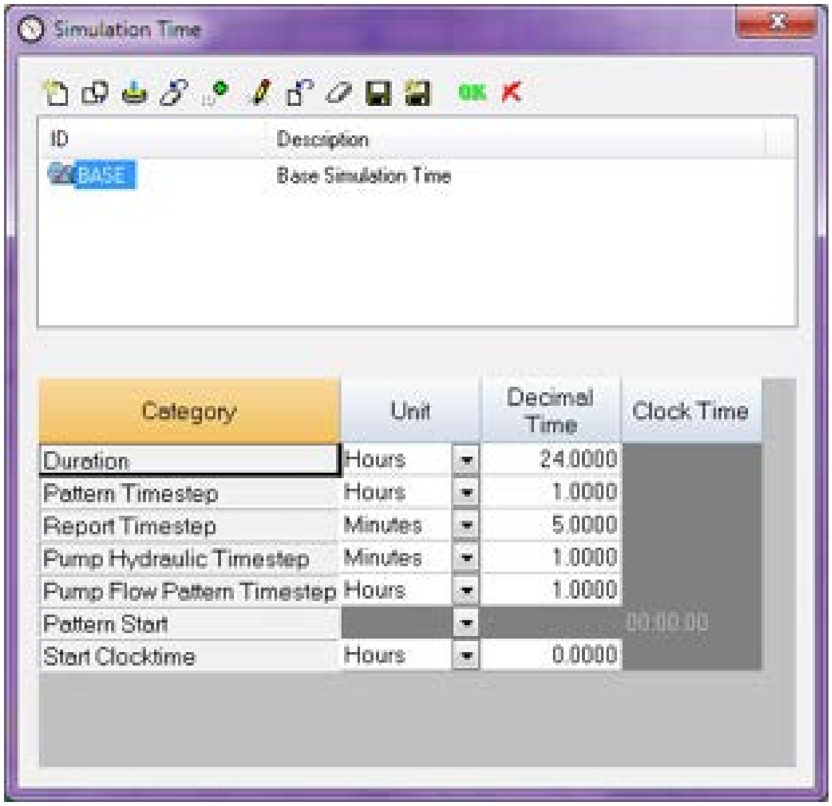
Refer to the EPS Modeling topic in the on-line InfoSewerHelp for further information on pump controls and control sets.

Now that you have completed the process for creating an EPS model, the next step is to run an extended period simulation of the collection system.

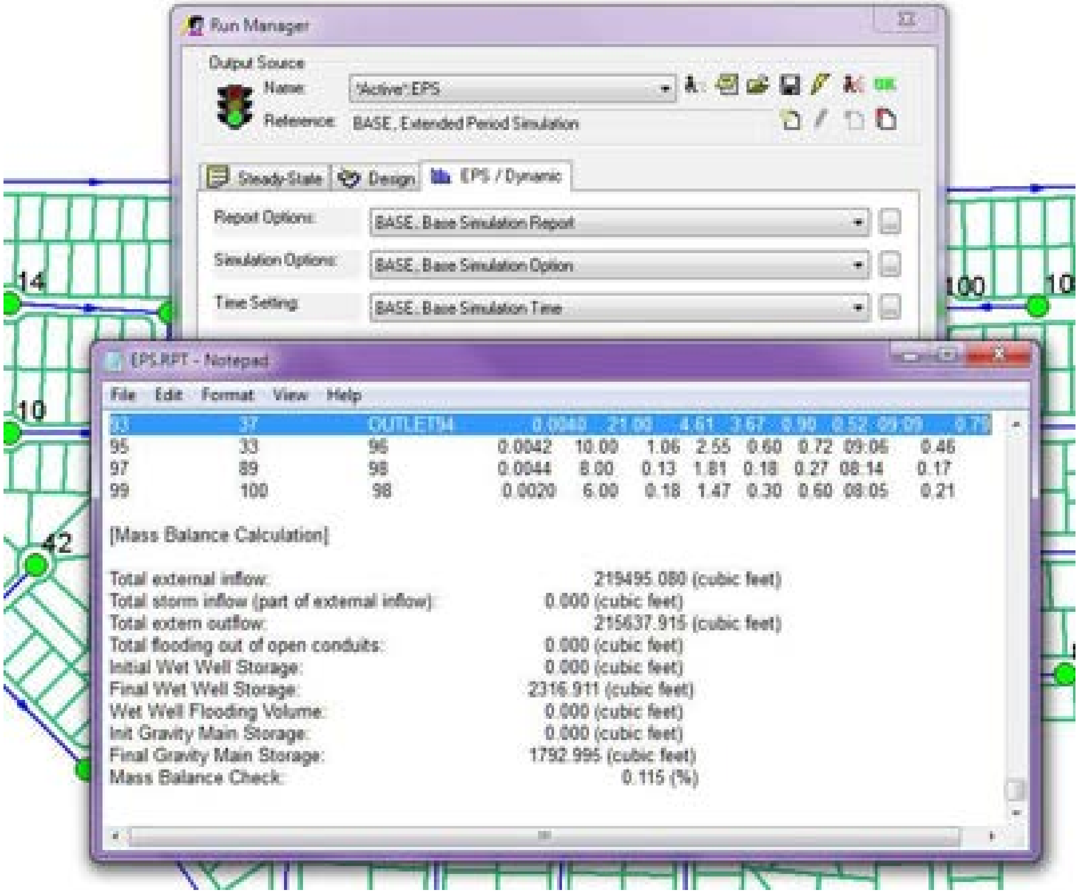
To run the simulation, perform the following:

1. From the *TOOLS* menu, select *RUN MANAGER* or alternately click the "Run Manager" icon from the InfoSewertoolbar. The *RUN MANAGER* dialog box appears on the screen.
2. Choose the *EPS / DYNAMIC* tab. Before running the simulation you can specify various report, simulation and time options to be applied to the simulation.
3. Click on the "Run" button at the top of the *RUN MANAGER* dialog box. Upon successful completion of the simulation, the status stoplight on the *RUN MANAGER* should show green, indicating successful completion of the simulation run. Choose the "OK" button to close the *RUN MANAGER* dialog box.





Simulation Time
ID
Description
BASE
Base Simulation Time
Category
Unit
Decimal Time
Clock Time
Duration
Hours
24.0000
Pattern Timestep
Hours
1.0000
Report Timestep
Minutes
5.0000
Pump Hydraulic Timestep
Minutes
1.0000
Pump Flow Pattern Timestep
Hours
1.0000
Pattern Start
00:00:00
Start Clocktime
Hours
0.0000



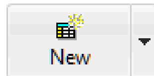
NOTE: *The model results are stored in an output source entitled:*

***Active*:EPS**

When you want to access results from this run, you will choose the output source with that name.

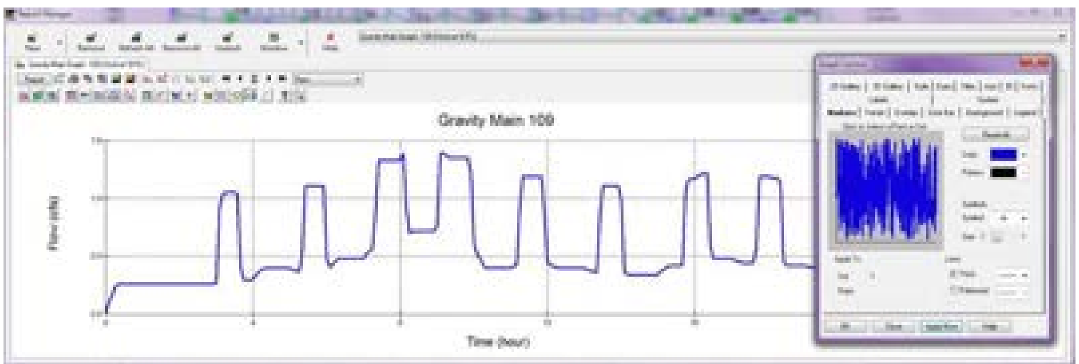
Step 11: Create Graphs

OPENING A GRAPH

1. From the *TOOLS* menu, select *OUTPUT REPORT / GRAPH*. Once opened, choose the "New" button 

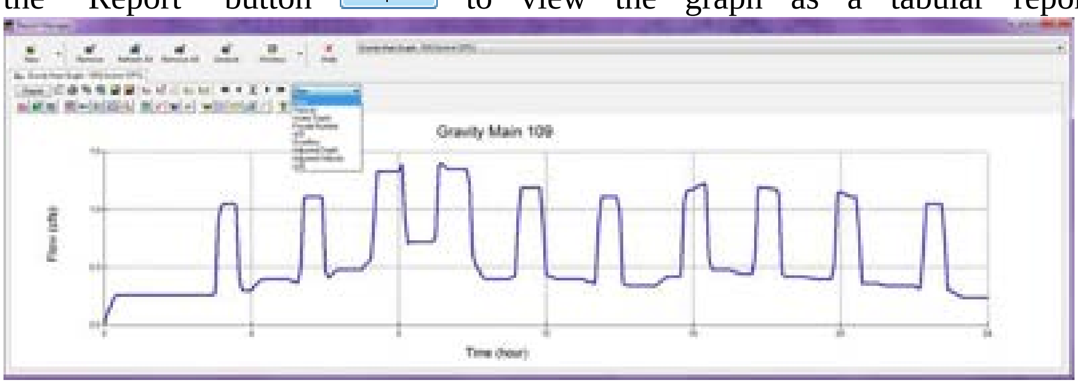
. When the *OUTPUT REPORT & GRAPH* dialog box appears on the screen, select ***Active*:EPS*** in the *ALL OUTPUT SOURCES* area. Choose *GRAVITY MAIN GRAPH* in the *GRAPH* tab, and then click on the "Open" button.

2. You are then prompted to select a pipe. Move and place the cursor on the desired pipe and press the left mouse button. A graph appears on the screen showing the daily flow variation for the selected pipe.



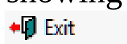
3. To change to a velocity, water depth, or d/D graph, choose the desired variable from the drop-down list above the graph. You may use any of the available buttons above the graph to customize the graph display. You can also click on the "Report" button 

to view the graph as a tabular report.



4. To create a pipe profile graph, choose the “New” button 

. When the *OUTPUT REPORT & GRAPH* dialog box appears on the screen, select ***Active*:EPS*** in the *ALL OUTPUT SOURCES* area. Choose *PIPE PROFILE GRAPH* in the *GRAPH* tab, and then click on the “Open” button.

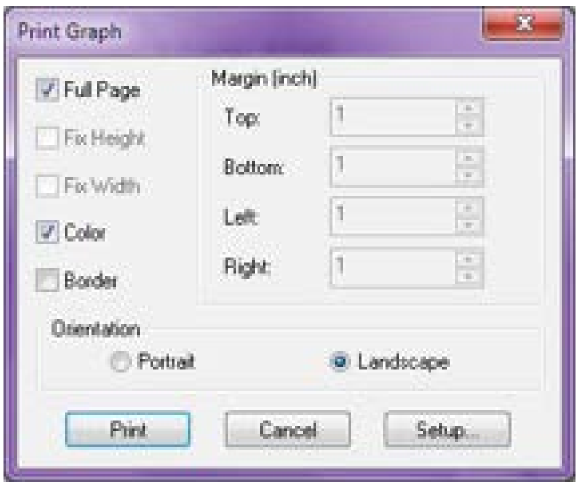
5. You are then prompted to select one or more pipes. Move and place the cursor on the desired pipe(s) and press the left mouse button. A graph appears on the screen showing the water depth for the selected pipe(s). Click on the “Exit” button 

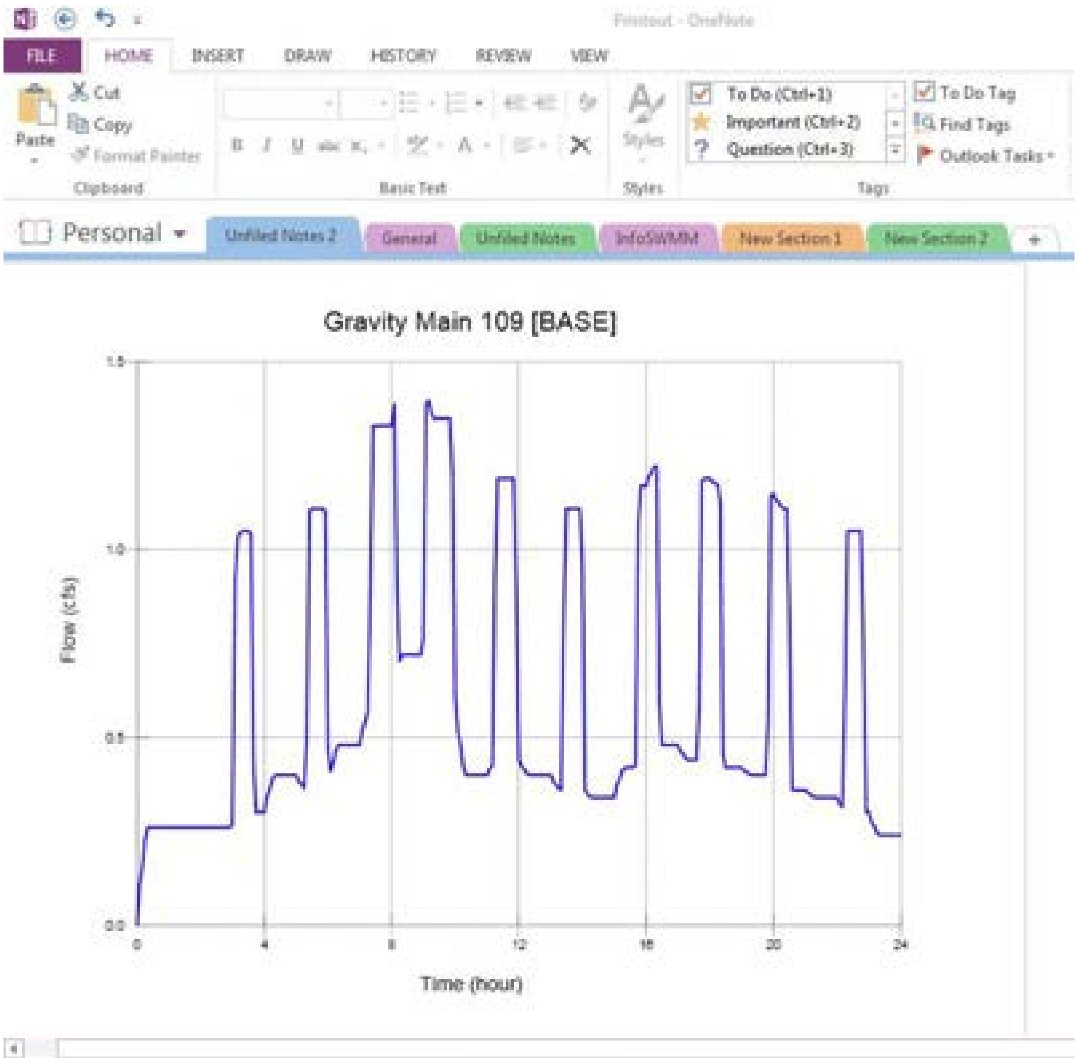
 to close the *FLOW PROFILE* dialog box and to return to the *OUTPUT REPORT / GRAPH* dialog box.

Step 12: Printing Graphs / Reports and Exporting Results

You can print or export the content of an output graph or report. Alternately you can copy selected results and paste them into other applications using standard Windows copy-and-paste functionality.

1. Open a new graph or report, or alternately make active the desired graph or report by clicking on the *GRAPH/REPORT* tab of the *OUTPUT REPORT MANAGER*.
2. Choose the “Print” button . For graphs the current graph display is sent to the printer. For reports you can specify report header and footer information. Additionally, for tabular data you can select any number of rows or columns to print. If nothing is selected (highlighted) on the table, the entire table will be printed.

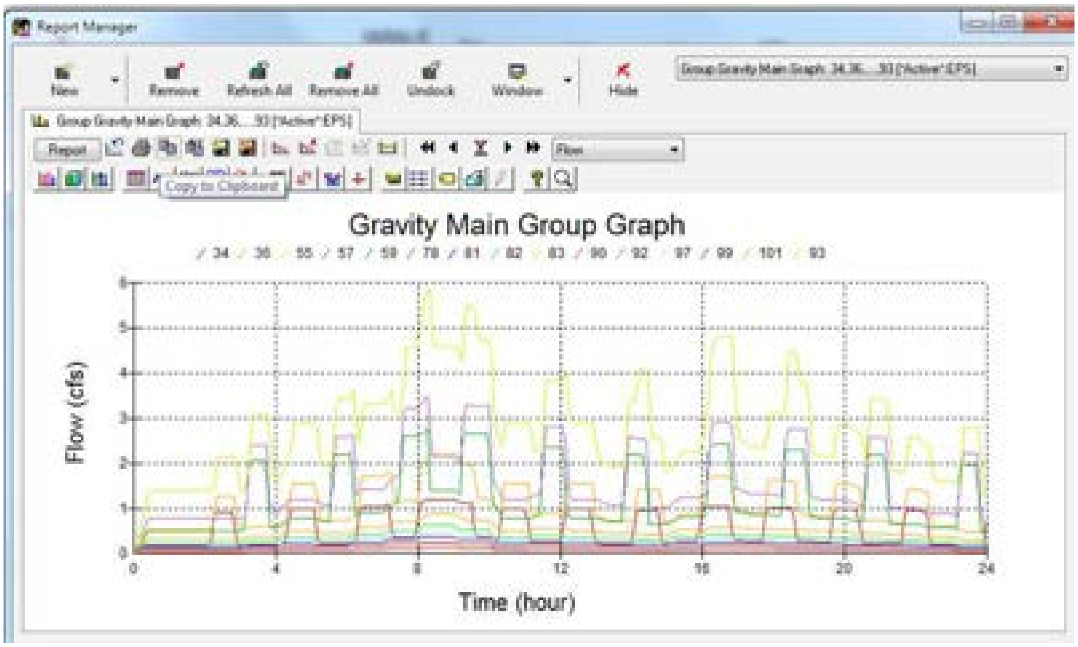




NOTE: You can also use the “System” button of the OUTPUT REPORT MANAGER to print a graph or a report.

Exporting Graphs

InfoSewergraphs can be exported to other applications via Windows *copy* and *paste* tools. After bringing a graph to the screen and optionally customizing the graph, the user can save the graph to a Windows Metafile (WMF) or Bitmap (BMP) format or export the graph display to word processors, presentation packages, or other Windows-based software applications.



Where Do I Go From Here?

Congratulations! You have completed the Quick-Start tutorial. You have been introduced to the most commonly used InfoSewercommands and functions. You have opened a model, entered network data for that model, developed and performed steady state analysis and design, as well as 24-hour hydraulic simulations, and used several tools to review simulation results.

At the end of each tutorial step, you were referred to one or more locations in the InfoSeweron-line Help system to gain more information on that particular function in InfoSewer Pro. You may now want to review those chapters and experiment with some of the new InfoSewerfeatures you learn about while reading those chapters. It is suggested that you experiment with new features using the “**Tutorial**” project. Alternately, you can begin to create a model of your sewage collection system.

For those who want to learn more about InfoSewer Pro, it is suggested that you review the following topics in the on-line Help system before experimenting with new commands and features:

· Introduction

· Data Elements

· Scenarios and Data Sets

· Database Management

You may want to print these sections for easier review. However, the on-line Help system has numerous links allowing you to quickly and easily move between related topics.

EXTENDED TUTORIALs

Two **Extended Tutorials** are provided to introduce additional capabilities of InfoSewer Pro. These tutorials illustrate the basics of the following capabilities:

· Scenario Management

· Storm Modeling

All two extended tutorials use the “**Tutorial**” project you have been using up to this point.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Step 1: Scenario Example – Planning for Future Development

A developer is proposing a new development in the community. Proposed date of construction is sometime in the future. You need to add that new development to the model and determine system performance given that new development.

In this tutorial you will create three custom scenarios.

- **Future Loadings + No New Development** – The first scenario represents your sewage collection system without the new development, but with increased loadings forecasted for the future.
- **Future Loadings + New Small Business Complex** – The second scenario represents the same future loadings but includes the proposed facilities and the loadings associated with the proposed development.
- **Future Loadings + New Large Office Complex –** The third scenario is similar to the second, but uses a larger loading for the new development.

After running the model for the three scenarios, you will use several tools to compare system performance among the three scenarios.

Refer to the SCENARIO EXPLORER topic in the InfoSewer Pro on-line Help for more information on the Scenario Manager.

Add new network components

The first step is to add facilities associated with the new development to the “Tutorial” model. At this stage of planning, a very simple piping representation is sufficient. You will add a pipe representing the planned connection to the existing system and will add a single manhole representing the estimated loading at the new development.

The pipe and manhole represent the future development will be distinguished from existing components by flagging the new components with a ZONE value of “Proposed”.

MANHOLE ATTRIBUTES

MANHOLE ID	TYPE	DIAM. (ft)	RIM ELEVATION (ft)	BASE LOAD (cfs)	LOAD TYPE
900	Normal	4.00	293.00	0.20	Type 0: Unpeakable Flow

PIPE ATTRIBUTE

PIPE ID	FROM NODE	TO NODE	DIAM. (in)	LENGTH (ft)	FROM INVERT (ft)	TO INVERT (ft)	ROUGHNESS COEFFICIENT

910	**900**	**87**	**12**	**235**	**292**	**290**	**0.013**

Add the new manhole and then add the pipe. Refer to Adding a Manhole for instructions on adding manholes and to Adding Pipes for instructions on adding pipes.

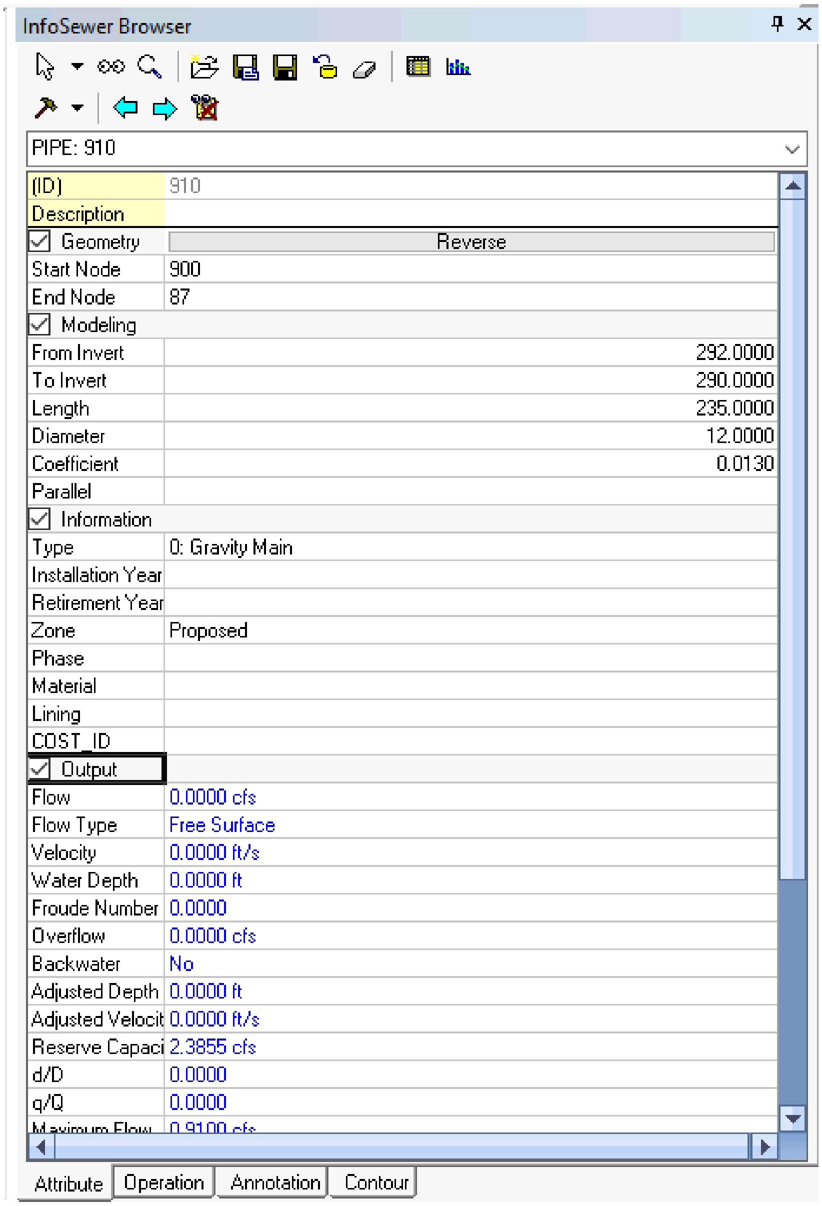
Flagging Proposed Elements

Use the service area field ZONE to store a flag indicating that the new components (i.e. manhole 900 and pipe 910) are proposed.

1. Click on the “Select” button, place the cursor on the component and then press the left mouse button.

2. In the Information section of the ATTRIBUTE BROWSER window, click once on the ZONE field and enter “Proposed”. Make sure that you enter “PROPOSED” using all capital letters.

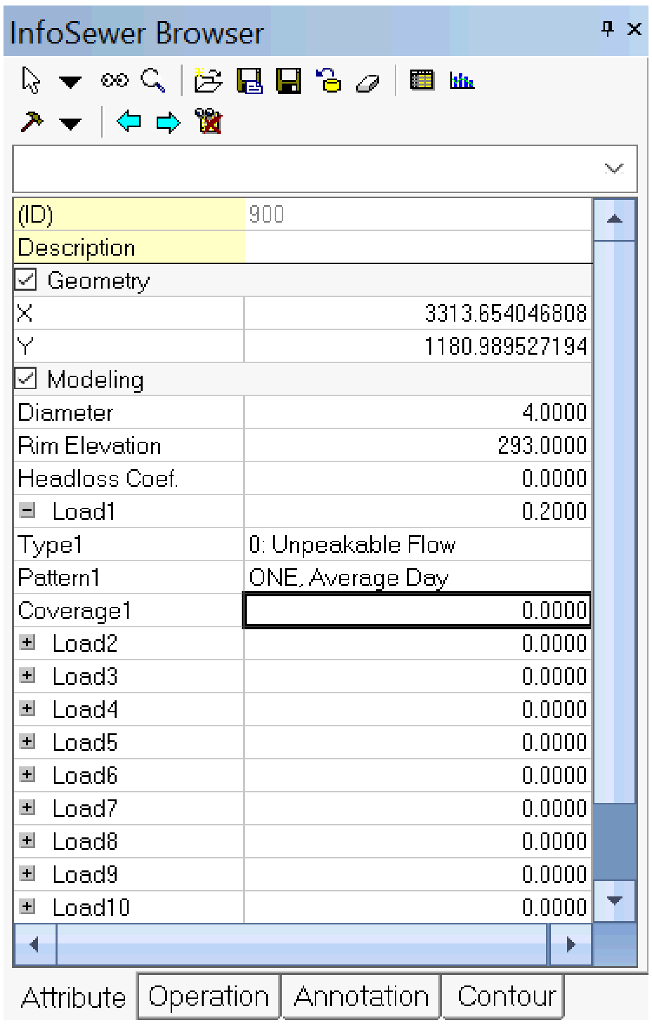
3. Finally, click once on the “Save” button in the ATTRIBUTE BROWSER window. Repeat the above procedure for the other component.


InfoSewer Browser
PIPE: 910
(ID) 910
Description
Geometry
Reverse
Start Node 900
End Node 87
Modeling
From Invert 292.0000
To Invert 290.0000
Length 235.0000
Diameter 12.0000
Coefficient 0.0130
Parallel
Information
Type 0: Gravity Main
Installation Year
Retirement Year
Zone Proposed
Phase
Material
Lining
COST_ID
Output
Flow 0.0000 cfs
Flow Type Free Surface
Velocity 0.0000 ft/s
Water Depth 0.0000 ft
Froude Number 0.0000
Overflow 0.0000 cfs
Backwater No
Adjusted Depth 0.0000 ft
Adjusted Velocit 0.0000 ft/s
Reserve Capaci 2.3855 cfs
d/D 0.0000
q/Q 0.0000
Attribute
Operation
Annotation
Contour

Adding Loadings to Manholes

To add a baseline loading for manhole 900, perform the following:

1. Click on the "Select" button, place the cursor on manhole node 900 and then press the left mouse button.
2. In the *Modeling* section of the *ATTRIBUTE BROWSER-NODE* window, click once on the *LOAD1* field and enter "0.2". In the *PATTERN1* field, use the drop-down list box to select the "One, Average Day" pattern.
3. Once complete, click once on the "Save" button in the *ATTRIBUTE BROWSER-NODE* window.



Step 2: CREATE THE NEW SCENARIO

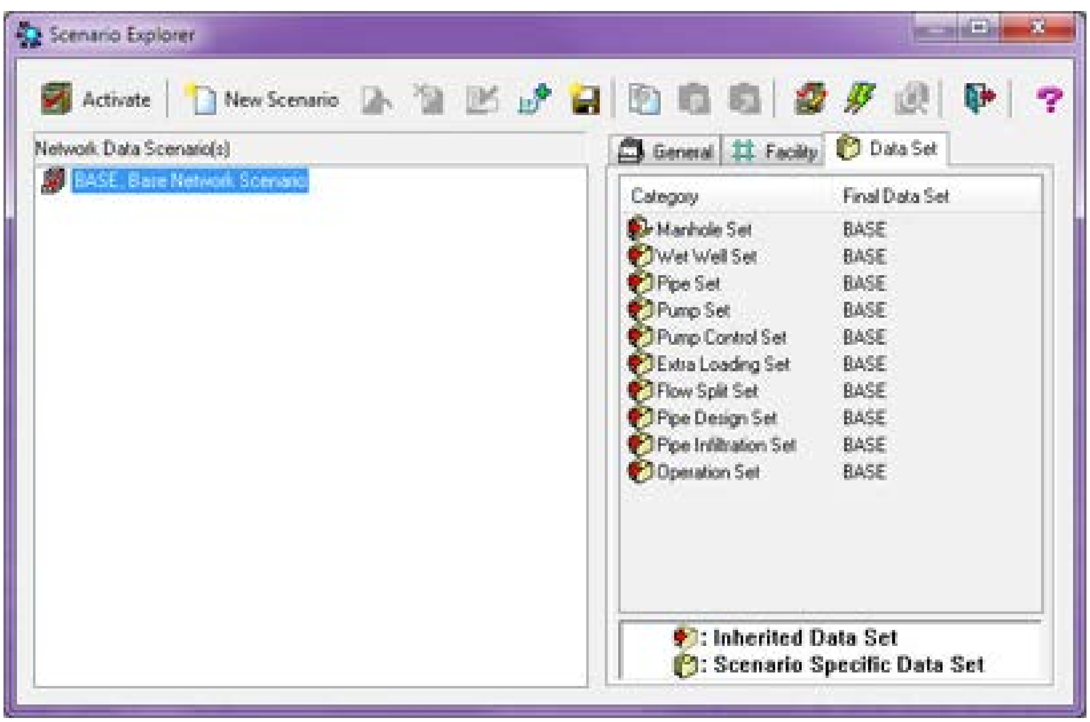
Your first custom scenario will contain the following:

· **Only existing facilities (not the new manhole and pipe)** – You will use a database query to exclude proposed facilities from this first scenario.

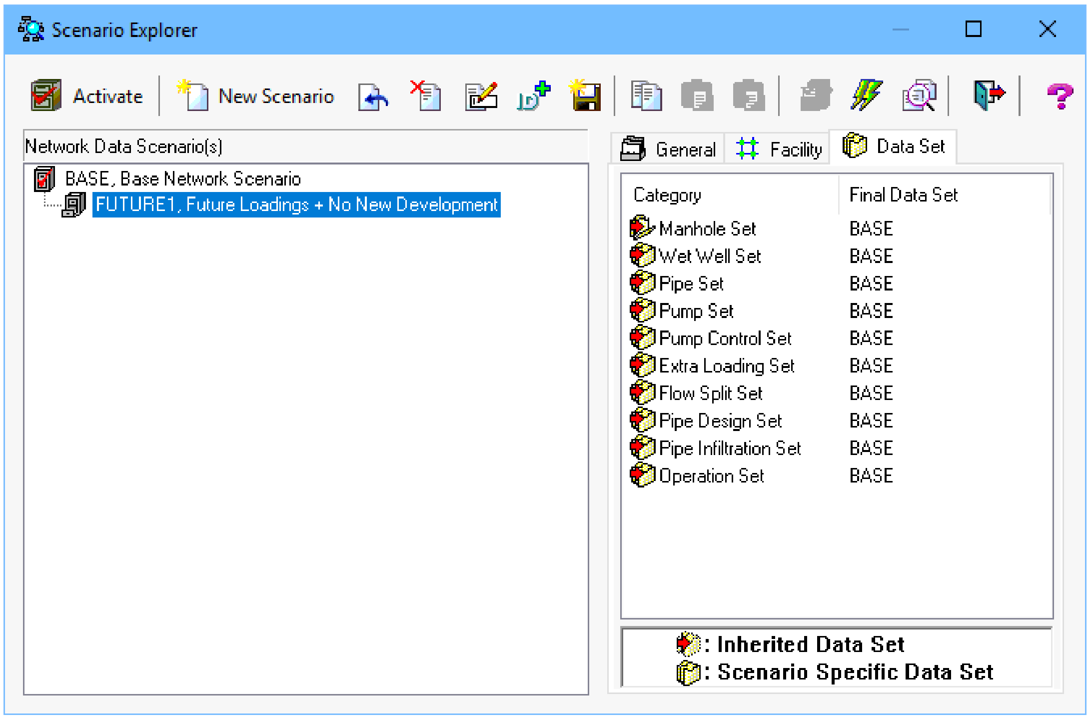
· **Future loadings** – You will adjust existing loadings by a factor of 1.4 for existing manholes in this scenario.

1. Choose the *SCENARIO EXPLORER* command from the *SCENARIO* menu or alternately choose the corresponding icon from the InfoSewer Pro toolbar. The *SCENARIO EXPLORER* dialog box appears on the screen.

2. The folder entitled **BASE, Base Scenario Data** is highlighted. This is the default and active (currently loaded) scenario. Note that the data sets associated with the BASE scenario are also entitled "BASE". Each data set is a logical grouping of modeling data. You can create and load any data set with a given scenario. The **red check** indicates the active scenario.



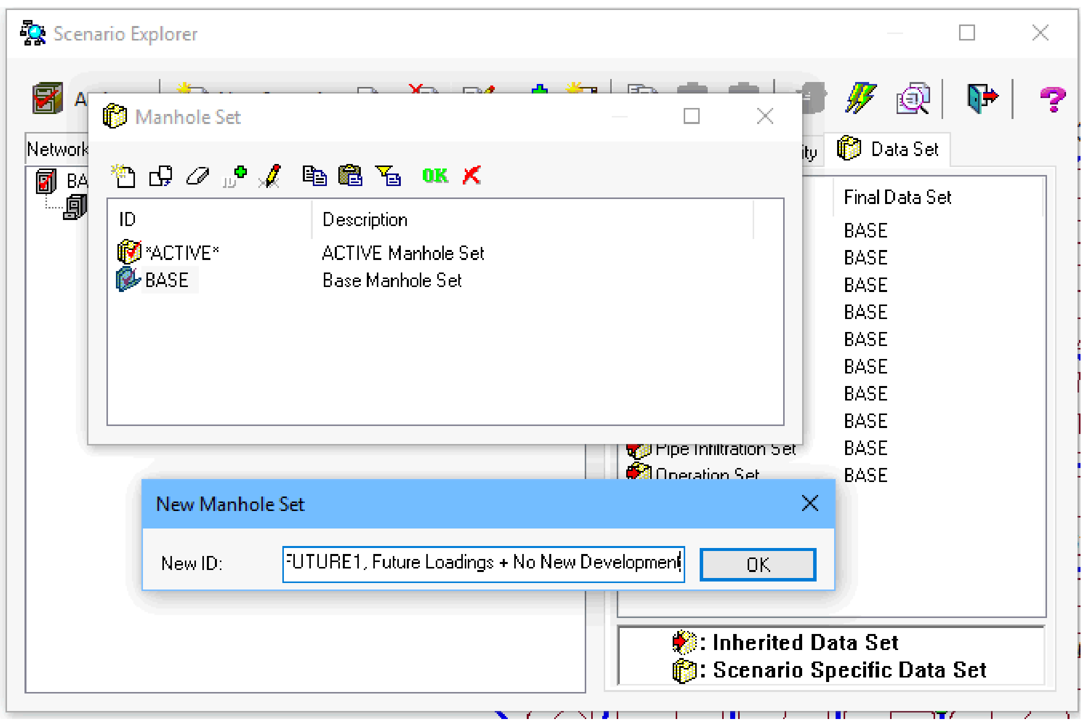
3. Create your first custom scenario by clicking on the “New Child” button (second from the left at the top of the dialog box). When prompted, enter a scenario name and description (separated by comma) of “**FUTURE1, Future Loadings + No New Development**”. Click once on the “OK” button to close the *NEW DATA SCENARIO* dialog box. The FUTURE1 scenario is now a *child* of the BASE scenario. It inherits all its properties from the parent BASE scenario. The red arrow indicates inheritance of that Data Set from the parent (Base) to the child scenario (FUTURE1).



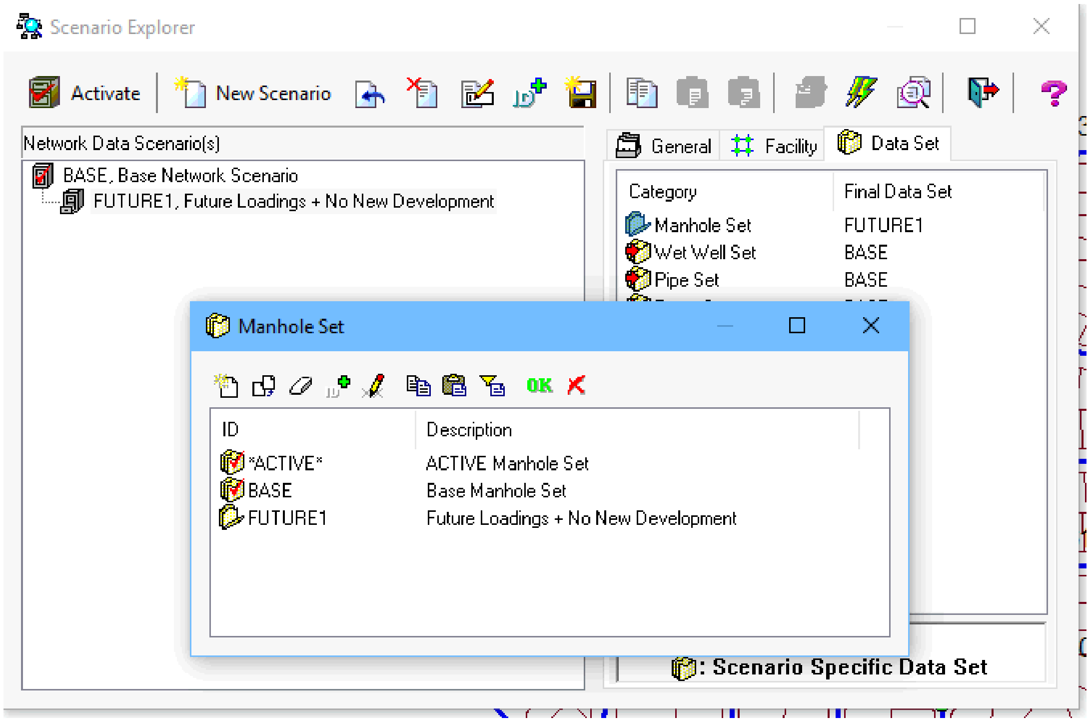
Create Unique Manhole set to store future loadings

Two elements of the new FUTURE1 scenario will be unique. You will first create a unique manhole set to store future loadings.

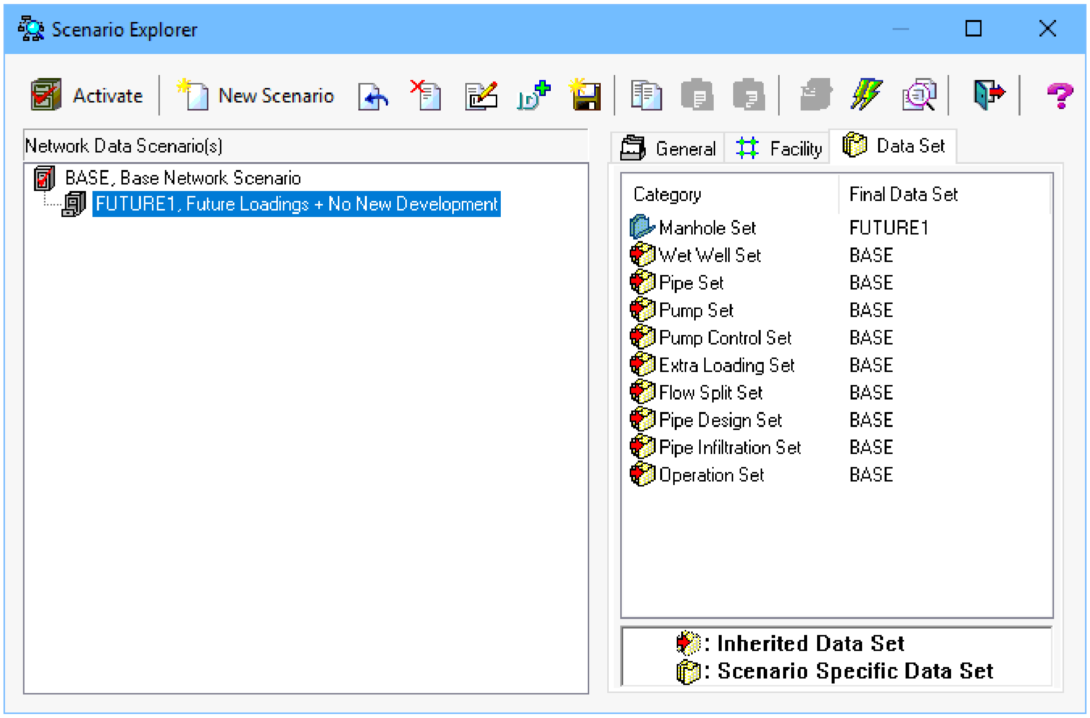
1. While the *SCENARIO EXPLORER* dialog box is open and the new FUTURE1 scenario is selected, select the *DATA SET* tab and double-click on *MANHOLE SET* in the *CATEGORY* area. The *MANHOLE SET* dialog box appears on the screen.



2. You will clone the *ACTIVE MANHOLE SET*, which contains your current loadings. Select **Active** and click on the "Clone" button (second from the left at the top of the *MANHOLE SET* dialog box). When prompted, enter the name and description of the new manhole set "**FUTURE1, Future Loadings + No New Development**". Click on the "OK" button of the *NEW MANHOLE SET* dialog box.



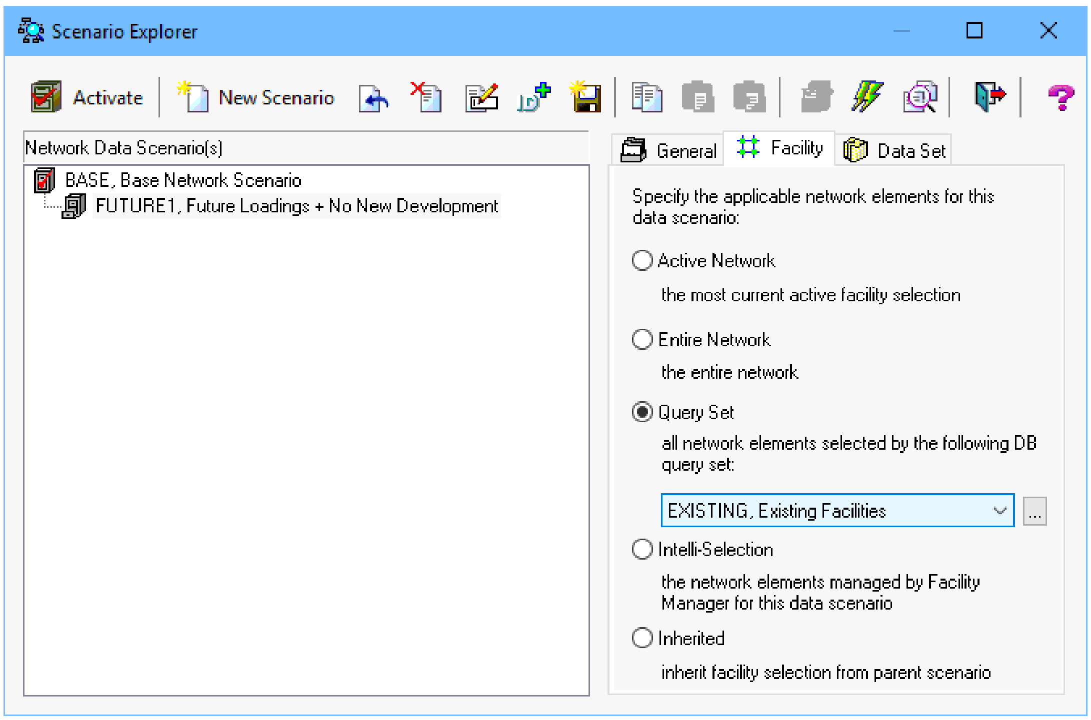
3. Click once on the “OK” button  to close the *MANHOLE SET* dialog box and to return to the *SCENARIO EXPLORER* dialog box. You will now note that the FUTURE1 manhole set (in the *FINAL DATA SET* area) is associated with the FUTURE1 scenario. The manhole set for this scenario is no longer inherited from its parent; it is unique to this scenario. Currently however, the FUTURE1 manhole set is storing existing loading values. The *SCENARIO EXPLORER* dialog box should appear as follows.

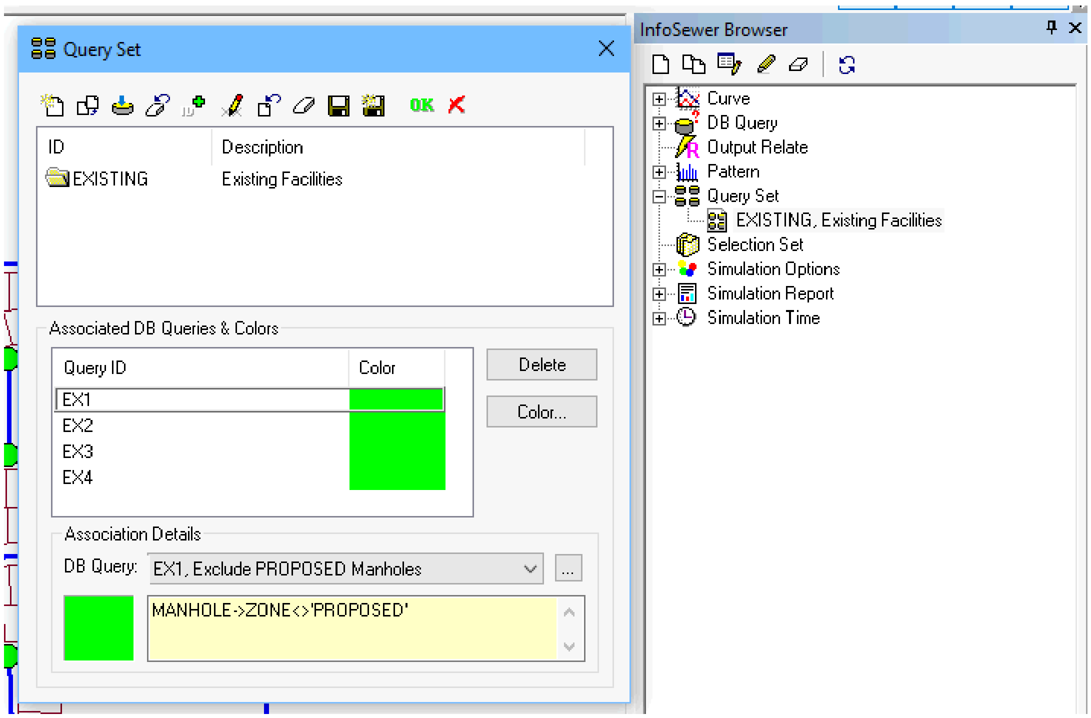

Scenario Explorer
Activate
New Scenario
Network Data Scenario(s)
BASE, Base Network Scenario
FUTURE1, Future Loadings + No New Development
General
Facility
Data Set
Category
Final Data Set
Manhole Set
FUTURE1
Wet Well Set
BASE
Pipe Set
BASE
Pump Set
BASE
Pump Control Set
BASE
Extra Loading Set
BASE
Flow Split Set
BASE
Pipe Design Set
BASE
Pipe Infiltration Set
BASE
Operation Set
BASE
: Inherited Data Set
: Scenario Specific Data Set

Create a unique facility set excluding proposed elements

The facility set defines which portions of your network will be included with a given scenario. You will now define a facility set representing existing facilities. Earlier in this scenario tutorial you added a manhole and pipe and flagged those elements with a ZONE value of "PROPOSED". Using an InfoSewer Pro QUERY SET (collection of database query statements), you can exclude the proposed components.

1. On the *SCENARIO EXPLORER* dialog box with the FUTURE1 scenario selected, choose the *FACILITY* tab. When the Facility panel appears, choose the *QUERY SET* option. One query set has been prepared for you, entitled "**EXISTING – Existing Facilities**". Choose this query set.



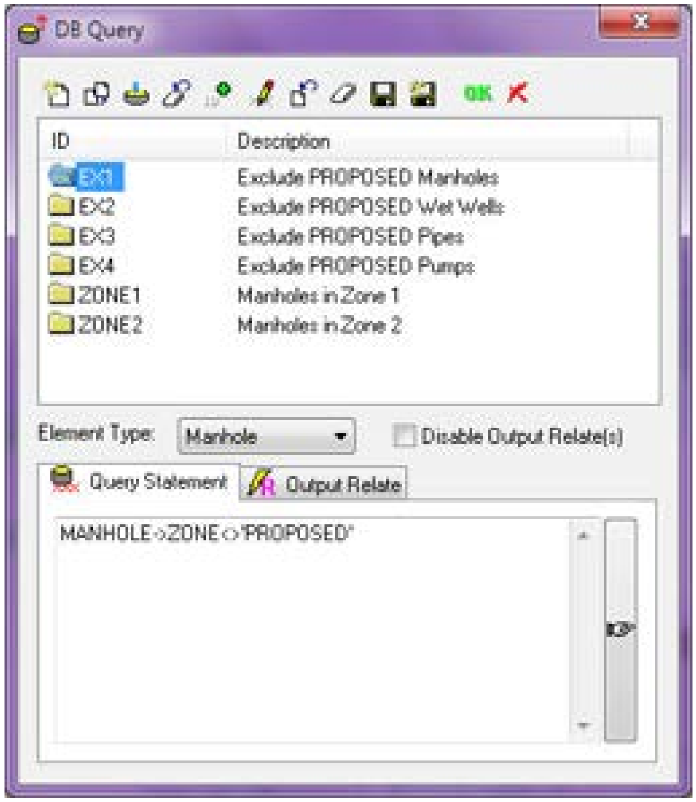


2. It is suggested that you review the contents of this query set before continuing. Choose the “Browse” button to the right of the drop-down list to open the *QUERY SET* dialog box. You will note that the query set is comprised of four individual query statements entitled “EX1” through “EX4”. Examine the first query statement by clicking once on “EX1” in the *QUERY ID* panel on the dialog box and then choosing the “Browse” button in the extreme right-hand corner to open the *DB QUERY* dialog box. You can now examine the contents of the query. The query should appear as (the symbol “<>” designates “not equal to”):

MANHOLE®ZONE <> 'PROPOSED'

The other three queries have an identical structure and represent the other three InfoSewer Pro network component types (pipes, pumps, and wet wells).

3. Close the *DB QUERY* and *QUERY SET* dialog boxes by clicking on the “OK” button and return to the *SCENARIO EXPLORER* dialog box.

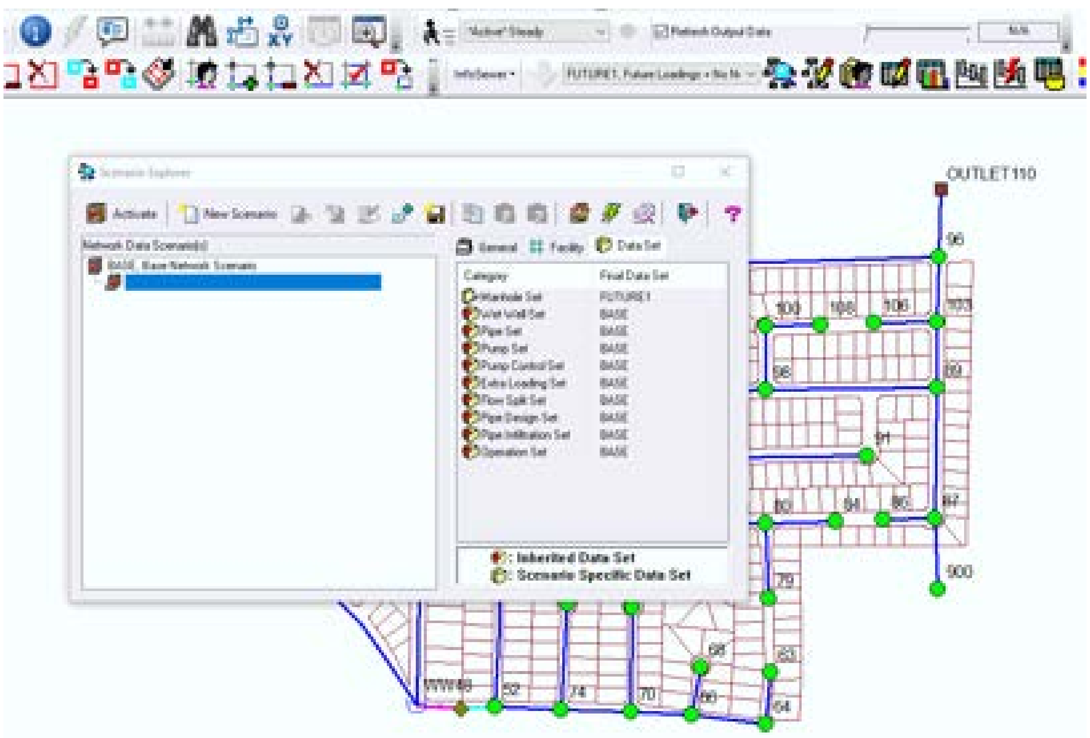


4. Before you close the *SCENARIO EXPLORER* dialog box by clicking on the "OK" button, note that the BASE scenario remains active (loaded) as indicated by the red check .

Step 3: Activate the Future1 Scenario

You can activate a scenario directly from the SCENARIO EXPLORER dialog box or from the InfoSewer Pro toolbar. The latter is illustrated here.

1. Choose the FUTURE1 scenario from the drop-down list on the toolbar and click on the “OK” button on the CONFIRMATION dialog box.



When you activate the FUTURE1 scenario, InfoSewer Pro applies the database query statements in the “EXISTING – Existing Facilities” QUERY SET to include and exclude network facilities in the model. Notice that when you load the FUTURE1 scenario the proposed facilities are removed from the display. They remain in the project but are deactivated for the current scenario. Finally, InfoSewer Pro loads your custom manhole set FUTURE1.

Adjust baseline loadings

Now that your custom manhole set is loaded and active, you can adjust those loadings to represent future sewer loadings without overwriting your existing loadings, which are stored in the BASE manhole set.

1. Choose the DOMAIN®DOMAIN MANAGER command from the TOOLS menu. When the DOMAIN MANAGER dialog box appears on the screen, perform the following:

Choose the NETWORK option and then choose ALL MANHOLES to add all active Another useful fore the current scenario) network components to the domain.

1. Choose the “Add” button .
2. Check the “Apply to Active Facility Only” option 

.
3. Choose the “Close” button 

. All active network components will now be highlighted in red.

2. Click on the “DB Editor” button  from the InfoSewer Pro toolbar.

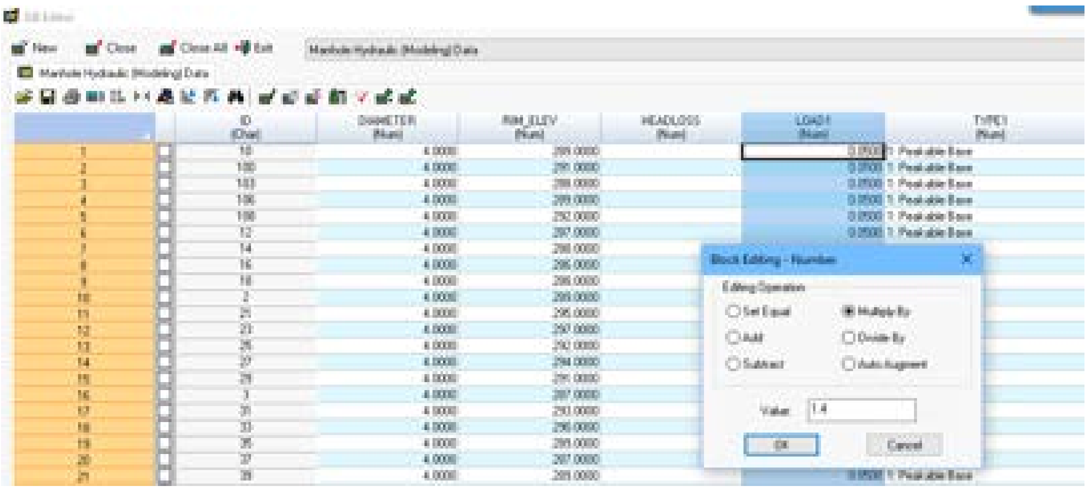
3. Choose the MANHOLE HYDRAULIC (MODELING) DATA table in the ELEMENT HYDRAULIC DATA section, select DOMAIN in the DISPLAY SCOPE area (indicating you want only active manholes; the proposed manhole will not be included and displayed for editing), and choose “OK”. The MANHOLE HYDRAULIC (MODELING) DATA table appears on the screen.

4. To adjust the contents of an entire field, in this case LOAD1, click once on the column header with the mouse. The entire column should be highlighted.

5. You adjust all baseline loadings by increasing the current loads by a factor of 1.4. Choose the “Block Editing” button  while the LOAD1 field remains highlighted. When the BLOCK EDITING dialog box appears on the screen, choose the MULTIPLY BY operation, enter “1.4” for the value, then choose “OK”. All loadings should now be increased by a factor of 1.4.

6. Select the “Save” button and then choose the “Close” button at the top of the DB EDITOR dialog box.

7. Close the DB EDITOR dialog box by selecting the “Exit” button at the top of the DB EDITOR dialog box.



8. Clear the domain by opening the DOMAIN MANAGER dialog box and then clicking on the “Reset” button. Close the DOMAIN MANAGER dialog box. All network components revert to their default colors. You can alternatively use the “Clear Domain” button .

Step 4: Run the Model

Now run an EPS simulation while the FUTURE1 scenario is active.

1. Choose the RUN MANAGER command 

from the TOOLS menu. The RUN MANAGER dialog box appears on the screen. Choose the EPS / DYNAMIC tab.

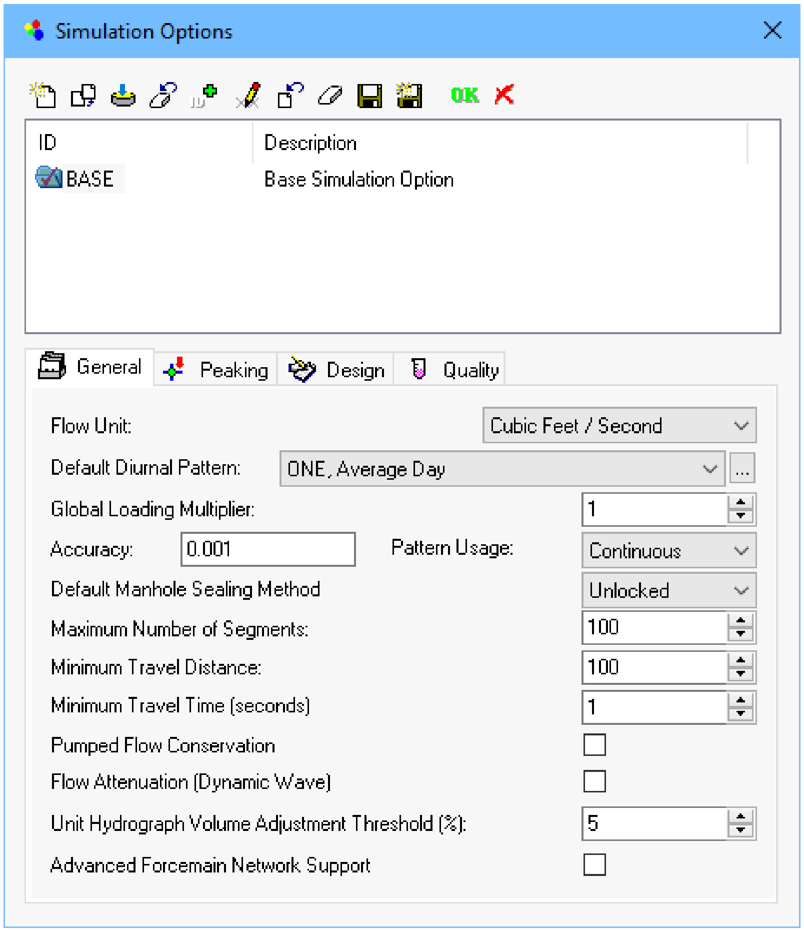
2. Click on the "Run" button 

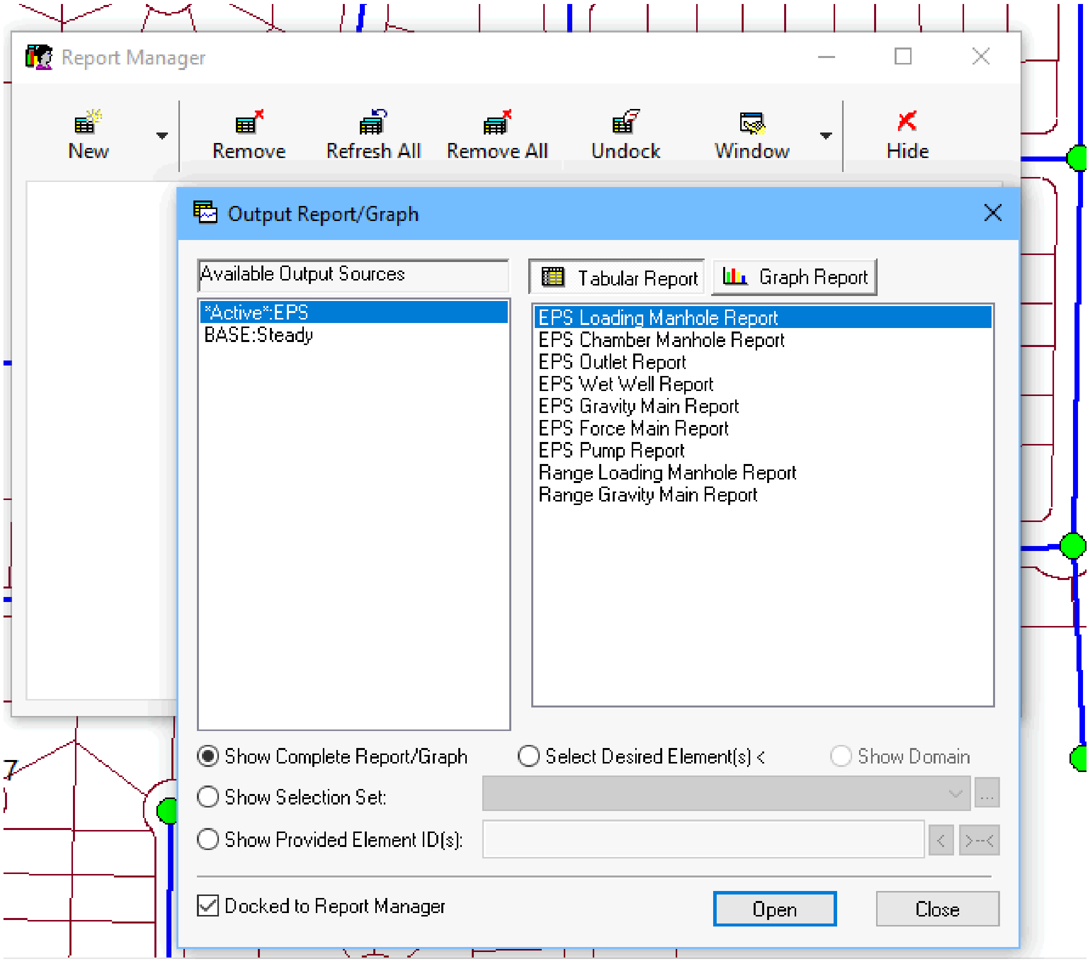
. Upon successful completion of the simulation, the Output Source Status Indicator (the stoplight) should show green.

3. Close the RUN MANAGER dialog box.

Note that the NAME of the OUTPUT SOURCE that will store the simulation results is *Active*:EPS. The Active Standard Output Source will store the results of the FUTURE1 scenario simulation. Note that the FUTURE1 scenario will use the same simulation options and time settings (duration, timesteps) as the BASE, or default, scenario.

Although not instructed to do so as part of this tutorial, you could associate a unique set of simulation options and times with this custom scenario.


Simulation Options
OK
ID
Description
BASE
Base Simulation Option
General
Peaking
Design
Quality
Flow Unit:
Cubic Feet / Second
Default Diurnal Pattern:
ONE, Average Day
Global Loading Multiplier:
1
Accuracy:
0.001
Pattern Usage:
Continuous
Default Manhole Sealing Method
Unlocked
Maximum Number of Segments:
100
Minimum Travel Distance:
100
Minimum Travel Time (seconds)
1
Pumped Flow Conservation
Flow Attenuation (Dynamic Wave)
Unit Hydrograph Volume Adjustment Threshold (%):
5
Advanced Forcemain Network Support


Report Manager
New
Remove
Refresh All
Remove All
Undock
Window
Hide
Output Report/Graph
Available Output Sources
Active:EPS
BASE:Steady
Tabular Report
Graph Report
EPS Loading Manhole Report
EPS Chamber Manhole Report
EPS Outlet Report
EPS Wet Well Report
EPS Gravity Main Report
EPS Force Main Report
EPS Pump Report
Range Loading Manhole Report
Range Gravity Main Report
Show Complete Report/Graph
Select Desired Element(s) <
Show Domain
Show Selection Set:
Show Provided Element ID(s):
Docked to Report Manager
Open
Close

Review model results for the first custom scenario

Review the model output using Map Display or the Map Display Icon. 

Review the results of the simulation using the OUTPUT REPORT MANAGER and the MAP DISPLAY tab  from the InfoSewer View Menu as described in the previous section.

Now you will create the next scenario representing future loadings, incorporating the new development and then run a simulation with that scenario.

OUTLET 110
WW-5
WW46
Map Display
Element Type
Manhole
Chamber
Outlet
Wetwell
Pipe
Forcemain
Pump
Classes
Label Properties
Data Source
Database
Active Output
Rendering Method
Colors
Symbol Sizes
From:
To:
Ramp Color
Data Field
FLOW
Classes:
Classify...
Set Breaks
Color
Size
Break
Label
Open
Save
OK
Cancel
Apply

Step 5: Create the New Scenario

Your second custom scenario will contain the following:

- Existing and proposed facilities – This scenario will include the existing facilities from the original model plus the pipe and manhole representing the proposed development.
- **Future loadings** – This scenario will use the same loadings as the first custom scenario.

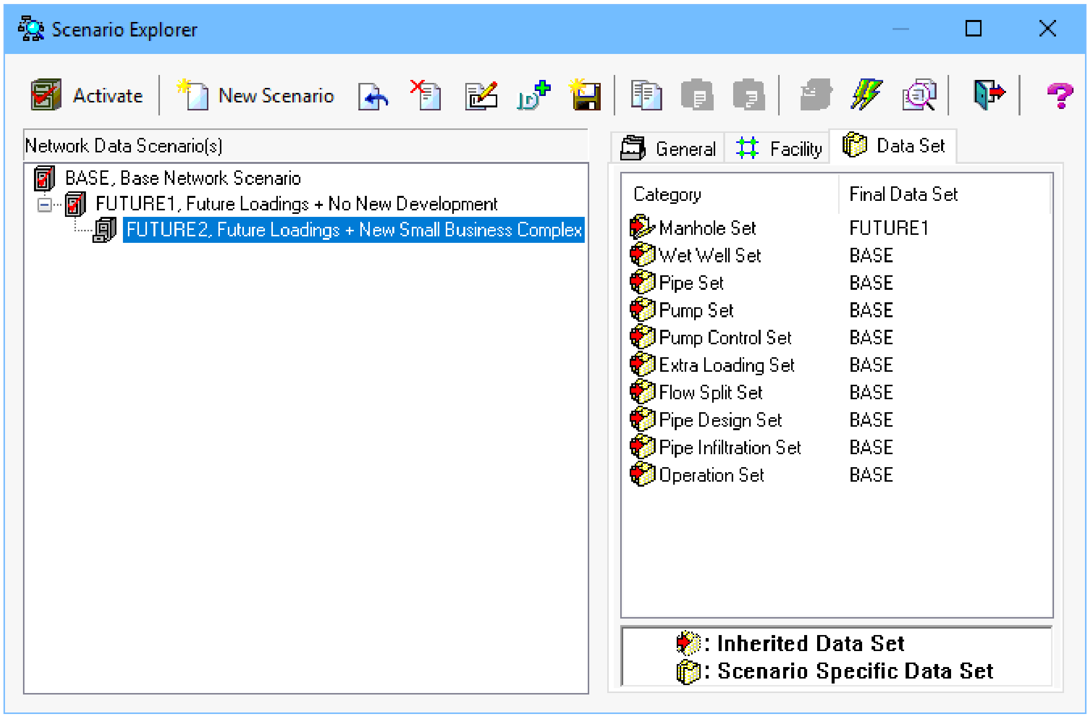
This new scenario will inherit its properties from the first custom scenario. If you were to modify the loadings in the first custom scenario, the loadings in this new scenario would also be modified.

CREATE THE NEW SCENARIO

1. Choose the *SCENARIO EXPLORER* command from the *SCENARIO* menu. The *SCENARIO EXPLORER* dialog box appears on the screen.

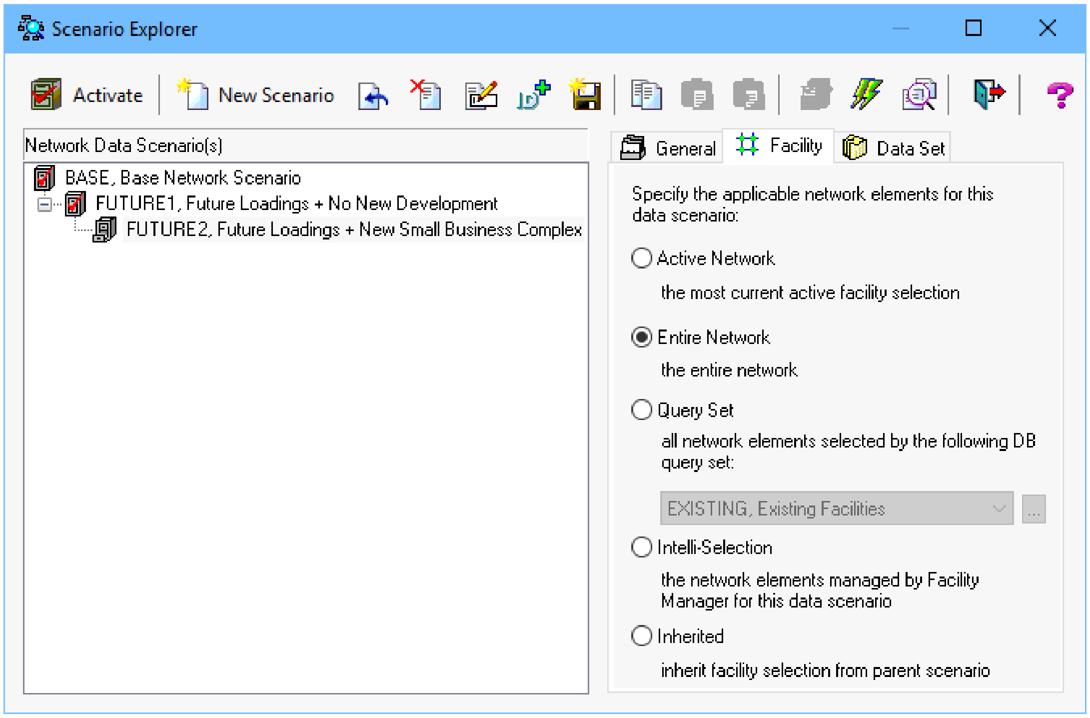
2. Choose the first custom scenario FUTURE1 by clicking once on the scenario name.

3. Choose the “New Child” button  (second from the left at the top left-hand corner of the dialog box). When prompted, enter the name and description for the new scenario, “**FUTURE2, Future Loadings + New Small Business Complex**”.



4. Click then on the "OK" button of the *NEW DATA SCENARIO* dialog box. Because you created the FUTURE2 scenario as a new child of the FUTURE1 scenario, FUTURE2 inherits its properties (data sets) from the FUTURE1 scenario, including the manhole set storing future loadings. Note, however, that scenarios **do not** inherit their facility set (network components) from their parent. Therefore you must explicitly define the network components that will be activated for modeling when you activate (load) the FUTURE2 scenario. The FUTURE2 scenario will include the entire suite of network components, existing **and** proposed.

5. On the *SCENARIO EXPLORER* dialog box, choose the *FACILITY* tab. When the *FACILITY* panel appears, choose the *ENTIRE NETWORK* option. The *SCENARIO EXPLORER* dialog box should appear as follows.



Scenario Explorer
Activate
New Scenario
Network Data Scenario(s)
BASE, Base Network Scenario
FUTURE1, Future Loadings + No New Development
FUTURE2, Future Loadings + New Small Business Complex
General
Facility
Data Set
Specify the applicable network elements for this data scenario:
Active Network
the most current active facility selection
Entire Network
the entire network
Query Set
all network elements selected by the following DB query set:
EXISTING, Existing Facilities
Intelli-Selection
the network elements managed by Facility Manager for this data scenario
Inherited
inherit facility selection from parent scenario

Step 6: Activate the Future 2 Scenario

You can now run a simulation using data from the second scenario.

You can activate a scenario directly from the SCENARIO EXPLORER dialog box or from the InfoSewer Pro toolbar. The former is illustrated here.

1. Choose the SCENARIO EXPLORER command from the SCENARIO menu. The SCENARIO EXPLORER dialog box appears on the screen.

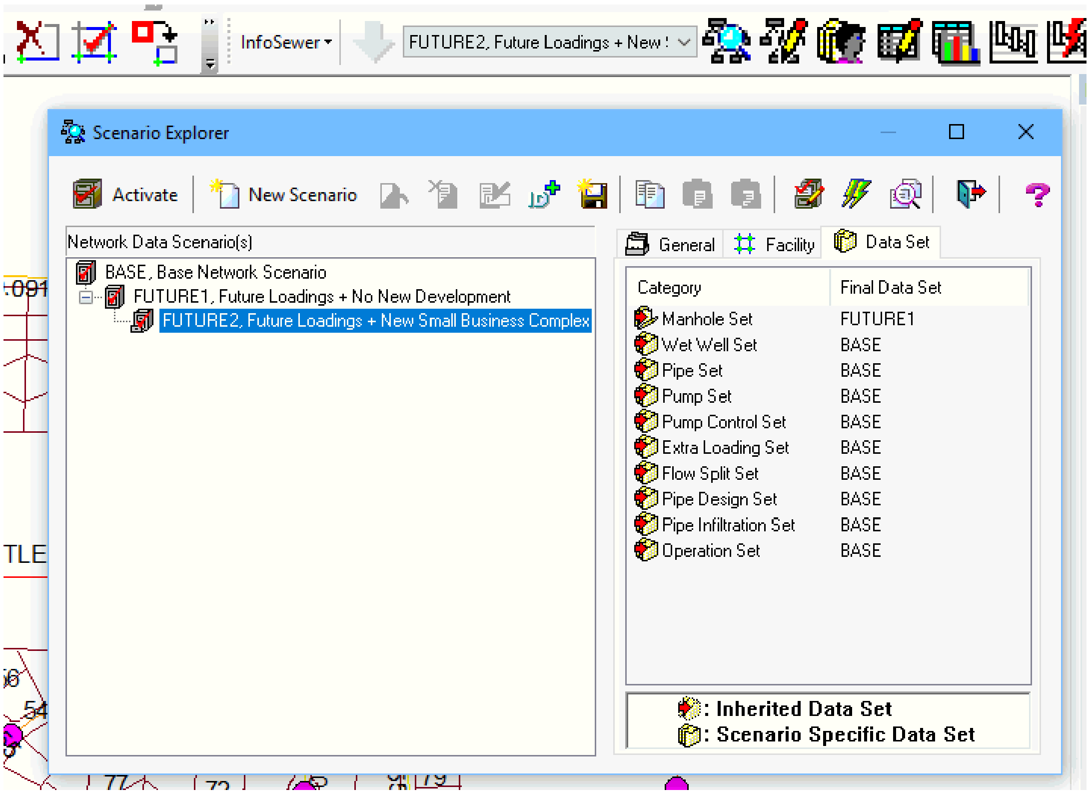
2. Select the second scenario FUTURE2 by clicking once on the scenario name.

3. Click once on the “Activate” button 

. The second scenario is now activated.

4. Click once on the “OK” button to close the SCENARIO EXPLORER dialog box.

When you activate the FUTURE2 scenario, InfoSewer Pro activates all existing and proposed facilities. Notice on the map display that the pipe and manhole added at the beginning of this tutorial re-appear on the screen. No additional data modification is necessary. You may now run a simulation using the FUTURE2 scenario.


InfoSewer
FUTURE2, Future Loadings + New S
Scenario Explorer
Activate
New Scenario
Network Data Scenario(s)
BASE, Base Network Scenario
FUTURE1, Future Loadings + No New Development
FUTURE2, Future Loadings + New Small Business Complex
General
Facility
Data Set
Category
Final Data Set
Manhole Set FUTURE1
Wet Well Set BASE
Pipe Set BASE
Pump Set BASE
Pump Control Set BASE
Extra Loading Set BASE
Flow Split Set BASE
Pipe Design Set BASE
Pipe Infiltration Set BASE
Operation Set BASE
: Inherited Data Set
: Scenario Specific Data Set

Step 7: Run the Model

Now run an EPS simulation while the FUTURE2 scenario is active.

1. Choose the RUN MANAGER command  from the TOOLS menu. The RUN MANAGER dialog box appears on the screen. Choose the EPS / DYNAMIC tab.

2. Click on the "Run" button 

. Upon successful completion of the simulation, the Output Source Status Indicator (the stoplight) should show green.

3. Close the RUN MANAGER dialog box.

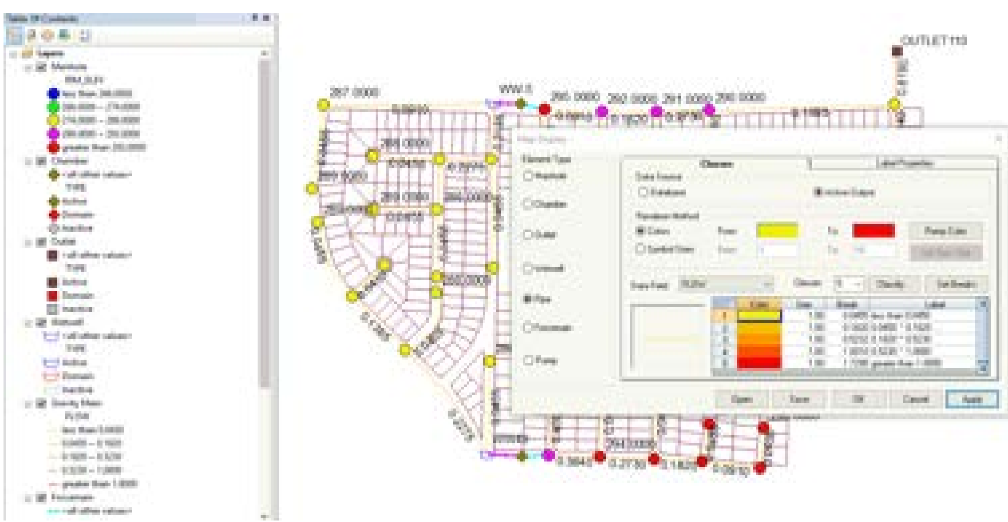
Note the NAME of the OUTPUT SOURCE that will store the simulation results is *Active*:EPS. The Active Standard Output Source will store the results of the FUTURE2 scenario simulation.

The previous simulation results, those associated with the FUTURE1 scenario, are automatically moved to a new OUTPUT SOURCE entitled FUTURE1:EPS.

Review model results for the second custom scenario

Review the model output using Map Display or the Map Display Icon. 

Review the results of the simulation using the OUTPUT REPORT MANAGER and the MAP DISPLAY tab  from the View Menu as described in the



Step 8: Create the New Scenario

Your third custom scenario will contain the following:

- Existing and proposed facilities – This scenario will include the existing facilities from the original model plus the pipe and manhole representing the proposed development.
- Future loadings plus additional loads for an expanded development – This scenario will use the same future loads as the previous scenarios, however you will increase the loading at the new development from 0.20 cfs to 0.35 cfs. For this you will need to create another new manhole set.

This new scenario will inherit its properties from the <u>first</u> custom scenario. If you were to modify the loadings in the first custom scenario (FUTURE1), the loads in this new scenario would also be modified.

1. Choose the *SCENARIO EXPLORER* command from the *SCENARIO* menu. The *SCENARIO EXPLORER* dialog box appears on the screen.

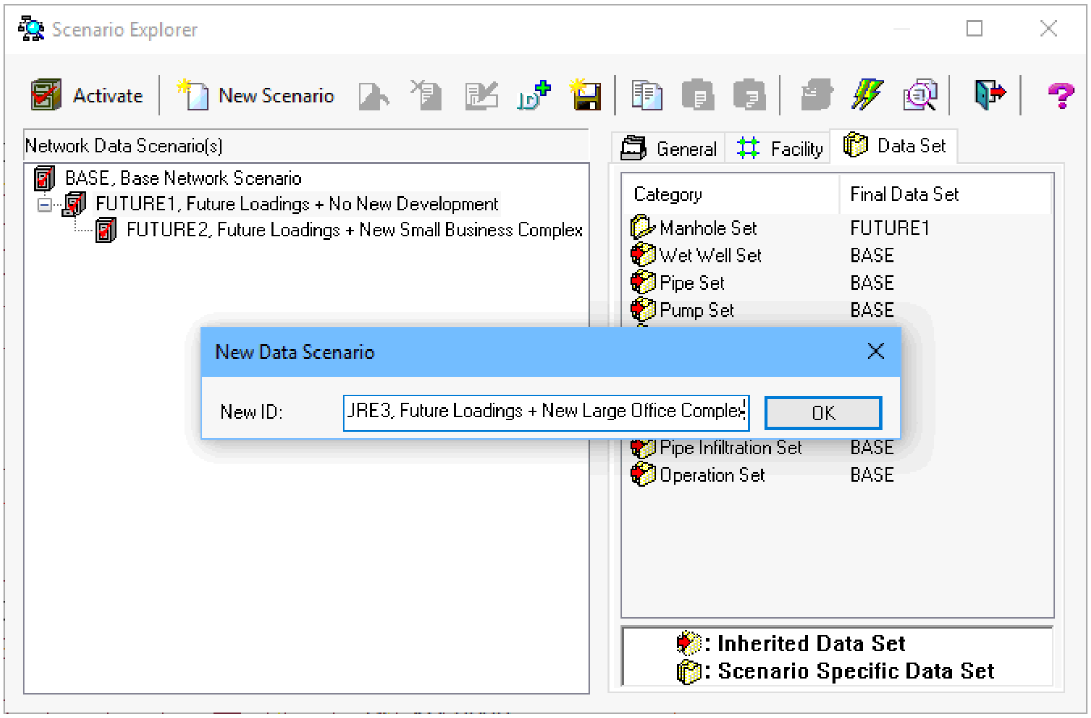
2. Choose the first custom scenario FUTURE1 by clicking once on that scenario name with the mouse.

3. Choose the “New Child” button  (second from the left at the top left-hand corner of the dialog box). When prompted, enter the name and description for the new scenario, “**FUTURE3, Future Loadings + New Large Office Complex**”. Once complete, click on the “OK” button of the *NEW DATA SCENARIO* dialog box.

4. Because you created the FUTURE3 scenario as a new child of the FUTURE1 scenario, FUTURE3 inherits its properties (data sets) from the FUTURE1 scenario, including the manhole set storing future loadings. However, you now need to create and activate a new manhole set to store the future loads for the entire network in addition to the increased loading at the new development.

5. Before creating the new manhole set, indicate that the new scenario will utilize the entire suite of network components. On the *SCENARIO EXPLORER* dialog

box, choose the *FACILITY* tab. When the *FACILITY* panel appears, choose the *ENTIRE NETWORK* option.



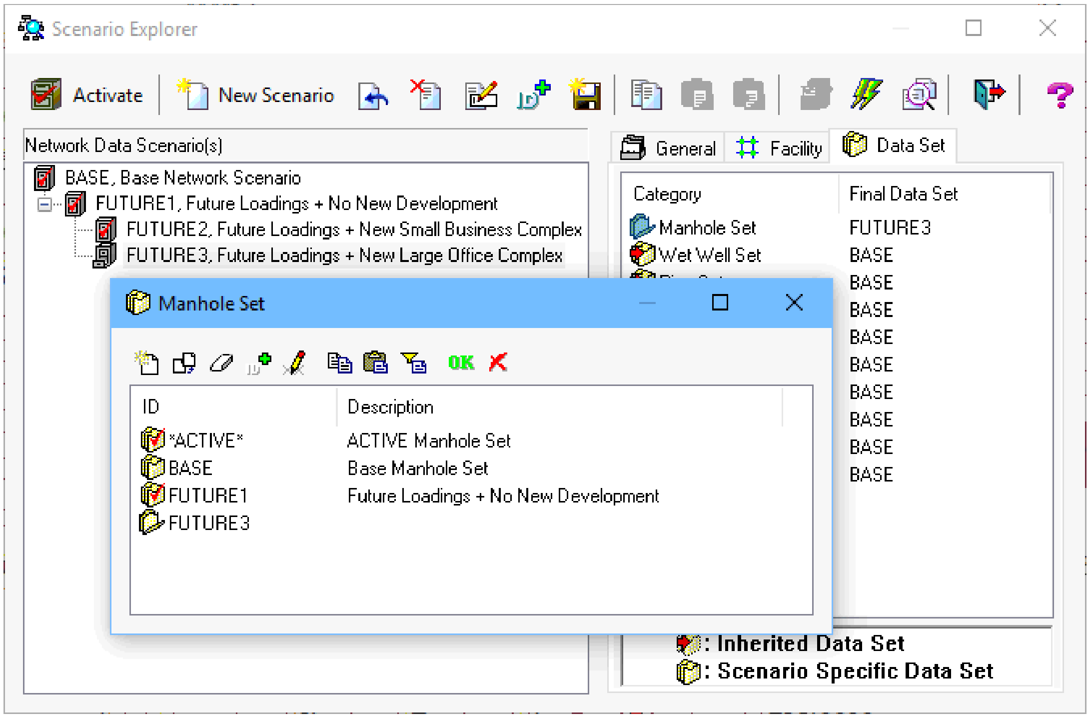
Create a unique Manhole set to store increased Loading at the proposed development

1. While the SCENARIO EXPLORER dialog box is open and the new FUTURE3 scenario is selected, click on the DATA SET tab and then double-click on MANHOLE SET. The MANHOLE SET dialog box appears on the screen.

2. You will clone the active manhole set (FUTURE1), which contains the future loadings – manhole node 900. Click on the “Clone” button (second from the left at the top of the MANHOLE SET dialog box). When prompted, enter the name and description of the new manhole set “FUTURE3, New Large Office Complex”. Click on the “OK” button of the NEW MANHOLE SET dialog box.

3. Choose the “OK” button to close the MANHOLE SET dialog box and return to the SCENARIO EXPLORER dialog box. You will now note that the FUTURE3 manhole set is associated with the FUTURE3 scenario.

4. Close the SCENARIO EXPLORER dialog box by selecting the “OK” button. Before closing, the dialog should now appear as follows.



Scenario Explorer
Activate
New Scenario
Network Data Scenario(s)
BASE, Base Network Scenario
FUTURE1, Future Loadings + No New Development
FUTURE2, Future Loadings + New Small Business Complex
FUTURE3, Future Loadings + New Large Office Complex
General
Facility
Data Set
Category
Final Data Set
Manhole Set
FUTURE3
Wet Well Set
BASE
Manhole Set
OK
ID
Description
ACTIVE
ACTIVE Manhole Set
BASE
Base Manhole Set
FUTURE1
Future Loadings + No New Development
FUTURE3
: Inherited Data Set
: Scenario Specific Data Set

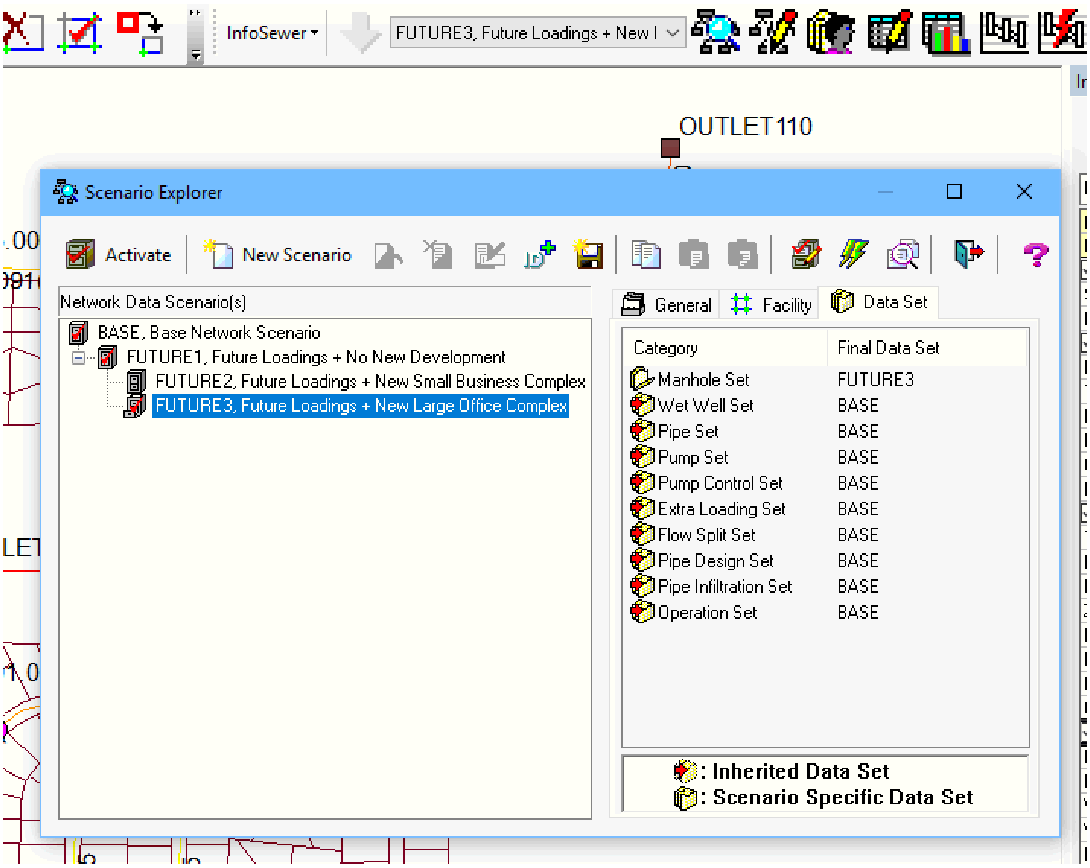
Step 9: Activate the future3 scenario

We will now load and activate the third scenario - FUTURE3.

You can activate a scenario directly from the SCENARIO EXPLORER dialog box or from the InfoSewer Pro toolbar. The latter is illustrated here.

1. Choose the FUTURE3 scenario from the drop-down list on the InfoSewer Pro toolbar and select "OK" when the CONFIRMATION dialog box appears.

When you activate the FUTURE3 scenario, InfoSewer Pro activates all existing and proposed facilities. Moreover, InfoSewer Pro loads your custom manhole set "FUTURE3, New Large Office Complex", which at this point in the tutorial contains the future loading values. The new proposed manhole will need to be assigned an appropriate sewer loading.



Increase Loading a the New Development

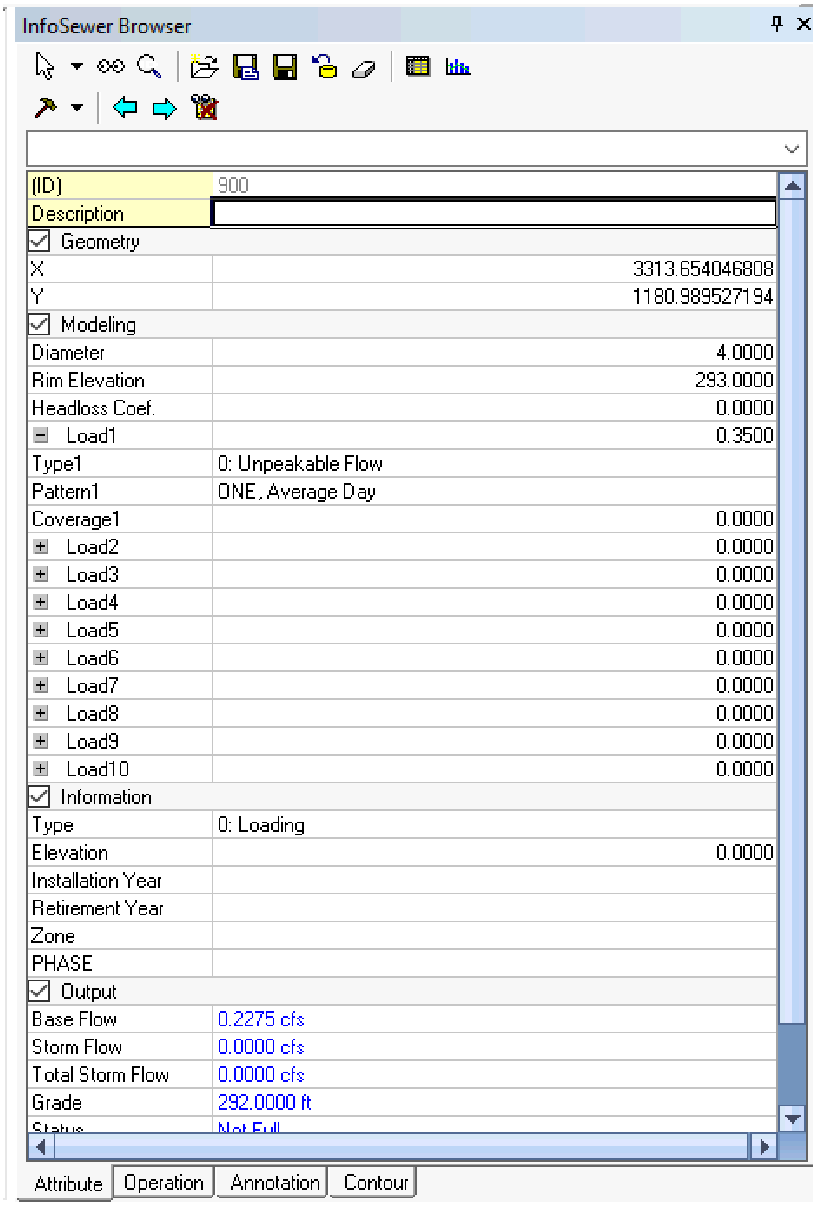
Before continuing, increase the loading at manhole 900 from 0.20 cfs to 0.35 cfs:

1. Click on the “Select” button , place the cursor on the manhole node 900 and then press the left mouse button.

2. In the Modeling section of the ATTRIBUTE BROWSER-NODE window, click once on the LOAD1 field and enter “0.35”.

3. Finally, click once on the “Save” button  in the ATTRIBUTE BROWSER-NODE window.

The adjusted loading is automatically saved in the active manhole set, “FUTURE3, New Large Office Complex”.


InfoSewer Browser
(ID)
900
Description
Geometry
X
3313.654046808
Y
1180.989527194
Modeling
Diameter
4.0000
Rim Elevation
293.0000
Headloss Coef.
0.0000
Load1
0.3500
Type1
0: Unpeakable Flow
Pattern1
ONE, Average Day
Coverage1
0.0000
Load2
0.0000
Load3
0.0000
Load4
0.0000
Load5
0.0000
Load6
0.0000
Load7
0.0000
Load8
0.0000
Load9
0.0000
Load10
0.0000
Information
Type
0: Loading
Elevation
0.0000
Installation Year
Retirement Year
Zone
PHASE
Output
Base Flow
0.2275 cfs
Storm Flow
0.0000 cfs
Total Storm Flow
0.0000 cfs
Grade
292.0000 ft
Attribute
Operation
Annotation
Contour

Step 10: Run a model using the third custom scenario

Now run an EPS simulation while the FUTURE3 scenario is active.

1. Choose the RUN MANAGER command  from the TOOLS menu. The RUN MANAGER dialog box appears on the screen. Choose the EPS / DYNAMIC tab.

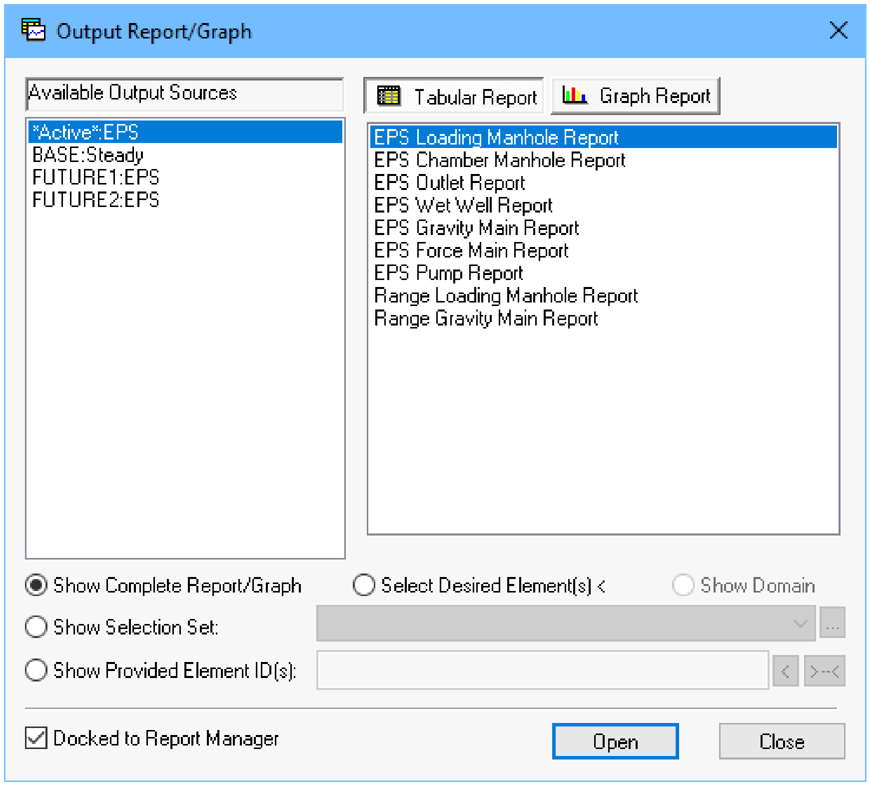
2. Click on the “Run” button . Upon successful completion of the simulation, the Output Source Status Indicator (the stoplight) should show green.

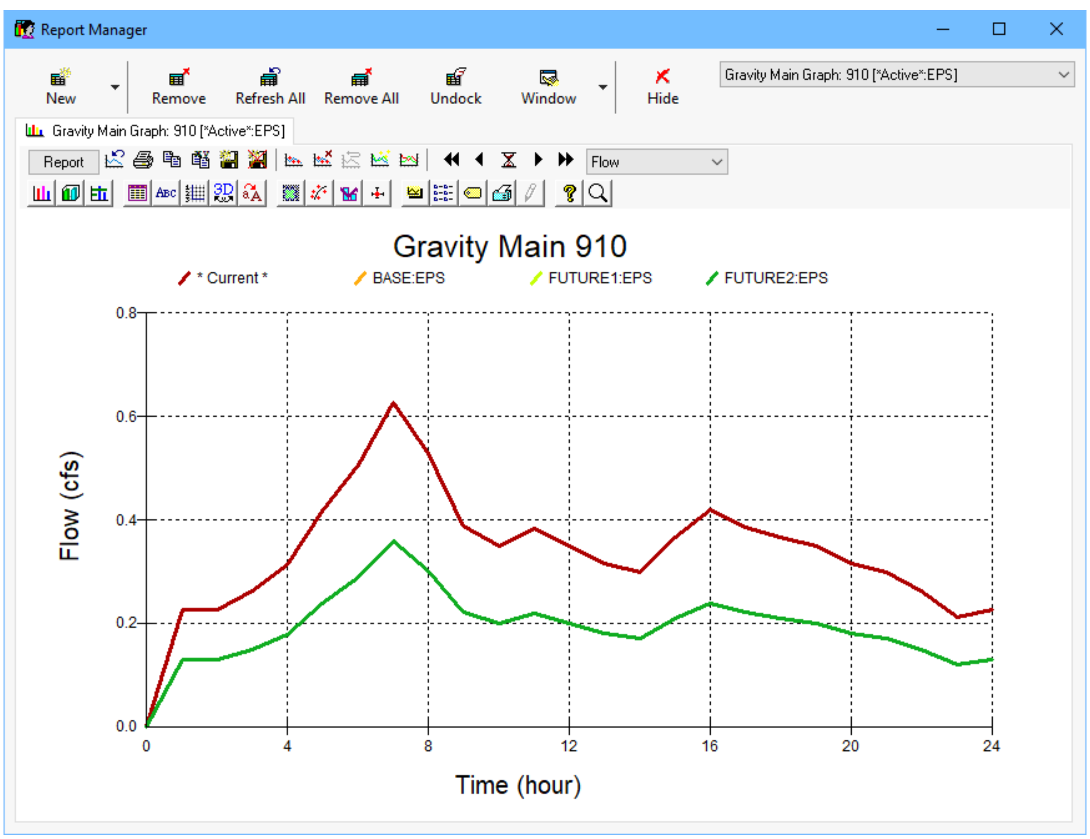
3. Close the RUN MANAGER dialog box.

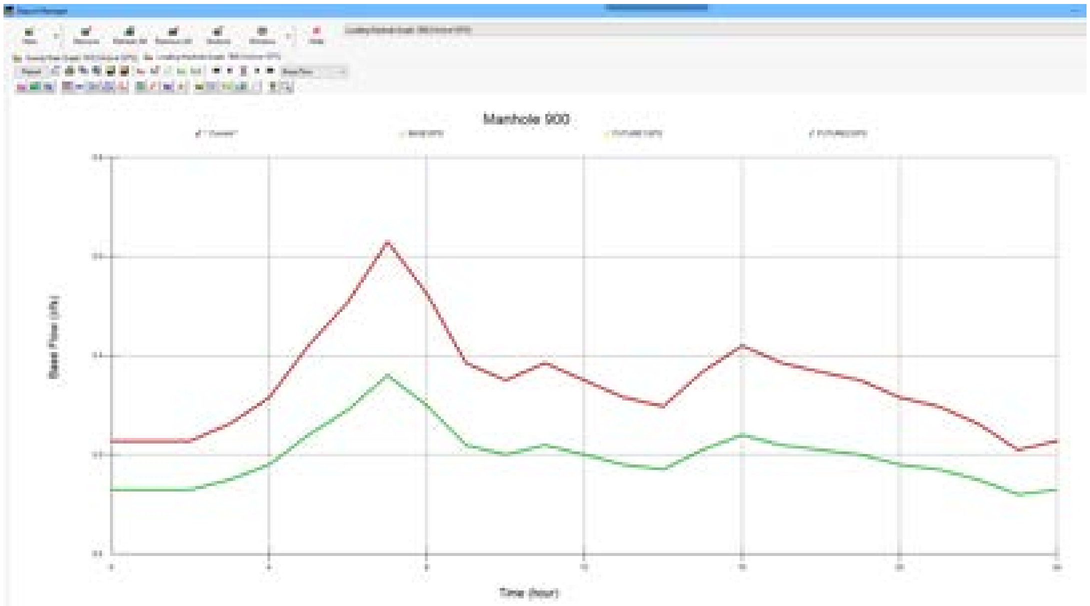
Note the NAME of the OUTPUT SOURCE that will store the simulation results is *Active*:EPS. The Active Standard Output Source will store the results of the FUTURE3 scenario simulation.

The previous simulation results, those associated with the FUTURE2 scenario, are automatically moved to a new *OUTPUT SOURCE* entitled *FUTURE2:EPS*. Upon completion of this simulation, you should have the following Output Sources available:

- ***BASE:EPS*** – The original network, prior to adjusting loadings and adding new facilities.
- ***FUTURE1:EPS*** – The existing network, excluding proposed facilities, with the original loadings increased by a factor of 1.4.
- ***FUTURE2:EPS*** – The entire network **<u>including</u>** proposed facilities and the same loadings as the FUTURE1 scenario with an additional 0.20 cfs at the new development.
- ****Active*:EPS*** – The entire network **<u>including</u>** proposed facilities and the same loadings as the FUTURE1 scenario. For this scenario, the loading at the new development has been increased to 0.35 cfs.


Output Report/Graph
Available Output Sources
Active:EPS
BASE:Steady
FUTURE1:EPS
FUTURE2:EPS
Tabular Report
Graph Report
EPS Loading Manhole Report
EPS Chamber Manhole Report
EPS Outlet Report
EPS Wet Well Report
EPS Gravity Main Report
EPS Force Main Report
EPS Pump Report
Range Loading Manhole Report
Range Gravity Main Report
Show Complete Report/Graph
Select Desired Element(s) <
Show Domain
Show Selection Set:
Show Provided Element ID(s):
Docked to Report Manager
Open
Close


Report Manager
New
Remove
Refresh All
Remove All
Undock
Window
Hide
Gravity Main Graph: 910 [*Active*:EPS]
Report
Flow
Gravity Main 910
* Current *
BASE:EPS
FUTURE1:EPS
FUTURE2:EPS
Flow (cfs)
0.0
0.2
0.4
0.6
0.8
0
4
8
12
16
20
24
Time (hour)

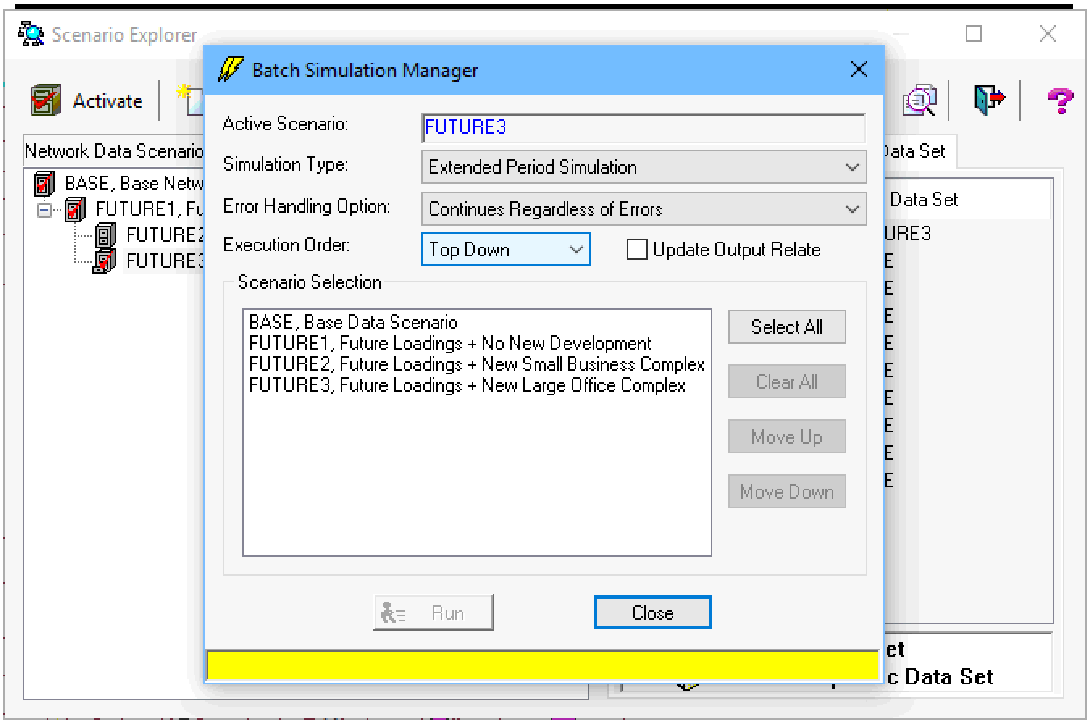


Manhole 900
Base Flow (cfs)
Time (hour)

Review model results for the second custom scenario

Review the results of the simulation using the OUTPUT REPORT MANAGER and the MAP DISPLAY tab  from the View Menu as described in the previous section.

NOTE: You can also run multiple scenarios simultaneously using the Batch Sim*ulation option under the Tools menu.*



Step 11: Open a graph for the proposed development at PIPE 910

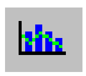
- **1.** Select the *OUTPUT REPORT/GRAPH* command from the *TOOLS* menu or alternately choose the corresponding button  from the InfoSewer Pro toolbar. The *OUTPUT REPORT MANAGER* appears on the screen.

- **2.** On the *OUTPUT REPORT MANAGER* choose the "New" button 

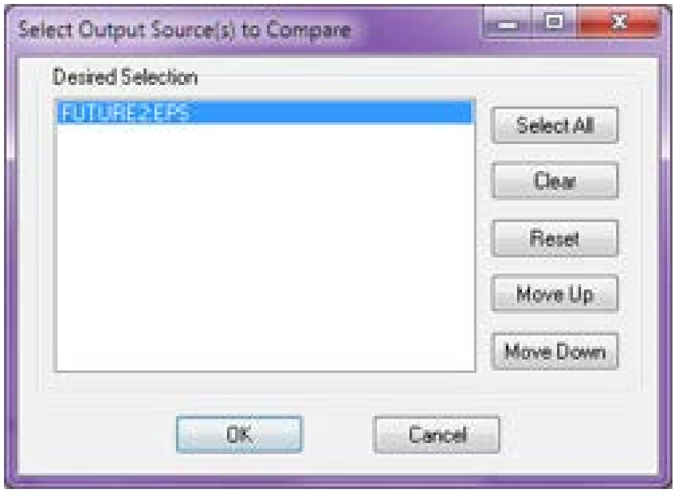
. When the *OUTPUT REPORT & GRAPH* dialog appears on the screen, select ***Active*:EPS*** (containing results from your most recent model run) in the *ALL OUTPUT SOURCES* area.

- **3.** Choose *GRAVITY MAIN GRAPH* in the *GRAPH* tab, and then click on the "Open" button.

- **4.** You are then prompted to select a pipe. Move and place the cursor on the new pipe 910 and press the left mouse button. A graph appears on the screen showing the daily flow variation for the selected pipe.

- ***NOTE**: Alternatively, you can make sure that *Active*:EPS Output Results is selected in the bottom left corner, select Pipe No. 910, and click on the Graph Icon*  *in the upper right corner of the Attribute Browser.*

5. Now load the output data for pipe 910 from the previous scenario run, i.e. **FUTURE2:EPS**. Click on the "Compare Graph" button . The *SELECT OUTPUT SOURCE(S) TO COMPARE* dialog box appears on the screen.

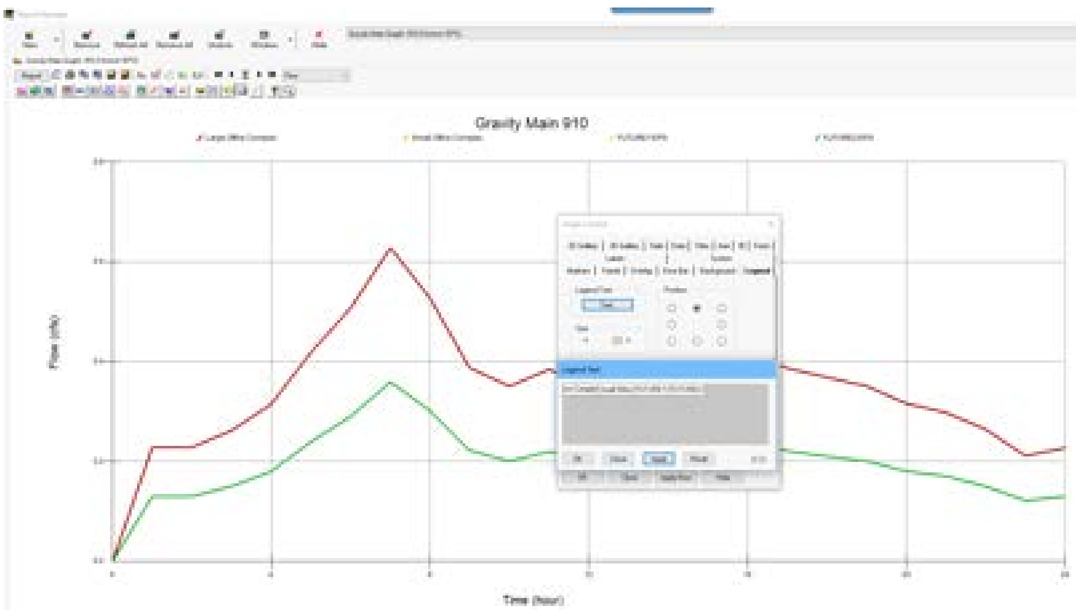


6. Choose the **FUTURE2:EPS** Output Source and then choose "OK". The graph now shows the temporal variation in flow for pipe 910 for both scenarios.

CUSTOMIZE THE GRAPH LEGEND

Finally, customize the graph legend, renaming legend entries such that they are intuitive.

1. While the graph is active, click on the “Legend” button. When the *GRAPH CONTROL* dialog box appears on the screen, click on the “Text” button of the *LEGEND* tab. The *LEGEND TEXT* dialog box appears on the screen.



2. In the left-most cell of the *LEGEND TEXT* dialog box, replace “*Current*” with “**Large Office Complex**”, representing the flow at the new development at 0.35 cfs base loading. In the right-most cell, replace “**FUTURE2:EPS**” with “**Small Business Complex**”. Choose the “OK” button to close the *LEGEND TEXT* dialog box.

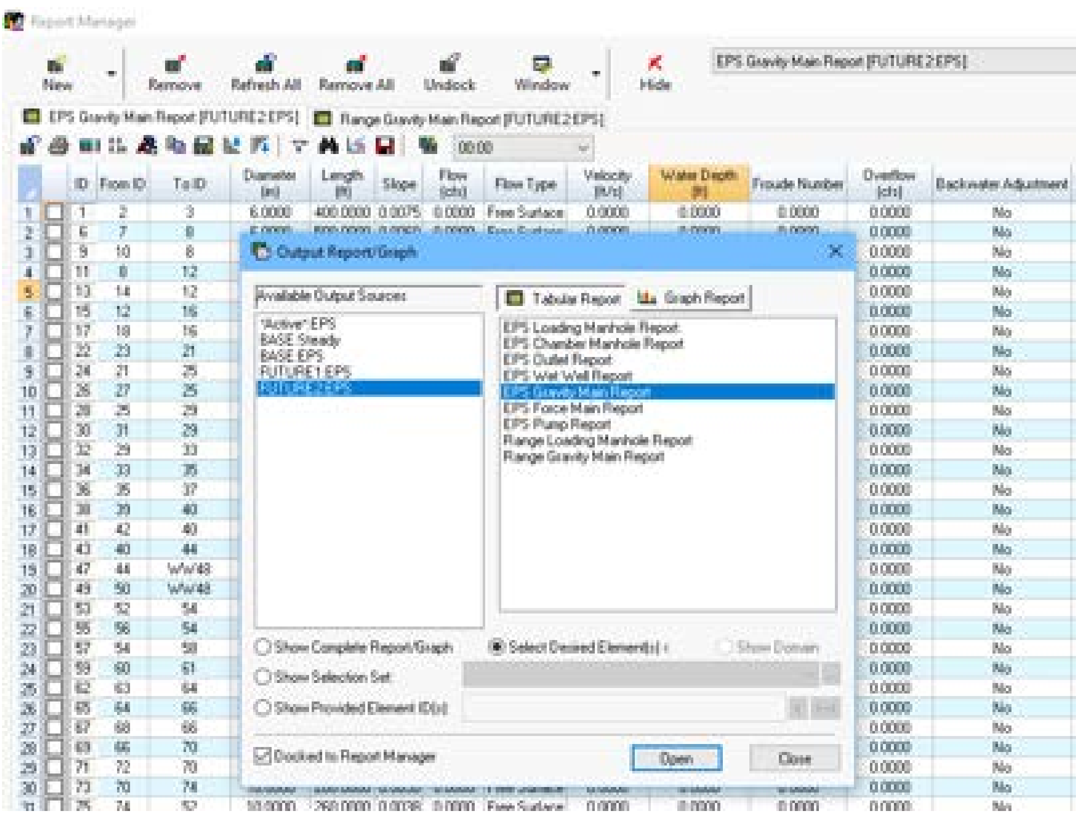
3. Close the *GRAPH CONTROL* by clicking on the “OK” button. The graph legend should now reflect your changes in the *OUTPUT REPORT MANAGER* graph window.

Create a custom output report

1. On the *OUTPUT REPORT MANAGER* choose the “New” button .

2. When the *OUTPUT REPORT & GRAPH* dialog appears on the screen, select the ***Active*:EPS** Output Source, choose *EPS GRAVITY MAIN REPORT* from the *REPORT* tab, and then choose “Open”.

3. The *EPS GRAVITY MAIN REPORT* for the ***Active*:EPS** Output Source (FUTURE3, Future Loadings + New Large Office Complex) appears in the *OUTPUT REPORT MANAGER.* The results are displayed for the first extended period simulation timestep: “00:00 hrs”.

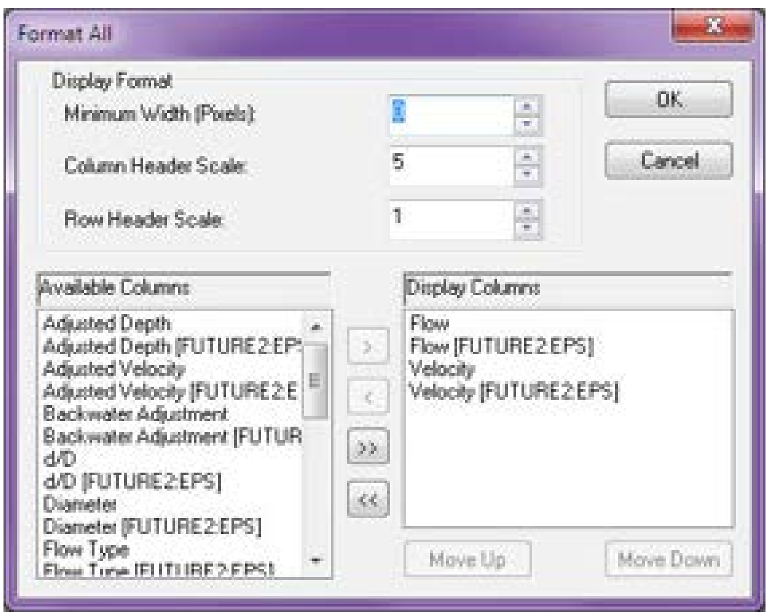


4. Now load the output data from the previous scenario run, i.e. **FUTURE2:EPS**. Click on the “Compare Report” button and the *SELECT OUTPUT*

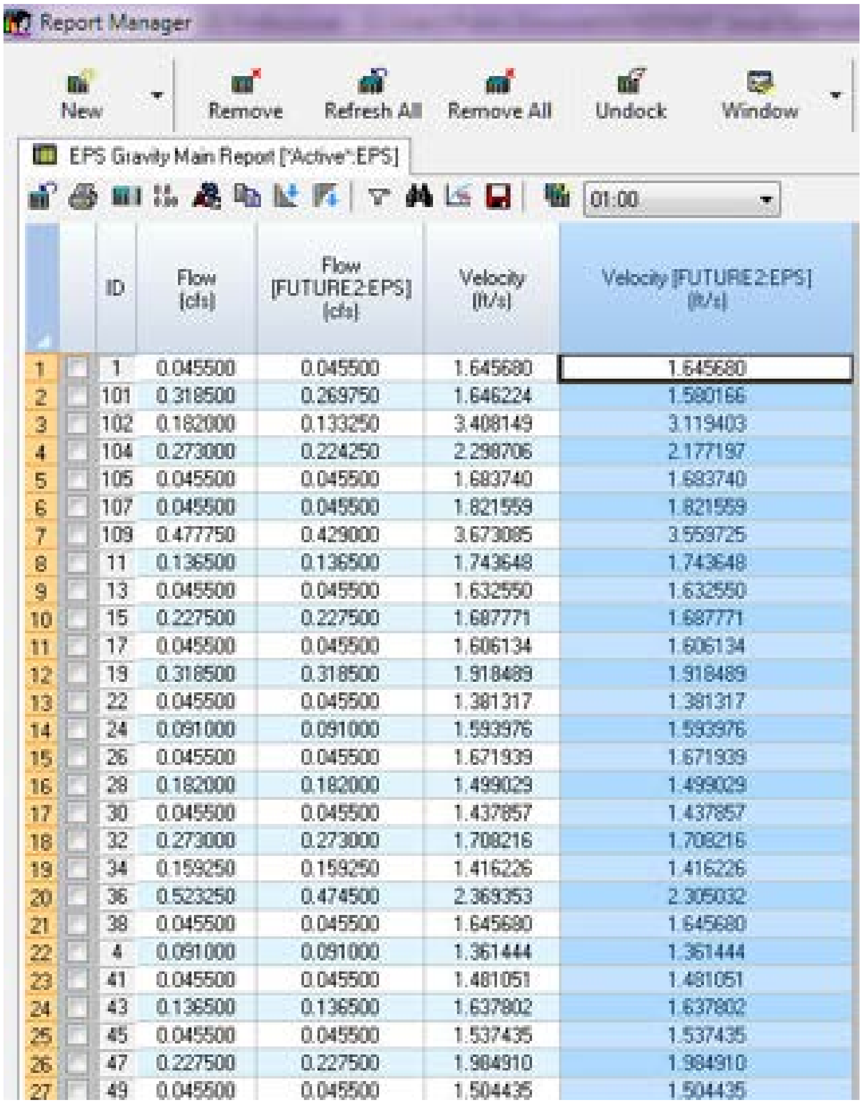
SOURCE(S) TO COMPARE dialog box appears on the screen.

5. Choose the **FUTURE2:EPS** Output Source and then choose “OK”. The report now shows the pipe results for both FUTURE2 and FUTURE3 modeling scenarios.

6. Now customize your report to display only the flow and velocity results for both scenarios. Select the “Format Report” button on the *OUTPUT REPORT MANAGER* and the *FORMAT ALL* dialog box appears on the screen. The *DISPLAY COLUMNS* area initially contains all the output variables for display. Click on the button and the output variables are shifted to the *AVAILABLE COLUMNS* area. Hold down the CTRL key and select “Flow”, “Flow [**FUTURE2:EPS**]”, “Velocity”, and “Velocity [**FUTURE2:EPS**]” then choose the button. The *DISPLAY COLUMNS* area now shows the four selected output variables.



Click on the “OK” button to close the *FORMAT ALL* dialog box. The *PIPE REPORT* now shows the selected pipe results for both FUTURE2 and FUTURE3 modeling scenarios. Click on the “Hide” button.



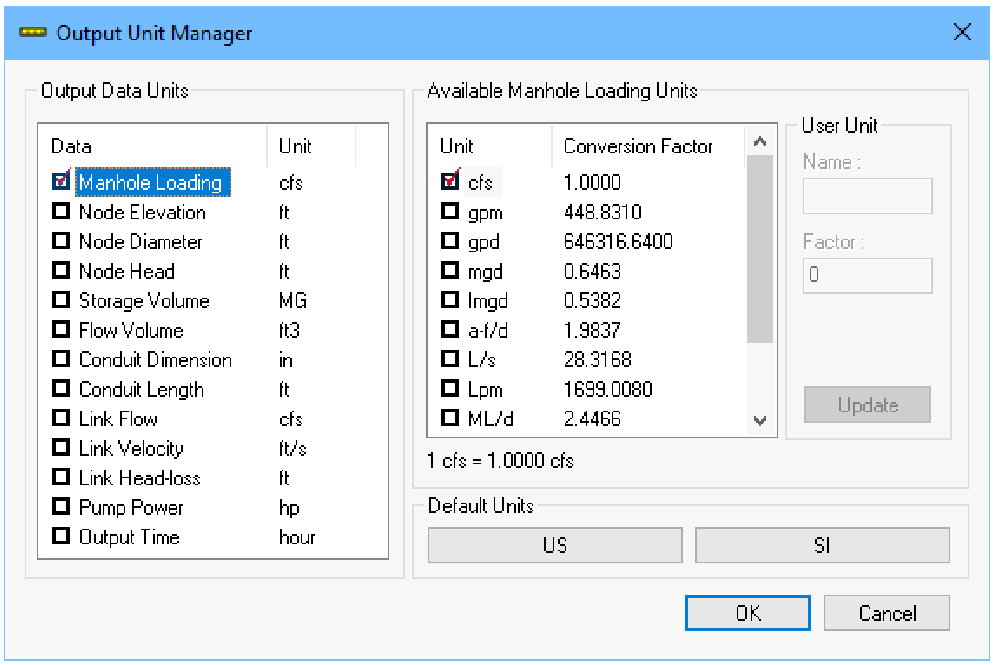
		ID	Flow (cfs)	Flow [FUTURE2:EPS] (cfs)	Velocity (ft/s)	Velocity [FUTURE2:EPS] (ft/s)
1		1	0.045500	0.045500	1.645680	1.645680
2		101	0.318500	0.269750	1.646224	1.580166
3		102	0.182000	0.133250	3.408149	3.119403
4		104	0.273000	0.224250	2.298706	2.177197
5		105	0.045500	0.045500	1.683740	1.683740
6		107	0.045500	0.045500	1.821559	1.821559
7		109	0.477750	0.429000	3.673085	3.559725
8		11	0.136500	0.136500	1.743648	1.743648
9		13	0.045500	0.045500	1.632550	1.632550
10		15	0.227500	0.227500	1.687771	1.687771
11		17	0.045500	0.045500	1.606134	1.606134
12		19	0.318500	0.318500	1.918489	1.918489
13		22	0.045500	0.045500	1.381317	1.381317
14		24	0.091000	0.091000	1.593976	1.593976
15		26	0.045500	0.045500	1.671939	1.671939
16		28	0.182000	0.182000	1.499029	1.499029
17		30	0.045500	0.045500	1.437857	1.437857
18		32	0.273000	0.273000	1.708216	1.708216
19		34	0.159250	0.159250	1.416226	1.416226
20		36	0.523250	0.474500	2.369353	2.305032
21		38	0.045500	0.045500	1.645680	1.645680
22		4	0.091000	0.091000	1.361444	1.361444
23		41	0.045500	0.045500	1.481051	1.481051
24		43	0.136500	0.136500	1.637802	1.637802
25		45	0.045500	0.045500	1.537435	1.537435
26		47	0.227500	0.227500	1.984910	1.984910
27		49	0.045500	0.045500	1.504435	1.504435

Refer to the Reports and Graphing à Output Report Manager topic in the on-line InfoSewer Pro Help for more information on comparison and other custom output reports.

Changing Output Units

With InfoSewer Pro, you can change the units of any output variable in any report and graph display. You can even mix US customary and SI units in the same project if you so desire. For this exercise, we will use the OUTPUT UNIT MANAGER to display the pipe flow in liters per second (L/s) and the pipe velocity in meters per second (m/s).

1. Choose the OUTPUT UNIT MANAGER from the TOOLS menu. The OUTPUT UNIT MANAGER appears on the screen as shown below. The OUTPUT UNIT MANAGER stores the selected output display units.



2. From the *OUTPUT UNIT MANAGER* dialog box, click on the *LINK FLOW* in the *OUTPUT DATA UNITS* area and select “L/s” from the *AVAILABLE LINK FLOW UNITS* area.

3. Similarly, click on the *LINK VELOCITY* in the *OUTPUT DATA UNITS* area and select “m/s” from the *AVAILABLE LINK FLOW UNITS* area. Click on the “OK” button to close the *OUTPUT UNIT MANAGER* dialog box.

4. Go back to the *OUTPUT REPORT MANAGER* and choose the 

button on the dialog box. The report now displays the desired output variable units.

Refer to the Utilities and Options à Output Unit Manager topic in the on-line InfoSewer Pro Help for more information on specifying units for use on graphs and reports.

Conclusion of the Tutorial

Using InfoSewer Pro scenario management capabilities you were able to take a single InfoSewer Pro project, representing your existing collection system under current average day conditions, and create three additional alternative model representations. These represent your existing system under future loading conditions, your existing system with a small proposed development, and the proposed development with a significantly large complex and loading condition. You have seen how, after developing modeling scenarios with the SCENARIO MANAGER, you can quickly retrieve each separate scenario and can easily compare the results between the different scenarios.

Extended Stormwater Modeling

This tutorial will introduce you to InfoSewer Pro's Stormwater (rainfall-runoff) modeling capability. Using InfoSewer Pro, you can model sanitary sewer systems, storm sewer systems, and combined sewer systems. InfoSewer Pro uses the Rational Method during steady state simulations of stormwater. For dynamic (EPS) simulations, the model utilizes the Unit Hydrograph Theory to derive runoff hydrographs. Up to this section of the tutorials, you have been using the sanitary sewer modeling capability of the software. ***Please note the stormwater capabilities described here are only available for InfoSewer Pro licenses.***

Using the "**Tutorial**" project you have been using, you will create and assign additional modeling data, run steady state and dynamic Stormwater simulations, and review modeling results. After completion of this tutorial, you will be familiar with the following:

- Creating additional data required to model storm and combined sewer systems.
- Performing steady state and dynamic (EPS) rainfall-runoff simulations.
- Reviewing and analyzing stormwater modeling results.

The estimated time to complete this section is approximately 30 minutes. If you have not completed the main quick start tutorial, then you should use the "**Sample**" project in the "C:\Users\Public\Documents\InfoSewer\Examples" directory (the path may be different for custom installations). Refer to tutorial page 3-3 for more information on opening an InfoSewer Pro project.

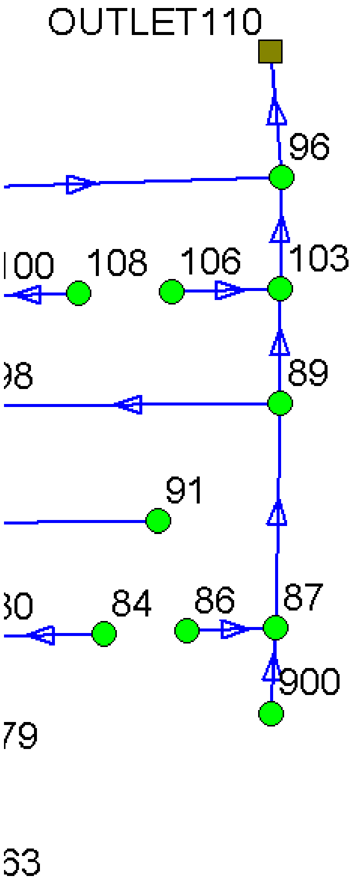
Step 12: Adding manhole data for Stormwater Modeling

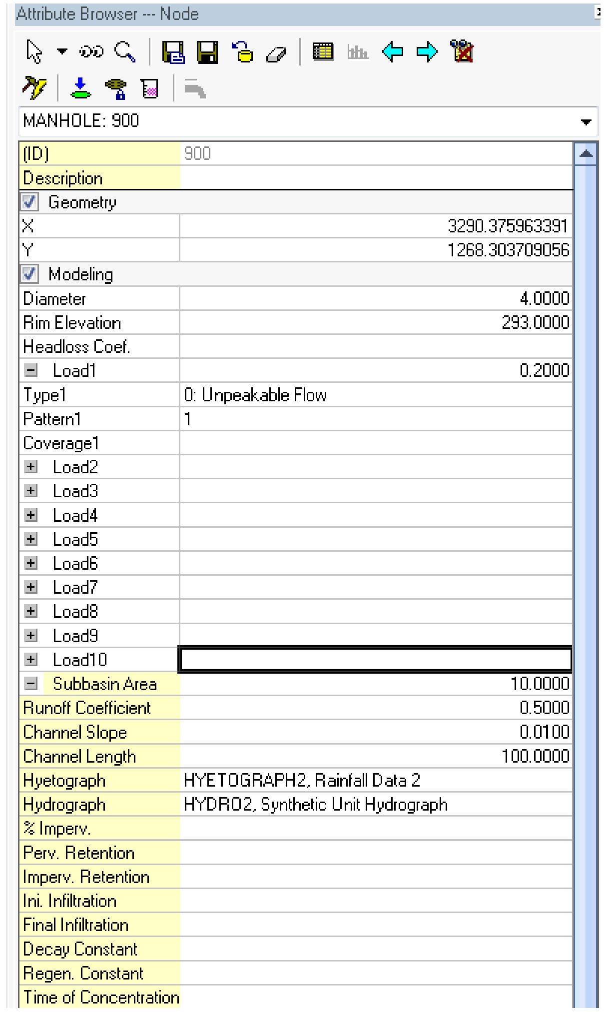
For each loading manhole, you will provide the area of the subwatershed that drains to the manhole, the average slope of the subwatershed, the length of the longest flow path in the subwatershed, and the runoff coefficient of the subwatershed. These data can be entered in the ATTRIBUTE BROWSER-NODE window one manhole at a time or globally using the DB EDITOR dialog box. Both methods are presented hereafter.

Perform the following three steps to add the data given in the table below to Manhole 900 using the *ATTRIBUTE BROWSER-NODE* window.

MANHOLE ID	SUBBASIN AREA (ACRES)	RUNOFF COEFFICIENT	CHANNEL SLOPE	CHANNEL LENGTH (ft)
900	10	0.6	0.01	200

- **1.** Click on the “Select” button , place the cursor on Manhole 900 and then press the left mouse button (use page 3-2 for graphical reference).
- **2.** In the *Modeling* section of the *ATTRIBUTE BROWSER-NODE* window on the right, perform the following:
  - Enter “10.00” in the *SUBBASIN AREA* field.
  - Enter “0.6 in the *RUNOFF COEFFICIENT* field.
  - Enter “0.01” in the *CHANNEL SLOPE* field.
  - Enter “200.00” in the *CHANNEL LENGTH* field.
- **3.** Click once on the “Save” button . The *ATTRIBUTE BROWSER-NODE* window should appear as shown below.

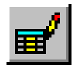




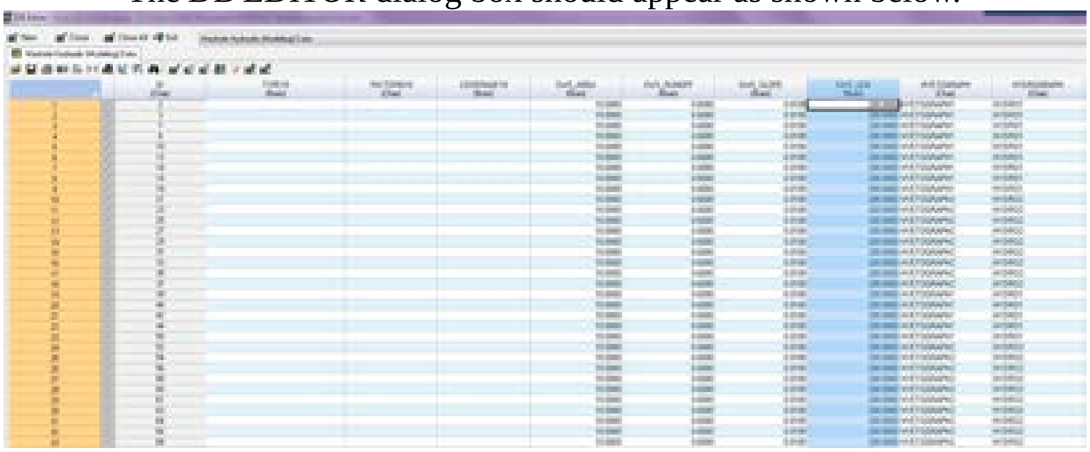
Using the *DB EDITOR* you will globally assign the Subbasin Area, the Runoff Coefficient, the Channel Slope, and the Channel Length data for all the loading manholes in the "**Tutorial**" project, including Manhole 900.

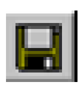
- **1.** Select the *DB TABLES* command from the *EDIT* menu or click once on the "DB Editor" button . The *DB EDITOR* appears on the screen.
- **2.** Under the *ELEMENT HYDRAULIC DATA* folder, select the *MANHOLE HYDRAULIC (MODELING) DATA* table. At the bottom of the dialog box, select *ENTIRE TABLE* in the *DISPLAY SCOPE* area, and then click the "OK" button. The *MANHOLE HYDRAULIC (MODELING) DATA* table appears on the screen.
- **3.** Go to the extreme right corner of the *MANHOLE HYDRAULIC (MODELING) DATA* table. To adjust the contents of an entire field, for example the Subbasin Area (*SWS_AREA*), click once on the column

header *SWS_AREA* with the mouse. The entire column should be highlighted.

- **4.** You will assign "10" for the Subbasin Area of all the manholes. Choose the "Block Editing" button  while the *SWS_AREA* field remains highlighted. When the *BLOCK EDITING* dialog box appears on the screen, choose the *Set Equal* editing operation, enter "10" for the value to be defined, then choose "OK". The subbassin area for the all the manholes should now be equal to 10.

- **5.** Repeat the above 2 steps to assign values of "0.6", "0.01", and "200" for the Runoff Coefficient (*SWS_RUNOFF*), the Channel Slope (*SWS_SLOPE*), and the Channel Length (*SWS_LEN*) fields respectively. The *DB EDITOR* dialog box should appear as shown below.



- **6.** Select the "Save" button  and then choose the "Close" button 

at the top of the *DB EDITOR* window. Close the *DB EDITOR* by selecting the "Exit" button 

at the top of the *DB EDITOR* window.

Adding Steady State Simulation Data

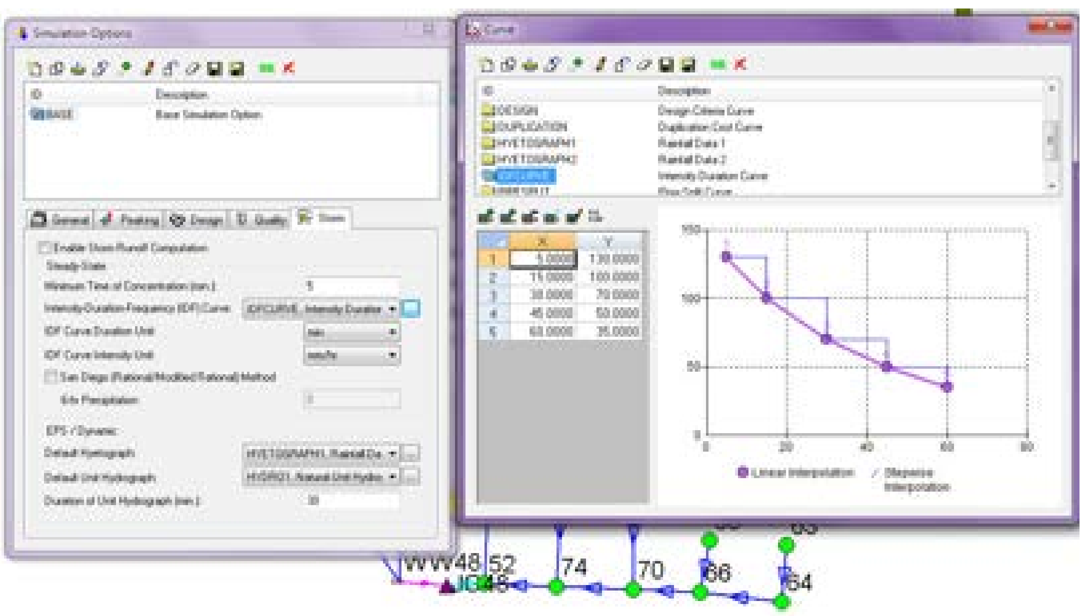
In addition to the manhole data you provided, steady state simulation of stormwater requires an Intensity-Duration-Frequency curve, commonly named as an IDF curve. In this section, you will create an IDF curve, assign an intensity unit and duration unit for the IDF curve, and assign the minimum time of concentration to be used by the Rational Method.

- 1. Select the *OPERATION DATA* tab from the *the Operations Tab in the Attribute Browser of InfoSewer*, select the SIMULATION OPTIONS ® BASE SIMULATION OPTIONS and double-click. When the SIMULATION OPTIONS dialog box appears choose the STORM tab.
- **2.** Enter "5" in the *MINIMUM TIME OF CONCENTRATION* field. For every loading manhole, the model compares this value with an internally calculated time of concentration, and uses the larger of the two.
- **3.** Next, you will create an IDF curve that will be used to derive the intensity of a design rainfall for every loading manhole. Next to the Intensity-Duration-Frequency (IDF) Curve drop down box, click on the *BROWSE* button .
- **4.** Click on the *NEW* icon in the *CURVE* dialog box. You are then prompted to enter the identifier and an optional description for the new curve. Enter "**IDFCURVE, Intensity Duration Curve**" in the *NEW ID* field.
- **5.** To define the curve data values for the IDF Curve, perform the following:
- Choose the "Set Rows" button , enter "5" as the value, and choose "OK".
- Enter the data values as defined below and press the *TAB* key to apply the change. As you add the values, the graph will be automatically updated on the dialog box.
- Click on the "Save" button at the top of the dialog box to apply the values to the IDF curve.

X Duration (minute)	Y Intensity (mm/hr)

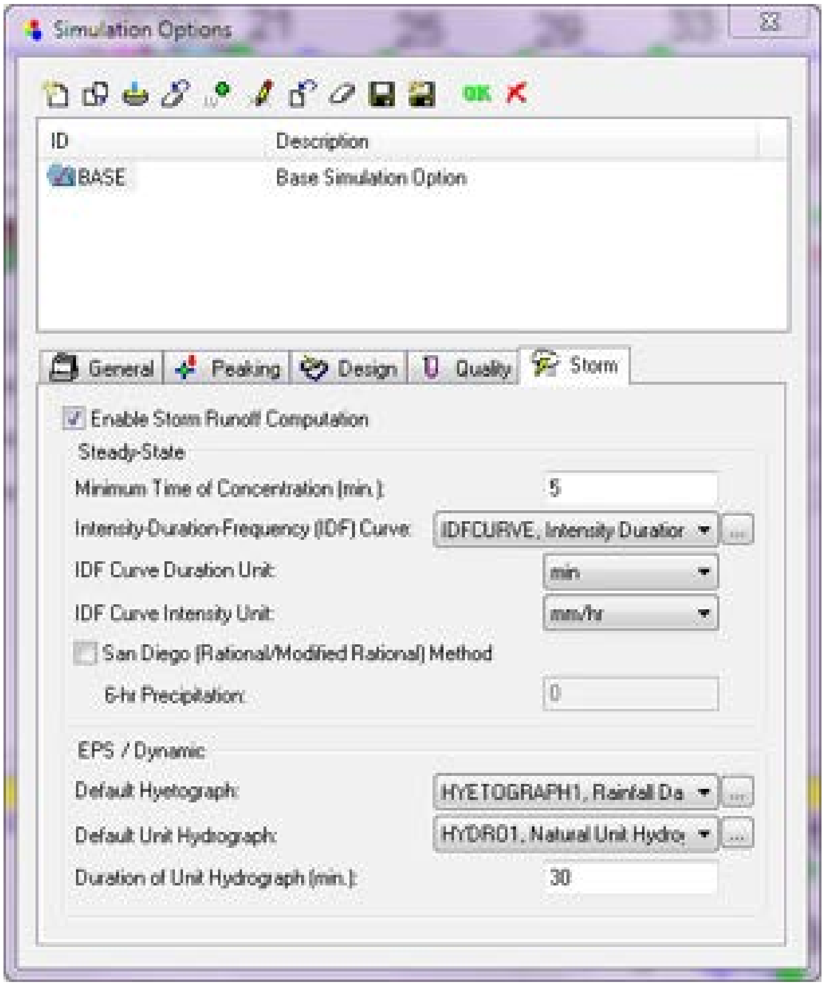
5.0	130.0
15.0	100.0
30.0	70.0
45.0	50.0
60.0	35.0

- **6.** Choose the "OK" button  to close the *CURVE* dialog box. The following picture illustrates how the IDF Curve should appear on the dialog box.



- **7.** Select *min* for the *IDF CURVE DURATION UNIT* field, and select *mm/hr* for *IDF CURVE INTENSITY UNIT* field. Click on the "Save" button  at the top of the dialog box to apply the changes. Finally, check the *ENABLE STORM RUNOFF COMPUTATION* option.

- **8.**. Choose the "OK" button  to close the *SIMULATION OPTIONS* dialog box. The following picture illustrates how the *SIMULATION OPTIONS* dialog box should appear on the screen.



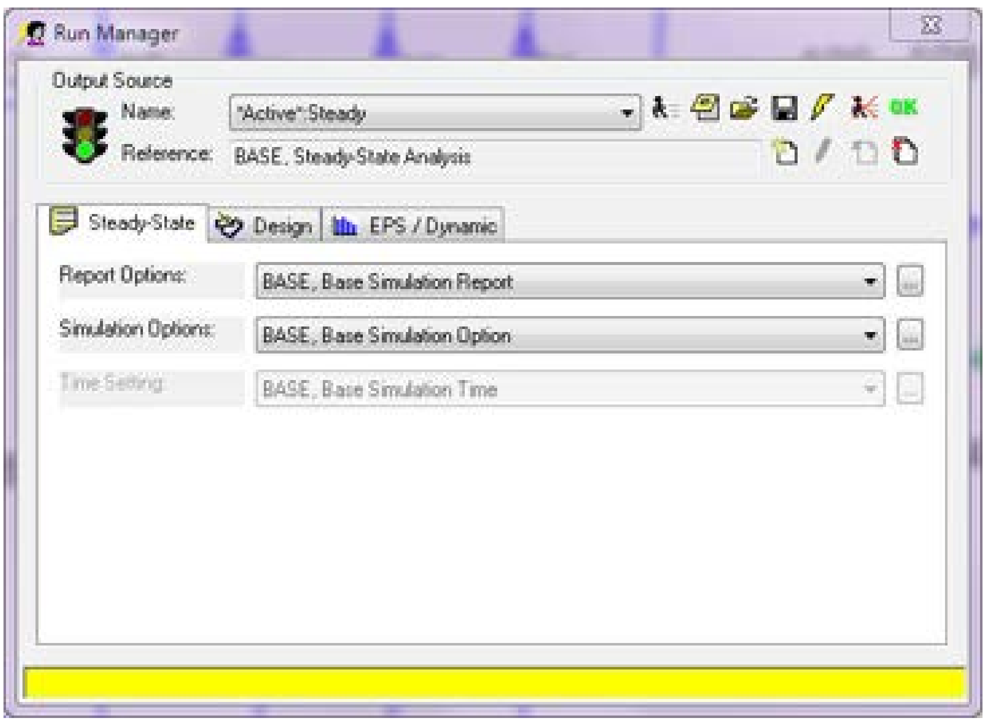
At this point of the tutorial, you have entered all the additional data required for steady state simulation of stormwater.

Run Steady State Simulation

To run a steady state simulation, perform the following:

- **1.** Choose the *RUN MANAGER* command from the *TOOLS* menu. The *RUN MANAGER* dialog box appears on the screen. Choose the *STEADY-STATE* tab.

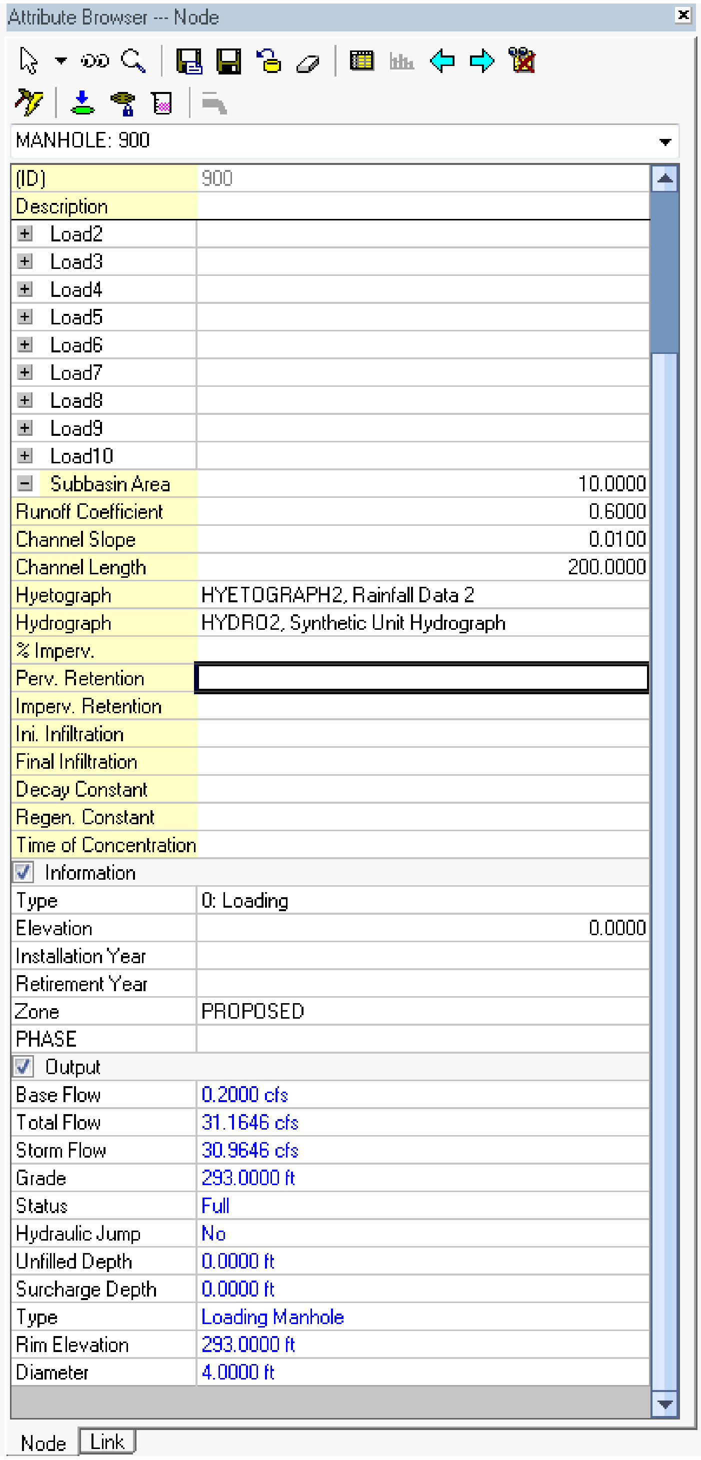
- **2.** Click on the "Run" button. Upon successful completion of the simulation, the Output Source Status Indicator (the stoplight) should show green.



Review the steady state analysis results using the *ATTRIBUTE BROWSER* window one network component at a time. To do this, perform the following:

- Click on the "Select" button and then click on the desired node or link. Results for that node or link are shown in the *Output* section of the *ATTRIBUTE BROWSER* window.

The model reports storm flows for the pipes and the loading manholes. Run Steady State Simulation



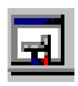
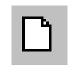
Attribute Browser --- Node
MANHOLE: 900
(ID) 900
Description
Load2
Load3
Load4
Load5
Load6
Load7
Load8
Load9
Load10
Subbasin Area 10.0000
Runoff Coefficient 0.6000
Channel Slope 0.0100
Channel Length 200.0000
Hyetograph HYETOGRAPH2, Rainfall Data 2
Hydrograph HYDRO2, Synthetic Unit Hydrograph
% Imperv.
Perv. Retention
Imperv. Retention
Ini. Infiltration
Final Infiltration
Decay Constant
Regen. Constant
Time of Concentration
Information
Type 0: Loading
Elevation 0.0000
Installation Year
Retirement Year
Zone PROPOSED
PHASE
Output
Base Flow 0.2000 cfs
Total Flow 31.1646 cfs
Storm Flow 30.9646 cfs
Grade 293.0000 ft
Status Full
Hydraulic Jump No
Unfilled Depth 0.0000 ft
Surcharge Depth 0.0000 ft
Type Loading Manhole
Rim Elevation 293.0000 ft
Diameter 4.0000 ft
Node
Link

Step 13: Creating a Unit Hydrograph

To derive a storm Hydrograph, InfoSewer Pro. uses the Unit Hydrograph Theory. If a natural unit hydrograph (a unit hydrograph created from observed field data) is available for the watershed being investigated, the user may provide it in the form of a pattern. If a natural unit hydrograph does not exist, InfoSewer Pro. can generate a synthetic unit hydrograph using four different techniques (i.e., SCS dimensionless unit hydrograph, SCS triangular unit hydrograph, the tri-triangle method, or the Colorado Urban Hydrograph Procedure). Theoretical detail of the methods is available in Section 4.5.

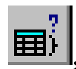
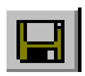
In this section, you will create one natural unit hydrograph and one synthetic unit hydrograph using the tri-triangle method. Later in the tutorial, you will assign the natural unit hydrograph to some of the loading manholes in the "**Tutorial**" project. The synthetic unit hydrograph will be assigned to the remaining loading manholes.

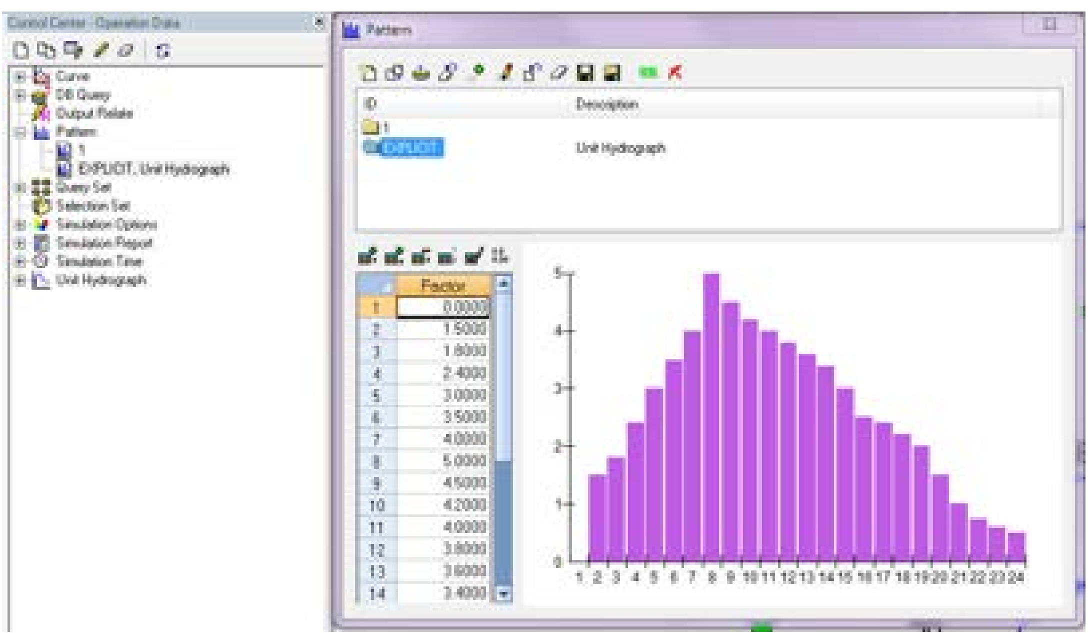
Create a natural unit hydrograph using the following procedure:

- Select the *OPERATION DATA* tab  from the *CONTROL CENTER in H2OMap Sewer or the Operations Tab in the Attribute Browser of InfoSewer*, select *PATTERN* and click on the "New" button  in the upper left-hand corner of the window.
- You are then prompted to enter a new ID and description for the pattern. Enter "**EXPLICIT, Natural Unit Hydrograph**" in the *PATTERN ID* field and choose the "OK" button.
- The *PATTERN* dialog box now appears on the screen. You will enter the following 24 unit hydrograph ordinates. The pattern time step is assumed to be one hour. Please note that unlike the sanitary load patterns, the ordinates of unit hydrograph pattern represent actual storm flow values i.e., they are not multiplication factors.
- To define the values of the twenty-four ordinates for the unit hydrograph pattern, perform the following:

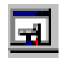
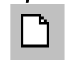
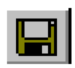
Ordinate 1	**Ordinate 2**	**Ordinate 3**	**Ordinate 4**	**Ordinate 5**	**Ordinate 6**
0.0	**1.5**	**1.8**	**2.4**	**3.0**	**3.5**
Ordinate 7	**Ordinate 8**	**Ordinate 9**	**Ordinate 10**	**Ordinate 11**	**Ordinate 12**
4.0	**5.0**	**4.5**	**4.2**	**4.0**	**3.8**

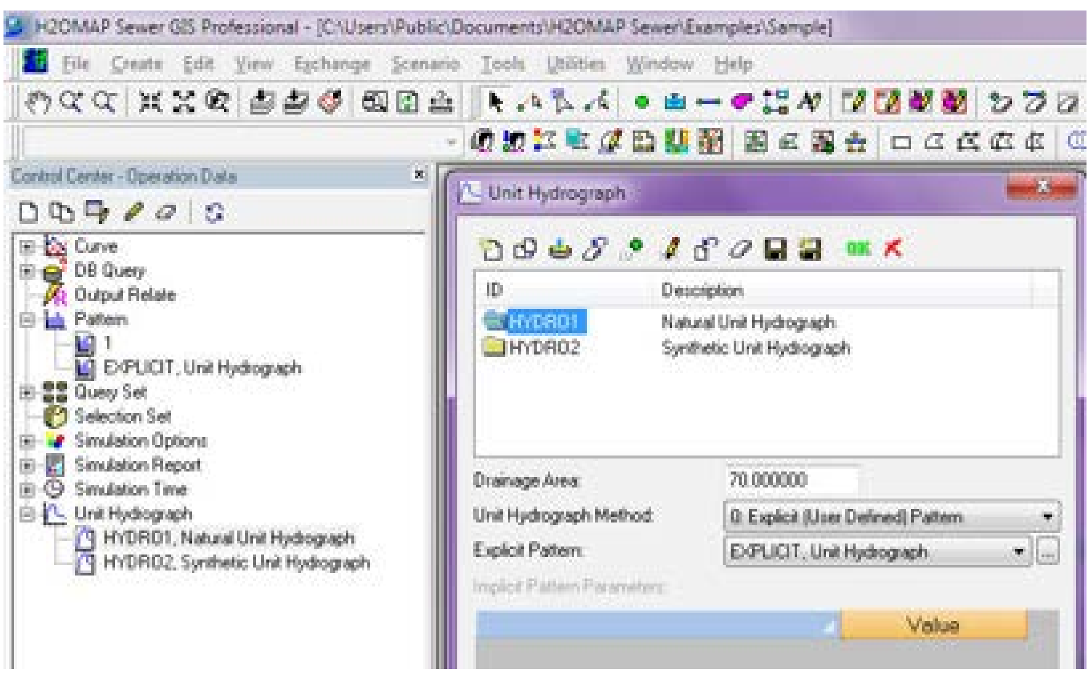
Ordinate 13	**Ordinate 14**	**Ordinate 15**	**Ordinate 16**	**Ordinate 17**	**Ordinate 18**
3.6	**3.4**	**3.0**	**2.5**	**2.4**	**2.2**
Ordinate 19	**Ordinate 20**	**Ordinate 21**	**Ordinate 22**	**Ordinate 23**	**Ordinate 24**
2.0	**1.5**	**1.0**	**0.75**	**0.6**	**0.5**

- Choose the “Set Rows” button , enter “24” as the value, and choose “OK”.
- Enter values of the ordinates as defined above and press the *ENTER* key to apply the change. As you add the values the graph will be automatically updated on the dialog box.
- Click on the “Save” button  at the top of the dialog box to apply the changes to the pattern.
- Choose the “OK” button  to close the *PATTERN* dialog box. Now you have created the pattern. The following picture illustrates how the pattern should appear on the screen.



Next, you will define this pattern as a natural unit hydrograph in H_2OMAP Sewer/Pro.

- **1.** Select the *OPERATION DATA* tab  from the *CONTROL CENTER or the Operations Tab in the Attribute Browser of InfoSewer.* select *UNIT HYDROGRAPH* and click on the "New" button  in the upper left-hand corner of the window.
- **2.** You are then prompted to enter a new ID and description for the hydrograph. Enter "**HYDRO1, Natural Unit Hydrograph**" in the *NEW ID* field and choose the "OK" button. The *UNIT HYDROGRAPH* dialog box should appear on the screen.
- **3.** Enter "70" in the *DRAINAGE AREA* field. This refers to the area of the subwatersheds that drain to the location where the unit hydrograph is derived. The unit is in acres if US Customary Units are used, and is in square meters for SI Units.
- **4.** Select the *EXPLICIT* option for the *UNIT HYDROGRAPH PATTERN TYPE*. Unit hydrographs provided by the user are *EXPLICIT* unit hydrographs. Implicit unit hydrographs refer to those internally generated by the model. Click on the *EXPLICIT PATTERN* drop down box, and select "**EXPLICIT, Natural Unit Hydrograph**" pattern.
- **5.** Click on the "Save" button  at the top of the dialog box to apply the changes. The following picture illustrates how the *UNIT HYDROGRAPH* dialog box should appear on the screen.



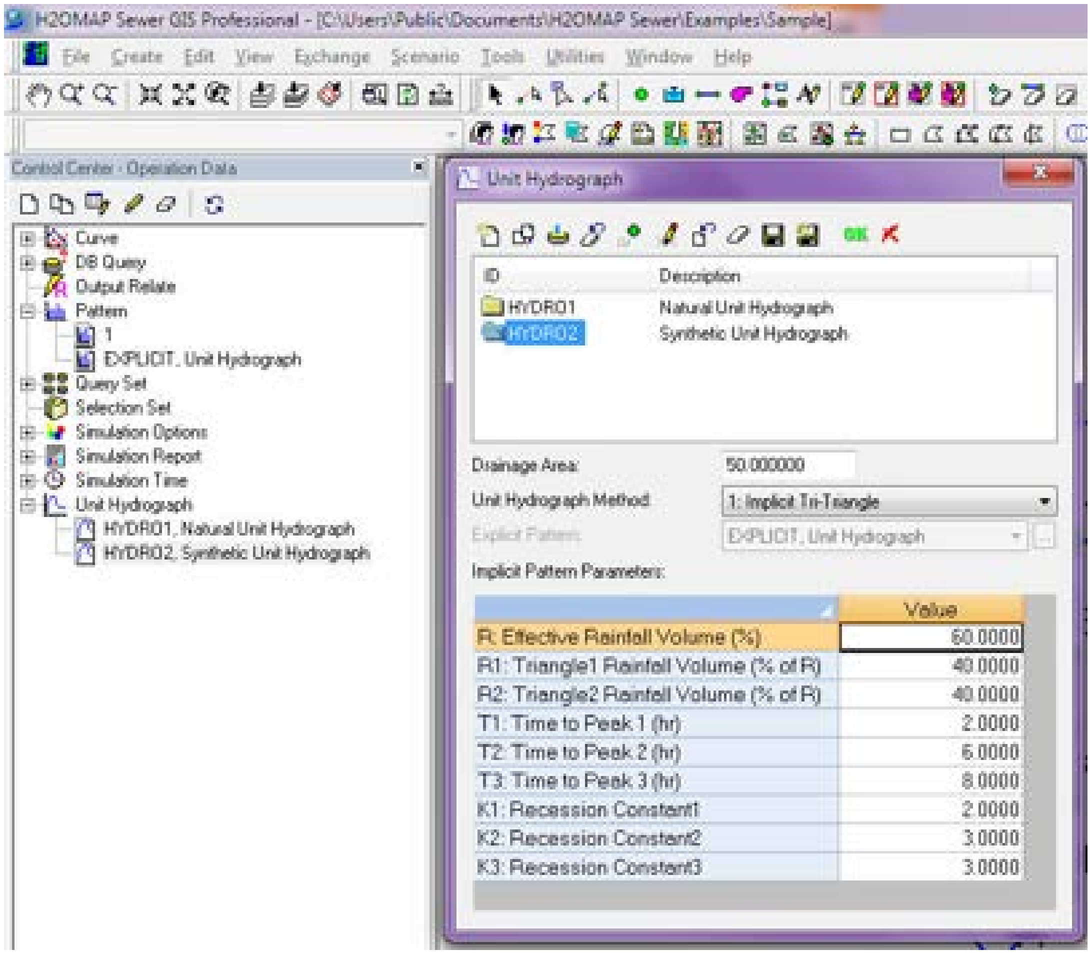
- **6.** Choose the “OK” button  to close the *UNIT HYDROGRAPH* dialog box.

Next, you will assign the data required to create an *Implicit* unit hydrograph using the tri-triangle method as follows:

- Select the *OPERATION DATA* tab from the *CONTROL CENTER*, select *UNIT HYDROGRAPH* and click on the “New” button in the upper left-hand corner of the window. Enter “**HYDRO2, Synthetic Unit Hydrograph**” in the *NEW ID* field and choose the “OK” button. The *UNIT HYDROGRAPH* dialog box should appear on the screen.
- Enter “50” in the *DRAINAGE AREA* field. Select the *IMPLICIT* option in the *UNIT HYDROGRAPH PATTERN TYPE*, and enter the values given below for the nine parameters of the tri-triangle method.

Tri-Triangle Parameters	Value
R: Effective Rainfall Volume (%)	60.000
R1: Triangle1 Rainfall Volume (%)	40.000
R2: Triangle2 Rainfall Volume (%)	40.000
T1: Time to Peak 1 (hr)	2.000
T2: Time to Peak 2 (hr)	6.0000
T3: Time to Peak 3 (hr)	8.0000
K1: Recession Constant1	2.0000
K2: Recession Constant2	3.0000
K3: Recession Constant3	3.0000

- Click on the “Save” button at the top of the dialog box to apply the changes. The following picture illustrates how the *UNIT HYDROGRAPH* dialog box should appear on the screen.



- Choose the "OK" button  to close the *UNIT HYDROGRAPH* dialog box. We have created one explicit unit hydrograph and one implicit unit hydrograph. Next, we will create analysis rainfall data.

Creating Rainfall

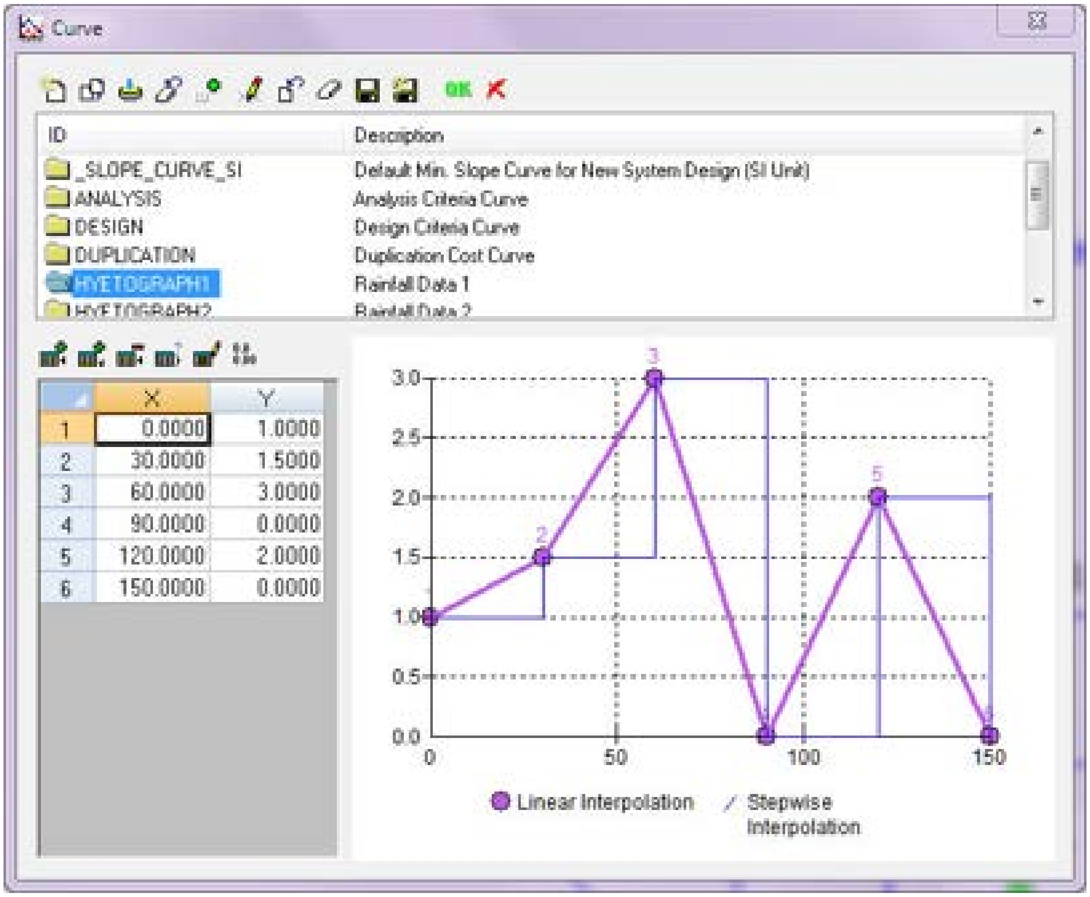
The objective of any stormwater modeling is to estimate the runoff resulting from known rainfall pattern. Therefore, rainfall is the driving force of stormwater modeling. Rainfall is commonly measured at rain gauge stations. A watershed may have more than one rain gauge station to capture spatial variability of a rainfall across the watershed. InfoSewer Pro has the capability to accept more than one rainfall pattern to model a sewer collection system. In this section of the Extended Tutorial, you will create two rainfall patterns (also known as hyetographs) which will be assigned to the loading manholes in the "**Tutorial**" project.

InfoSewer Pro accepts rainfall data in the form of a curve.

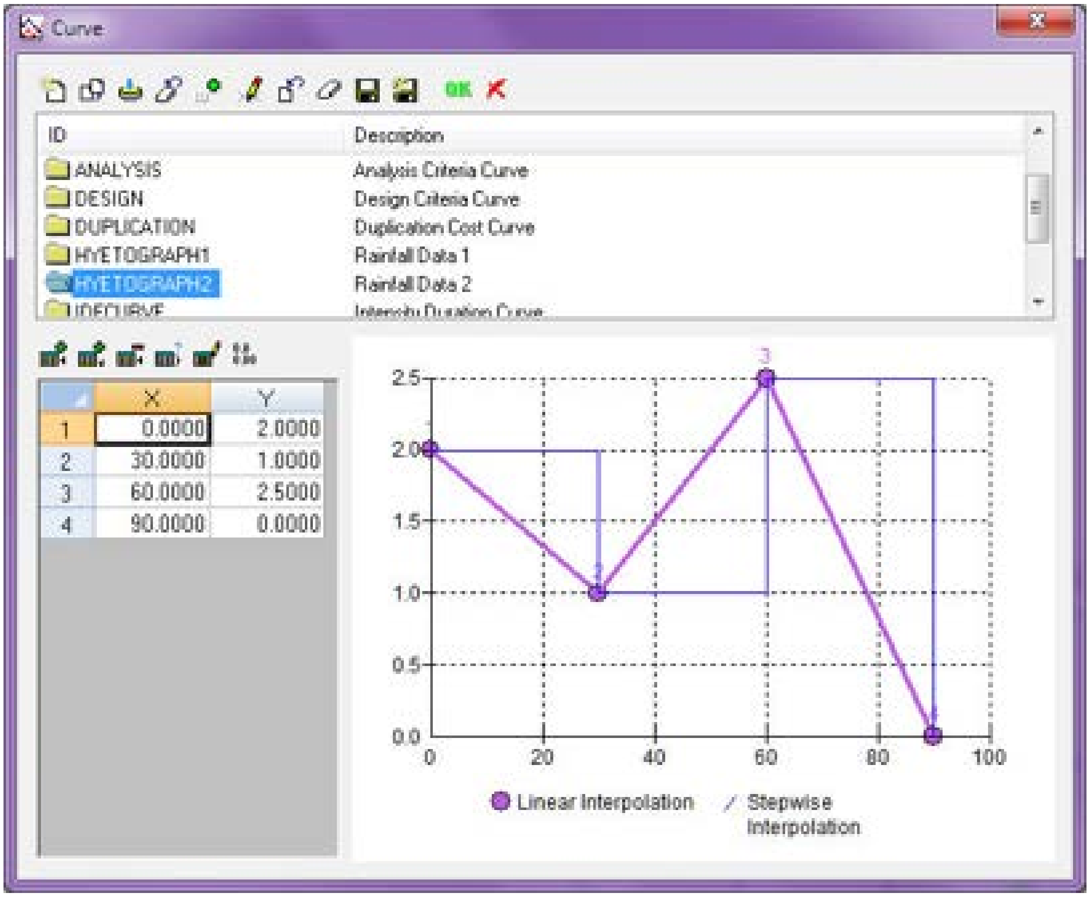
- Create a new curve and enter the following values. Use **"HYETOGRAPH1" and "Rainfall Data 1**" as a curve ID and description, respectively.

X TIME (minutes)	Y INTENSITY (inches/hr)
0.0	1.0
30.0	1.5
60.0	3.0
90.0	0.0
120.0	2.0
150.0	0.0

Refer to page 3-62 for instructions on creating curves. The following picture illustrates how the *CURVE* dialog box should appear on the screen. Note that InfoSewer Pro uses stepwise interpolation for rainfall data.



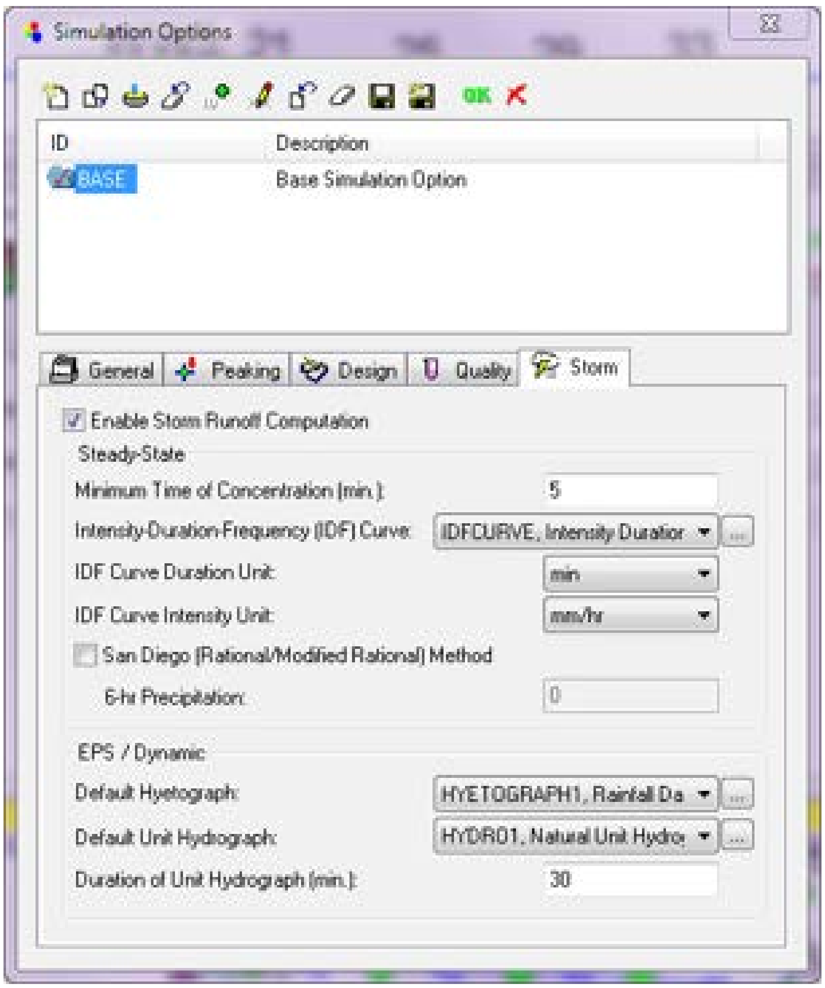
Following the same procedure as above, create a new hyetograph and enter the data given in the table below. Name the new hyetograph "**HYETOGRAPH2**", and use "**Rainfall Data 2**" for the description.

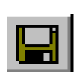


X TIME (minutes)	Y INTENSITY (inches/hr)
0.0	2.0
30.0	1. 0
60.0	2.5
90.0	0.0

Adding Dynamic Simulation Data

- **1.** Select the *OPERATION DATA* tab from the *CONTROL CENTER*, select the *SIMULATION OPTIONS* ® *BASE SIMULATION OPTIONS* and double-click. When the *SIMULATION OPTIONS* dialog box appears choose the *STORM* tab.
- **2.** In the *EPS / DYNAMIC* area select **HYETOGRAPH1** for the *DEFAULT HYETOGRAPH*, and choose **HYDRO1** for the *DEFAULT UNIT HYDROGRAPH*. Use "30" for the *DURATION OF UNIT HYDROGRAPH* field.
- **3.** Check the *ENABLE STORM RUNOFF COMPUTATION* box. The following picture illustrates how the *SIMULATION OPTIONS* dialog box should appear on the screen.



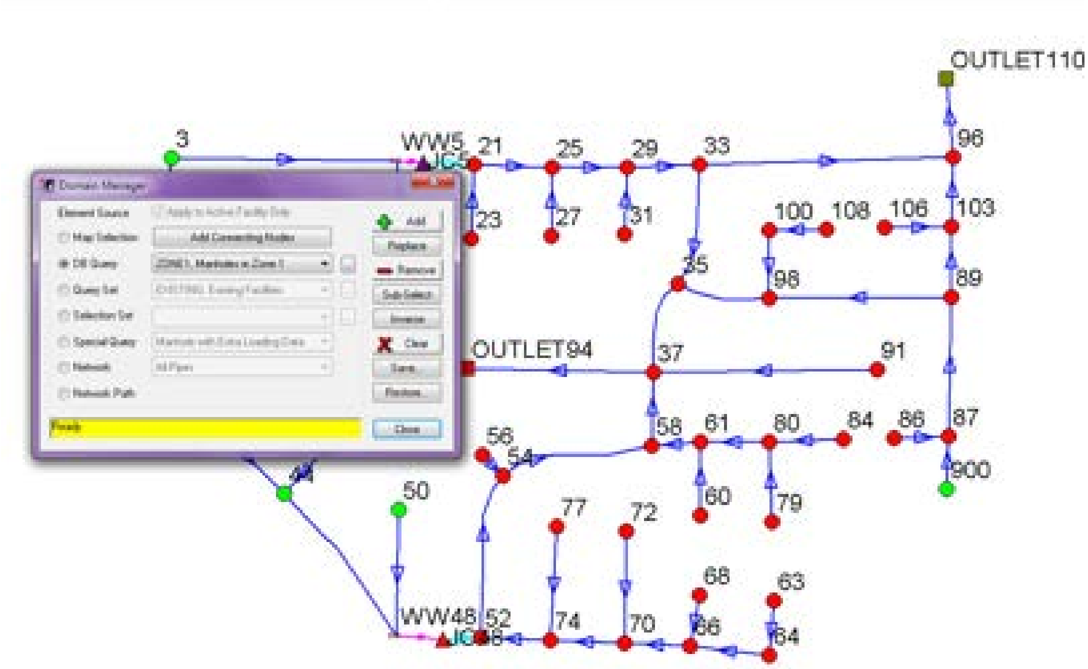
- **4.** Click on the "Save" button  at the top of the dialog box to apply the changes. Choose the "OK" button  to close the *SIMULATION OPTIONS* dialog box.

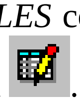
At this point of the tutorial, you have entered all the additional data required for dynamic simulation of stormwater.

Assigning Unit Hydrograph and Hyetograph to Loading Manholes

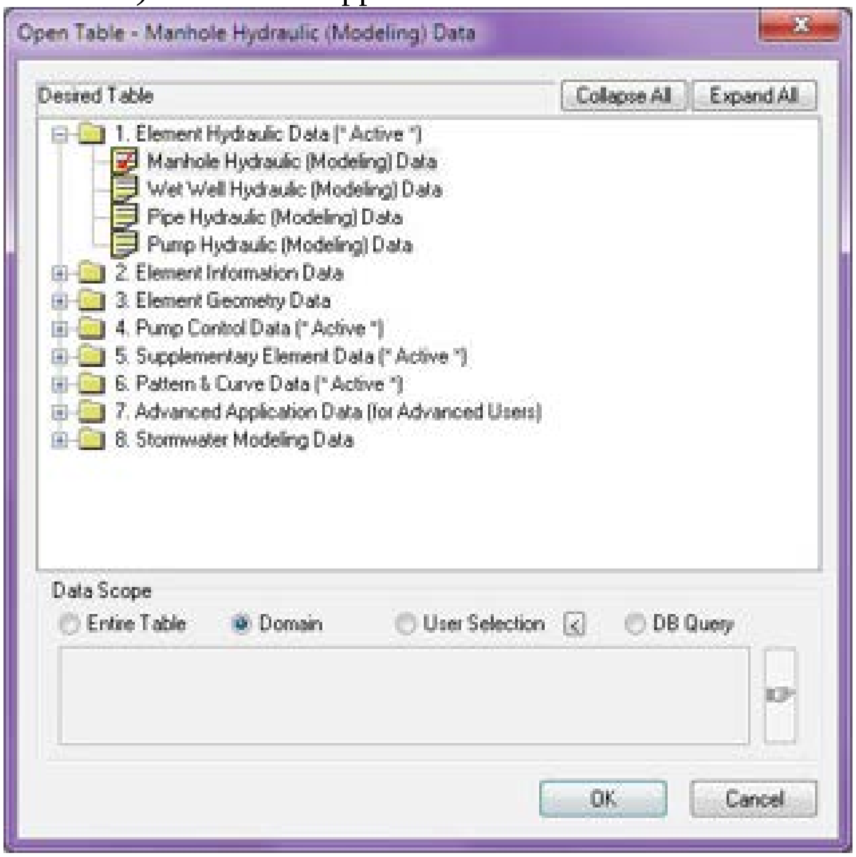
You will assign the unit hydrographs and the hyetographs created above to the loading manholes in the "**Tutorial**" project. We have created two unit hydrographs (**HYDRO1** and**HYDRO2***)* and two hyetographs (**HYETOGRAPH1** and**HYETOGRAPH2**). You will create two domains and assign *HYDRO1* and *HYETOGRAPH1* data to all loading manholes in the first domain (Manholes in Zone 1). The remaining loading manholes will be assigned *HYDRO2* and *HYETOGRAPH2* (Manholes in Zone 2).

- **1.** From the *TOOLS* menu, select *DOMAIN* then *DOMAIN MANAGER* command. The *DOMAIN MANAGER* appears on the screen. You will now select an existing query that selects the manholes in Zone 1. On the *DOMAIN MANAGER*, choose the *DB QUERY* option and choose the **ZONE1** query statement.
- **2.** Choose the *ADD* button on the *DOMAIN MANAGER* to add the manholes in Zone 1 to the current network domain. Click on the *CLOSE* button. The manholes in Zone 1 should now be highlighted in red.

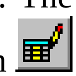


- **3.** Select the *DB TABLES* command from the *EDIT* menu or click once on the "DB Editor" button . The *DB EDITOR* appears on the screen.

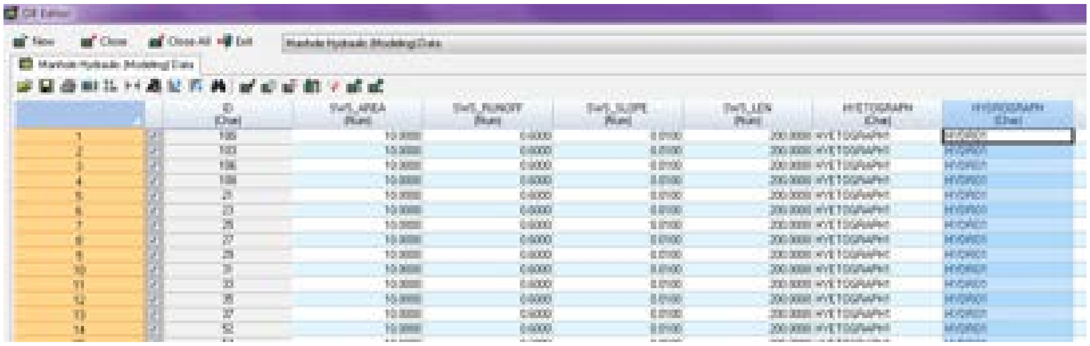
- **4.** Under the *ELEMENT HYDRAULIC DATA* folder, select the *MANHOLE HYDRAULIC (MODELING) DATA* table. At the bottom of the dialog box, select ***DOMAIN*** in the *DISPLAY SCOPE* area (indicating you only want the content of the manholes in the domain to be displayed and available for editing), and then click the “OK” button. The *MANHOLE HYDRAULIC (MODELING) DATA* table appears on the screen.



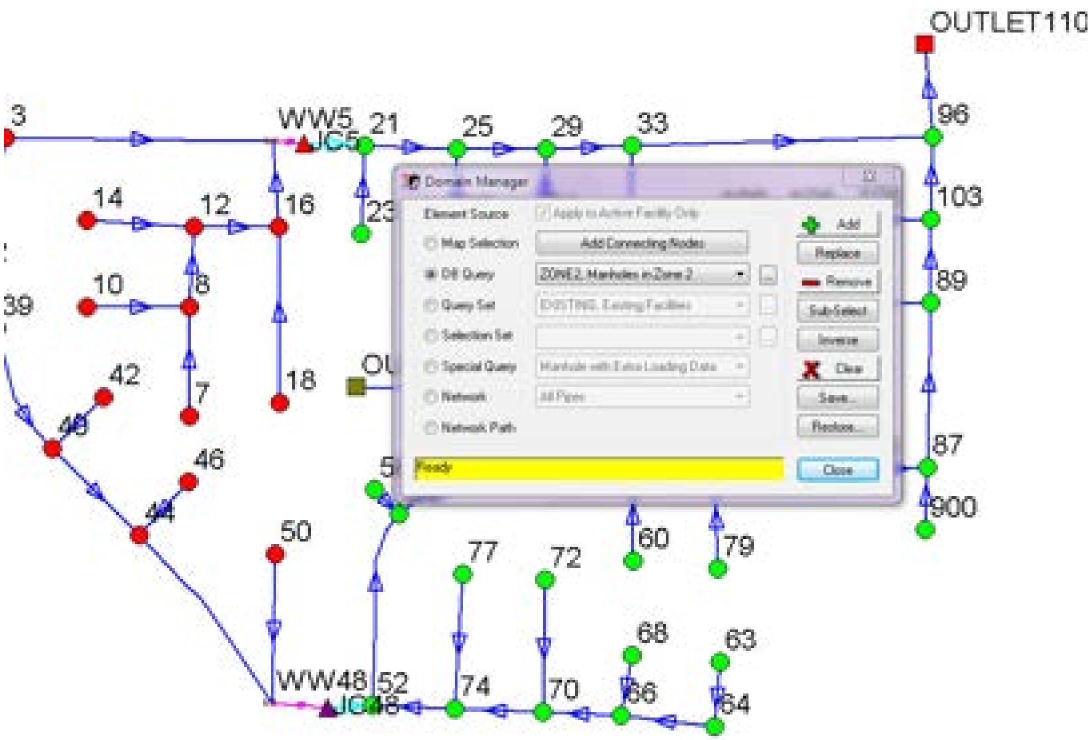
- **5.** Go to the extreme right corner of the *MANHOLE HYDRAULIC (MODELING) DATA* table.

- **6.** Click once on the header of column *HYETOGRAPH* with the mouse. The entire column should be highlighted. Choose the “Block Editing” button  while the *HYETOGRAPH* field remains highlighted. When the *BLOCK EDITING* dialog box appears on the screen, enter **HYETOGRAPH1** in the *REPLACE WITH* field, then choose “OK”.

- **7.** Click once on the header of column *HYEDROGRAPH* with the mouse. Choose the “Block Editing” button  while the *HYEDROGRAPH* field remains highlighted. When the *BLOCK EDITING* dialog box appears on the screen, enter **HYDRO1** in the *REPLACE WITH* field, then choose “OK”.



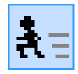
- **8.** Repeat the above steps to assign the **HYETOGRAPH2** and **HYDRO2** to the manholes in Zone 2. When you open the *DOMAIN MANAGER* dialog box, choose the *RESET* button before adding the **ZONE2** query statement.

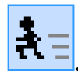


At this stage of the tutorial you have created the data required to run steady state and dynamic simulation of Stormwater using InfoSewer Pro.

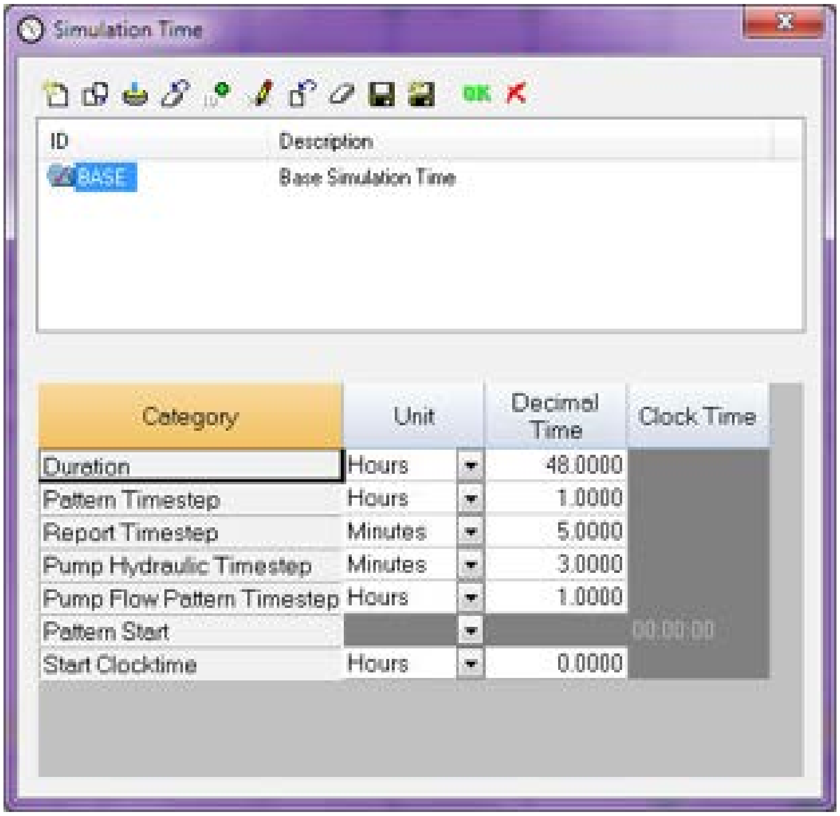
Running The Dynamic Simulation

To run the a dynamic simulation, perform the following:

- **1.** Choose the *RUN MANAGER* command  from the *TOOLS* menu. The *RUN MANAGER* dialog box appears on the screen. Choose the *EPS / DYNAMIC* tab and run it for 48 hours as shown in the Simulation time dialog.

- **2.** Click on the “Run” button . Upon successful completion of the simulation, the Output Source Status Indicator (the stoplight) should show green. Close the *RUN MANAGER* dialog box.

Dynamic results may be reviewed from the *ATTRIBUTE BROWSER* window, or using the *OUTPUT REPORT MANAGER* dialog box. The model reports *STORM FLOW* and *TOTAL STORM FLOW* for every loading manhole. *STORM FLOW* refers to storm load entering the system at the manhole while *TOTAL STORM FLOW* is the storm load contributed by all the subwatersheds draining to the manhole. Refer to the Output Report Manager section on page 3-32 for more information on displaying model results in tabular and graphical format.



[Mass Balance Calculation]

Total external inflow: 1504366.304 (cubic feet)

Total storm inflow (part of external inflow): 1029079.286 (cubic feet)

Total extern outflow: 810031.550 (cubic feet)

Total flooding out of open conduits: 0.000 (cubic feet)

Initial Wet Well Storage: 0.000 (cubic feet)

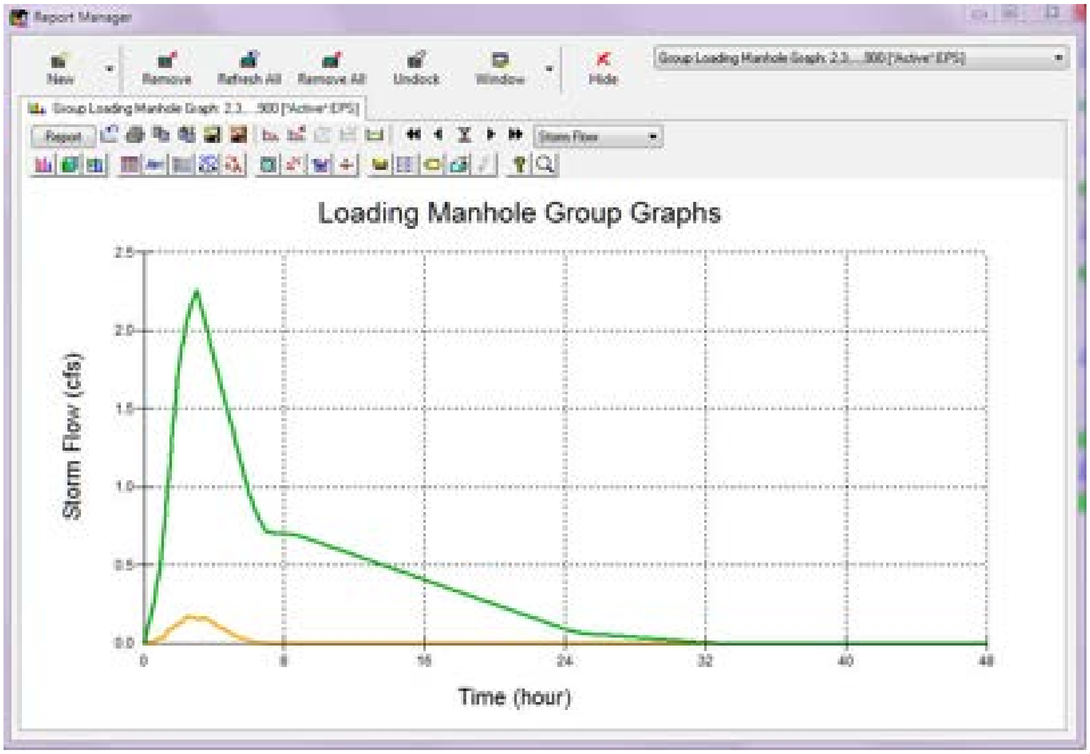
Final Wet Well Storage: 1220.915 (cubic feet)

Wet Well Flooding Volume: 691734.882 (cubic feet)

Init Gravity Main Storage: 0.000 (cubic feet)

Final Gravity Main Storage: 1518.164 (cubic feet)

Mass Balance Check: 0.009 (%)

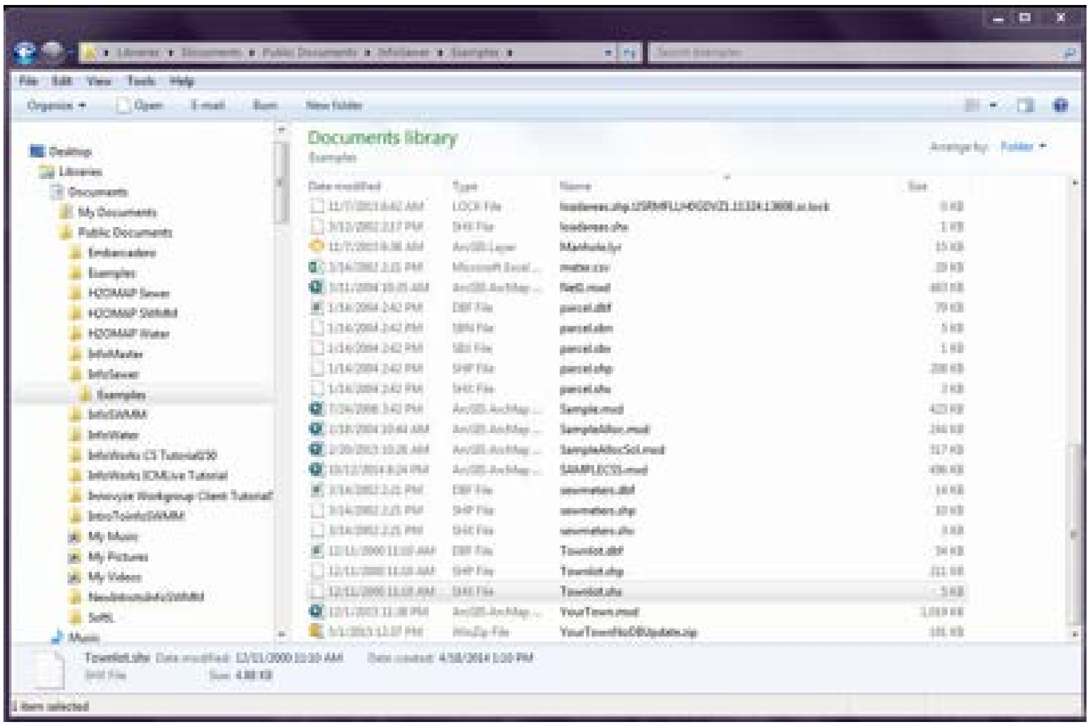


Report Manager
New
Remove
Refresh All
Remove All
Undock
Window
Hide
Report
Storm Flow
Loading Manhole Group Graphs
Storm Flow (cfs)
Time (hour)

Conclusion

Congratulations! You have now completed the InfoSewer Pro Tutorial. You should now have the skills necessary to develop and analyze sanitary and storm sewer collection system network models with the extensive and powerful suite of tools provided by InfoSewer Pro.

Your "**Tutorial**" project should now closely match the "**Sample**" project in the "**C:\Users\Public\Documents\InfoSewer\Examples**" directory (the path may be different for custom installations).



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

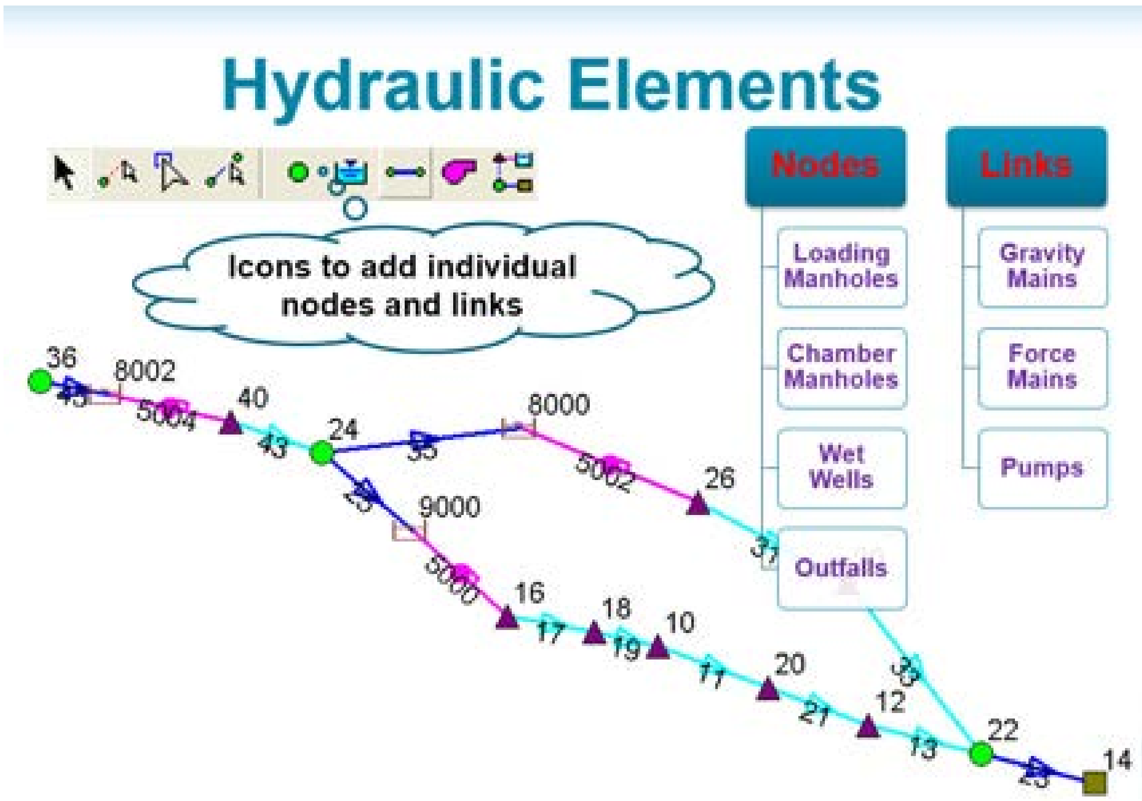
List of Physical Components

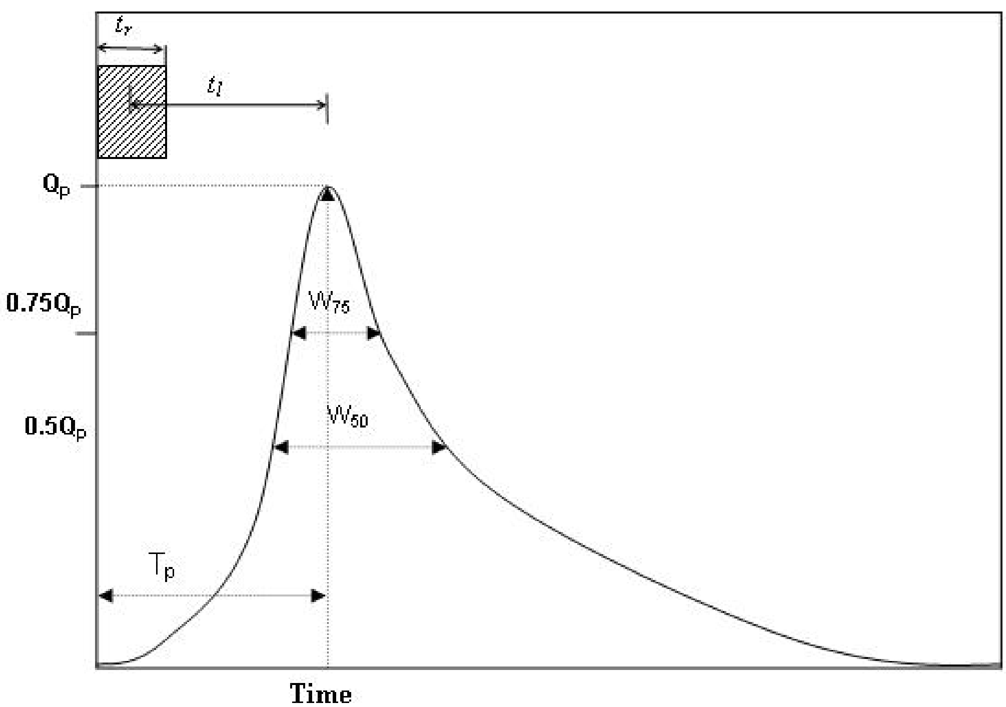
InfoSewer conceptualizes a sewer collection network as a collection of links connected together at their endpoints, called nodes. The nodes represent manholes, wet-wells, and outlets while the links represent pipes and pumps. The sewer system can be of any topological configuration and can contain multiple loops and outlets.

The following sewer network components are modeled in InfoSewer:

- **Manholes** represent points in the sanitary sewer system where loads enter into the system. Manholes are normally located at places where pipes connect and where pipe characteristics such as diameter and slope change.
- **Outlets** designate discharge points (e.g., treatment plant, ocean outfall) where flows exit the system and are the most downstream points of the collection system.
- **Wet wells** are structures in the collection system that collect wastewater flows before they are pumped into force mains for transport to another gravity system.
- **Pipes/Channels** are circular/non-circular conduits through which flow is transported either by gravity (i.e., gravity mains) or by the energy supplied from pumps (i.e., force mains).
- **Pumps** are devices that raise the hydraulic head of water through the sanitary sewer system.

How InfoSewer models the hydraulic behavior of each of these physical components is reviewed in the following sections. For the sake of discussion, all wastewater flows are expressed in cubic feet per second (cfs), although the program can also accept flow units in gallons per minute (gpm), million gallons per day (mgd), imperial million gallons per day (imgd), acre-feet per day (afd), cubic meters per hour (cmh), cubic meters per day (cmd), million liters per day (mld), liters per second (lps), or liters per minute (lpm). Wastewater represents the spent or used water from residences, commercial buildings, industrial plants, and institutions, together with minor quantities of ground, storm and surface waters that are not admitted intentionally.


Hydraulic Elements
Nodes
Links
Icons to add individual nodes and links
Loading Manholes
Chamber Manholes
Wet Wells
Outfalls
Gravity Mains
Force Mains
Pumps
36
8002
5004
40
24
43
35
8000
5002
26
25
9000
5000
16
17
18
19
10
11
20
21
12
13
33
22
14


t_r
t_l
Q_p
$0.75Q_p$
$0.5Q_p$
W_{75}
W_{50}
T_p
Time

Manholes in InfoSewer

Manholes are points in the network where links join together and where loads enter the network. They are also placed at locations where pipe characteristics change (e.g., diameter and slope). The basic input data required for manholes is:

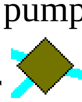
- The rim elevation , i.e. the top elevation of the manhole structure
- The diameter of the manhole structure
- The baseline load at the manhole

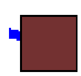
The manhole diameter and rim elevation parameters are used in the determination of manhole surcharge and flooding. The loads on the system, i.e., wastewater flows collected, accumulated and conveyed by the collection system are assumed to be positive. Manholes can have their load vary with time and different types of loading (e.g., low density residential, medium density residential, high density residential, industrial, commercial, etc.) can be assigned to them. A manhole can also have no loading defined.

The results computed for manholes for a simulation are:

- The base load
- The total load
- The overload storage
- The hydraulic grade

Two other types of manhole can also be modeled:

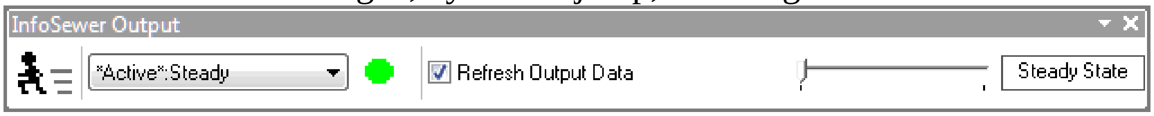
Junction chambers represent nodes in a pressurized sewer collection system connecting pumps to force mains. No loads can be defined for a junction chamber .

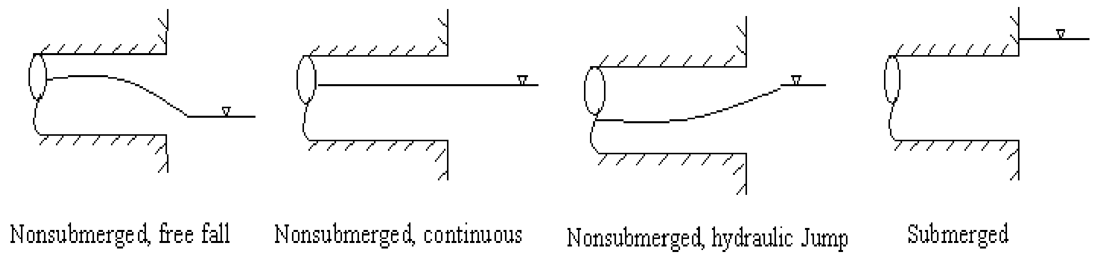
Outlets designated Facilities where flows exit the collection system. These nodes define the discharge end or the most downstream element of a sewer network. The exit condition can be grouped into four cases as shown below: 

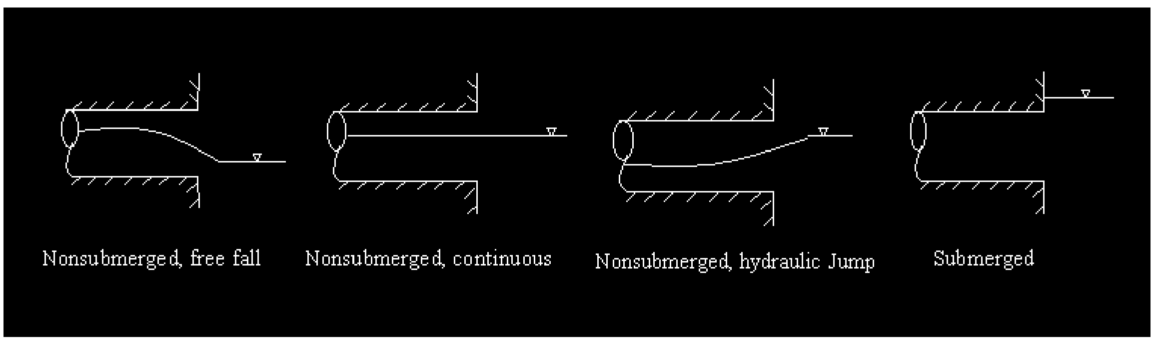
Nonsubmerged, free fall

Nonsubmerged, continuous

Nonsubmerged, hydraulic jump, Submerged







Flow splits (bifurcations) can also be modeled and represent points in the conveyance system where multiple pipes separate flows to different parts of the system. A flow split occurs whenever two or more pipes exit the same manhole. InfoSewer Pro provides four practical methods of allocating flows to the downstream (outgoing) pipes :

- **Fixed flow split percentage method -** The user specifies the percentage of the total flow in the manhole which is assigned to each of the downstream pipes.
- **Variable flow split percentage method -** The user defines a curve representing the percentage of the total flow in the manhole which is assigned to a downstream pipe as a function of the total incoming flow (See CURVE section).

- **Inflow-outflow flow split method -** The user supplies an inflow-outflow curve for each of the downstream pipes splitting from a manhole. The curve defines the amount of the incoming flow to be diverted to a downstream pipe. This flow split method is designed to accommodate flow conditions where hydraulic structures such as weirs and dams are used to regulate downstream flows.
- **Automatic flow split method** -InfoSewer automatically computes the fraction of the total flow in the manhole assigned to each of the downstream (outgoing) pipes based on their invert levels and diameters.

Note that InfoSewer checks that the sum of the fractions leaving a manhole equals 100 percent. In case, the sum is less or greater than 100%, InfoSewer will automatically adjust the flow allocations for the outgoing pipes proportionally to satisfy flow balance. Finally, note that InfoSewer assumes that a pipe is closed if its flow split percentage is zero.

HEADLOSS AT MANHOLES

Manhole structures can induce backwater effects to their connecting sewer pipes. However, the precise hydraulic description of the flow in manhole structures is complicated because of the complex degree of mixing, separation, turbulence, and energy losses. These losses are commonly estimated as a function of headloss coefficients and are used to calculate hydraulic grade lines for upstream pipes during backwater analysis.

The headloss at a sewer manhole is determined based on the exit pipe's velocity from the following equation:

$$H_s = K\,\frac{V_{exit}V_{exit}}{2g}$$

where

H_s	=	manhole headloss, ft (m)
V_{exit}	=	exit pipe velocity, ft/s (m/s)
g	=	gravitational acceleration, 32.184 ft/s^2 (9.806 m/s^2)
K	=	headloss coefficient (unitless).

The headloss coefficient, used for estimating headloss through a manhole, depends on the type of manhole and typically ranges from 0.5 to 1.0.

Wet Wells in InfoSewer

Wet-wells are nodes with storage capacity that can vary with time during an extended period simulation. The primary input properties for wet wells are:

- The bottom elevation where the wastewater level is assumed to be zero
- The minimum level, i.e., the lowest allowable wastewater level above the bottom elevation
- The maximum level, i.e., the highest allowable wastewater level above the bottom elevation
- The initial level above the bottom elevation
- The wet-well diameter

Wet-wells are required to operate within their minimum and maximum levels. A wet-well can be modeled as a constant diameter (cylindrical) where a representative diameter is specified as the diameter corresponding to the average wet-well area. In case a wet-well can not be adequately described by a representative diameter, InfoSewer allows you to define data for the stored volume as a function of depth of wastewater using a curve (see CURVE section). This allows wet-wells of any shape (variable cross-sectional area) to be accurately modeled.

For an extended period simulation, InfoSewer accurately calculates the change in levels of a wet-well. The wastewater level is allowed to vary between the user-specified range of minimum and maximum permissible levels. If the wastewater level in the wet-well drops below the minimum level specified, the program automatically closes the pumps and no additional flow can exit the wet-well. InfoSewer models the change in wastewater level of a wet-well using the following equation:

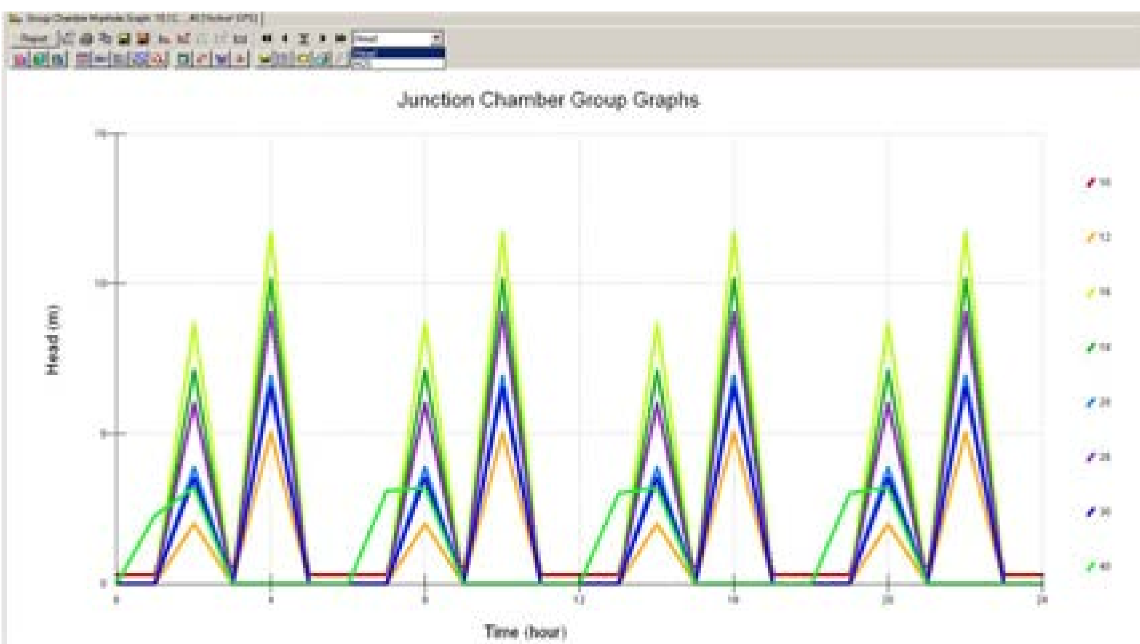
$$\Delta y = \frac{q}{A}\,\Delta t$$

where

Δy	=	change in wastewater level, ft (m)
Δt	=	time interval (sec).
q	=	flow into (+) or out of (-) wet-well (flow units)
A	=	cross-sectional area of the wet-well, ft^2 (m^2)

The results computed for wet-wells for each time period of a simulation are:

- The wastewater level
- Grade
- Volume
- The volume percentage
- The overflow rate



Pipes in InfoSewer

Pipes/open channels are links that convey wastewater from one point in the network to another. Pipes are closed conduits and are commonly used in sanitary sewer systems and combined sewer systems, whereas open channels are open to the atmosphere and their application is limited to storm sewer systems. InfoSewer can model a number of pipe/open channel shapes including circular, rectangular, trapezoidal, triangular, and parabolic. The model assumes that flow in the conduit can only be unidirectional. The principal hydraulic input parameters for conduits are:

- The conduit connectivity, i.e. from and to node IDs
- ..The upstream and downstream invert elevations
- The conduit length representing the distance between nodes
- The conduit diameter or depth, top or bottom width, left side slope, and right side slope
- The Manning roughness coefficient for gravity (open-channel) mains or the Hazen-Williams coefficient for force (pressurized) mains
- The number of (identical) parallel pipes of the same characteristics

Pipes can either be gravity mains or force mains. The main difference between the two is the hydraulic equation utilized for flow computation. The Manning equation is employed for gravity mains while the Hazen-Williams equation is used for force mains.InfoSewer assumes that a force main is always connected directly downstream of a pump, and its shape is considered as circular.

InfoSewer offers a number of useful methods to model infiltration including: count-based (e.g., defect-based), pattern-based, pipe surface area-based, pipe length-based, and pipe diameter length-based.

InfoSewer also models parallel pipes with the assumption that all parallel pipes are identical with the same characteristics (i.e. diameter, slope, length and coefficient).

MANNING AND HAZEN-WILLIAMS EQUATIONS

The Manning equation is used to determine the pipe flow in a gravity main and is given below in US Customary units:

$$Q = \frac{k}{n} AR^{2/3}S^{1/2}$$

where

Q = pipe flow (in flow units)

n = Manning roughness coefficient

R = hydraulic radius, i.e., the flow area divided by the wetted perimeter ft (m)

S = pipe slope

A = flow area, ft^2 (m^2)

k = constant (1.00 for SI units, 1.49 for US units)

The value of the Manning coefficient depends on the condition of the pipe and is therefore somewhat analogous to the pipe roughness used in pressured pipe networks.

The Hazen-Williams equation is used to determine the head loss across force mains and is given below in US Customary units:

$$h_l = \frac{kLQ^{1.852}}{D^{4.87}C^{1.852}}$$

where

L = pipe length, ft (m)

Q = pipe flow (in flow units)

C = Hazen-Williams roughness coefficient

D = pipe diameter or conduit depth for circular and non-circular shape types,

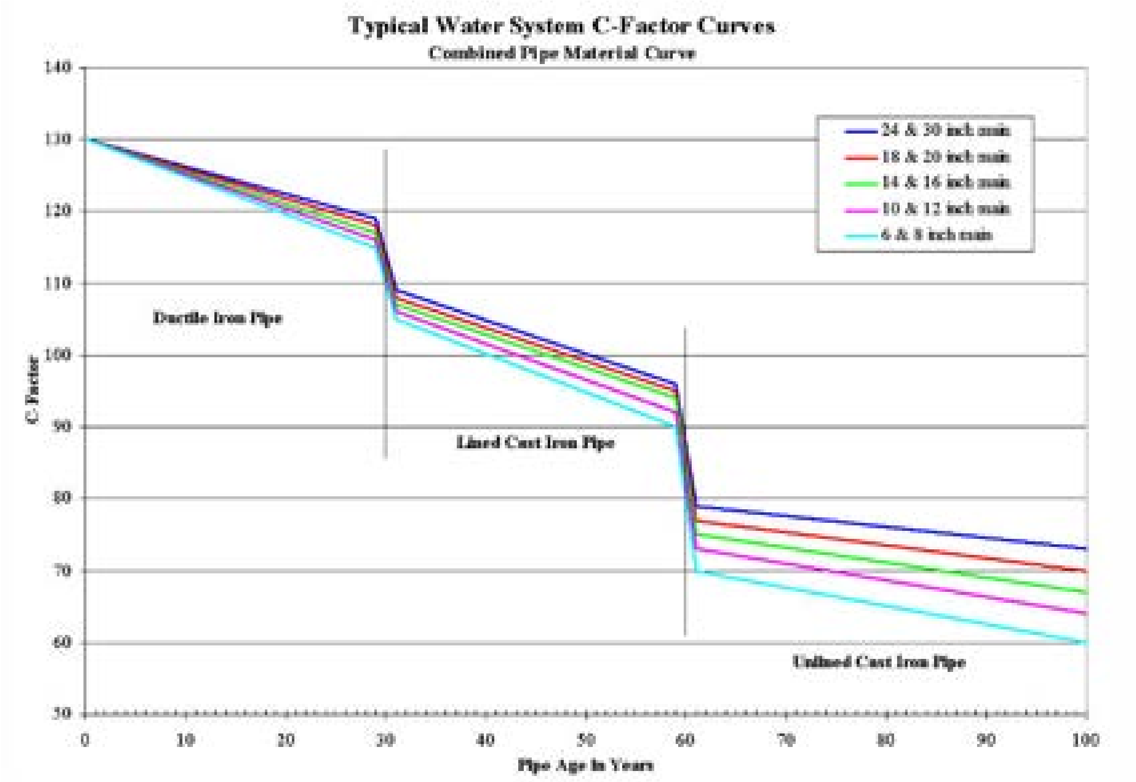
respectively. The unit is in ft (m).

k = constant (10.7 for SI units, 4.727 for US units).

Typical average design values of Manning and Hazen-Williams coefficients for commonly used sewer pipes (newly installed) are listed in the table below.

Material	Hazen-Williams C	Manning n
Asbestos Cement	140	0.011-0.015
Brass	130-140	0.009-0.013
Ductile Iron	130-140	0.012-0.015
Concrete	120-140	0.012-0.017
Copper	130-140	0.011-0.015
Steel	110-150	0.010-0.019
Plastic	140-150	0.009-0.015
Vitrified Clay Pipe	110-140	0.011-0.017
Wood Stave	115-125	0.010-0.014

Be aware that the roughness coefficient depends on type and condition of the pipe and can change considerably with age, diameter, material, soil type, and water quality characteristics. Typical water system C-factor curves are shown below (Innovyze InfoWater help file, 2016).



HYDRAULIC RADIUS

The hydraulic radius is defined as the ratio of the net cross-sectional area of a flow stream to the wetted perimeter of the section. That is:

$$R = \frac{A}{P}$$

where *P* represents the wetted perimeter, ft (m).

In the calculation of the hydraulic radius, the net cross-sectional area should be evident from the geometry of the section. The wetted perimeter is defined as the sum of the length of the boundaries of the section actually in contact with the water.

NORMAL DEPTH

Another useful form of the Manning equation is (in US Customary units):

$$A \cdot R^{2/3} = \frac{n \ Q}{1.486\, S^{1/2}}$$

where,

A = cross sectional area,
R = hydraulic radius,
S = Slope
n = manning's roughness
Q = flow

The term on the left side of the above equation is solely dependent on the geometry of the section. Therefore, for a given discharge (Q), slope (S), and surface type (n), the flow depth in the gravity main can be determined. This depth is called the normal depth. The normal depth of a force main is equal to its diameter.

CRITICAL DEPTH

The Froude number, N_F, is defined as the ratio of inertial forces to gravity forces:

$$N_F = \frac{v}{\sqrt{g \cdot \frac{A}{T}}}$$

where

v = velocity, ft/s (m/s)

A = cross sectional area, ft^2 (m^2),

T = width of the free surface of the water at the top of the pipe, ft (m)

g = gravitational constant, ft^2/s (m^2/s)

$$v = \frac{Q}{A}$$

where

v	=	velocity, ft/s (m/s)
T	=	width of the free surface of the water at the top of the pipe, ft (m)
g	=	gravitational constant, ft^2/s (m^2/s)

When the Froude number is equal to 1.0, the flow is called critical flow. When NF < 1.0, the flow is subcritical and when NF> 1.0, the flow is supercritical. The depth corresponding to the minimum specific energy is therefore called the critical depth dc. The resulting criterion for determining the critical depth using Equations (6) and (7) is:

$$\frac{Q^2 \cdot T}{g \cdot A^2} = 1$$

where,

A	= cross sectional area, ft^2 (m^2),
g	= gravity constant, ft^2/s (m^2/s),
T	= Top Width, ft (m)
Q	= flow, ft^3/s (m^3/s),

It is also important to distinguish between the following types of flows:

- Full Flow: represents the flow when the pipe is full, i.e. where the wetted perimeter is equal to the entire pipe perimeter.
- Maximum Flow: represents the maximum theoretical flow occurring in a circular pipe when the normal depth is equal to 0.938 x Diameter. Any increase in depth will decrease the flow, which is why the full flow is less than the maximum discharge for a circular pipe.
- Analysis Flow: represents the flow occurring in a pipe as derived from the analysis criteria curve (See CRITERIA CURVES section).
- Design Flow: represents the flow occurring in a pipe as derived from the design criteria curve

The computed results for pipes include:

- The total flow and velocity

- The full, maximum, design and analysis flows
- The excess full, design and analysis capacities
- The normal and critical depths
- The actual, design, and analysis d/D ratios

Force Main Network Solution

The Force Main Network Solution allows the simulation of multiple upstream and downstream force mains entering and leaving one chamber junction during an Extended Period Dynamic Simulation or EPS solution in Sewer. All of the force mains, pumps, wet wells and force main chamber junctions that are connected are considered as one force main network in the EPS solution. You can have more than one force main network in a large Sewer model separated by gravity pipes and loading manholes. The individual force main networks are solved iteratively with different upstream head and downstream tail manholes which connect the force main network(s) to the rest of the network.

A force main network consists of the following elements:

- Wet well
- Pump
- Junction Chamber
- Head Manhole where flow from other parts of the sewer system enters the force main network
- Tail manhole where the flow leaves the force main network.

The head and tail manhole for one force main network is determined by the program based on the geometry of the network. The force main network starts at a wet well, includes the pumps connecting the wet well to the force main links and also includes the actual force main links and force main connecting junction chambers. You can also connect a force main to the gravity mains without an intermediate wet well and pump(s).

The boundary conditions of the force main network are:

- Water heads at the wet wells which vary according to the inflow from the upstream sections of the sewer network and outflow to the force main network
- Water head at the tail manholes which are calculated as the maximum discharge head (invert + diameter) of all the force mains that end at that

manhole. Water entering the tail manholes will be routed downstream after the force main network flows are calculated.

For example, assuming there are n1 wet wells, n2 head manholes, n3 tail manholes, n4 junction chambers and p1 pumps and p2 force mains, the program must solve the network hydraulics to get n2+n4 water head values and p1+p2 flow values iteratively using the Newton-Raphson method. The solution iterates until the mass and energy of the force main network is in balance.

The hydraulic equations used in the solution are:

- Head/Flow relationship of the force mains and pumps (p1+p2 equations)
- Mass balance at head nodes and junction chambers (n2+n4 equations)

For head nodes, water entering the network from other sections of the sewer system must equal the flow sum of force mains that connect to it:

$$\sum_{i \in G_v} Q_i + \sum_{j \in G_f} Q_j = 0$$

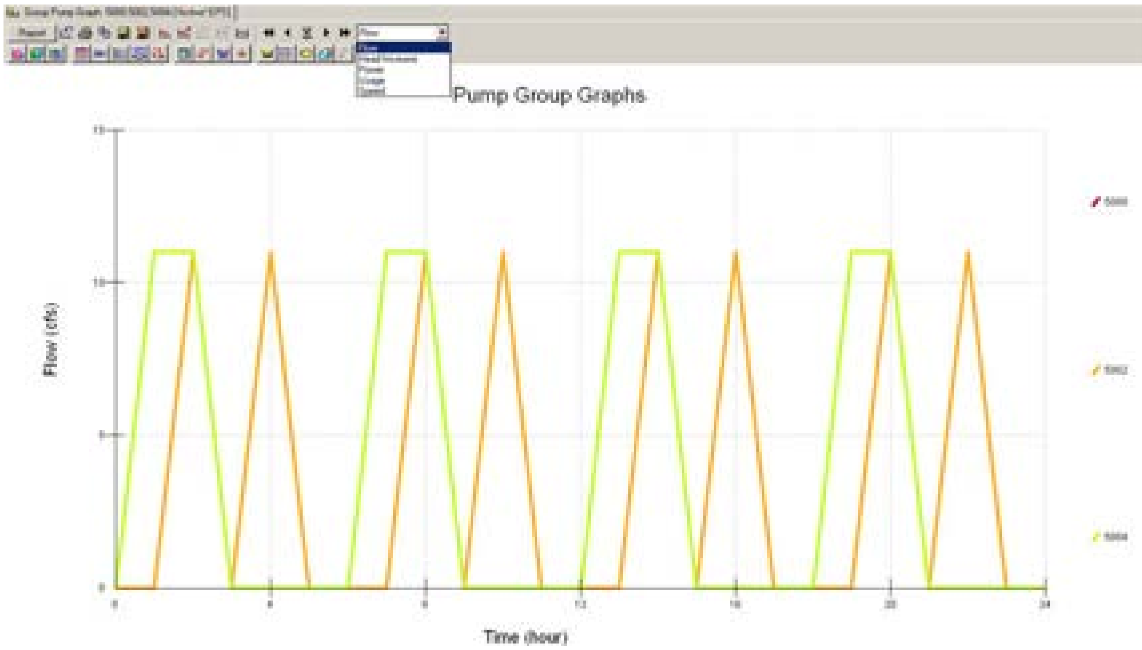
Where Q = Flow; Gv = group of gravity pipes connecting to the head manhole; and Gf = group of force mains connecting to the head manhole. The sum of the gravity flow into the wet well or head manholes is balanced by the sum or flow out of the force main network in the force main pipes.

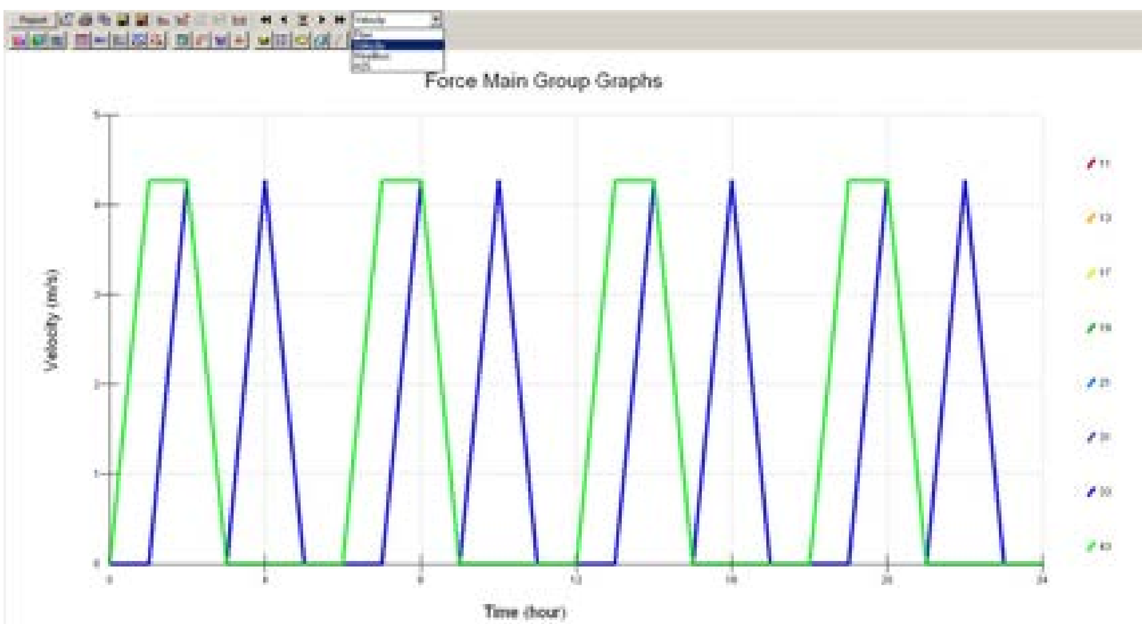
For junction chambers, which are connected to only force main pipes:

$$\sum_{j \in G_f} Q_j = 0$$

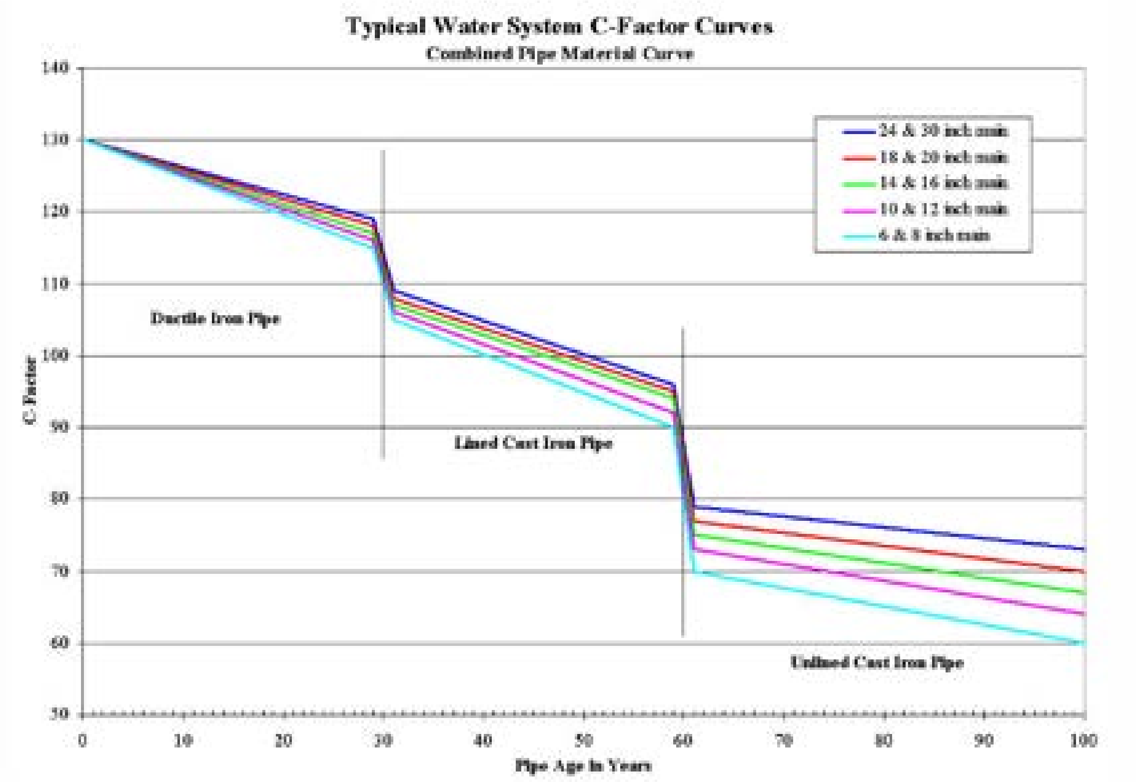
For force mains, Hazen-Williams equation describes the flow/head loss relationship within a force main. The flow out of and the flow into the junction chamber is in balance. The head at the junction manhole is iterated until the flows are in balance.

For pumps that are neither Inflow Control nor Discharge Control, the pump curve is used to estimate the flow and head gain relationship within a pump. For Inflow Control and Discharge Control pump, pump flow as control values are fixed and the equation Q = Qcontrol, where Qcontrol is the controlling pump value. For such situations, the pump is actually modeled as variable speed pump and pump speed will be calculated with Newton-Raphson method to achieve the flow control objective.





Be aware that the roughness coefficient depends on type and condition of the pipe and can change considerably with age, diameter, material, soil type, and water quality characteristics. Typical water system C-factor curves are shown below (Innovyze InfoWater help file, 2016).



Roughness Coefficients for New Pipe

Material	Hazen-Williams *C*	Darcy-Weisbach e , millifeet	Manning *n*
Asbestos Cement	140	0.005	0.011
Cast Iron	130-140	0.85	0.012-0.015
Concrete or Concrete Lined	120-140	1.0-10	0.012-0.017
Copper	135	0.005	0.011
Galvanized Iron	120	0.5	0.015-0.017
Lead	135	0.005	0.011
Plastic	140-150	0.005	0.011-0.015
Steel	140-150	0.15	0.015-0.017
Wood Stave	120	0.6	0.011-0.013

Pumps in InfoSewer

A pump is a link that imparts energy to a fluid thereby raising its hydraulic head. Pumps are needed in a gravity sewer system to prevent deep excavations, and are designed to lift the fluid to a desired level that produces gravity flow condition further downstream.

The relationship describing the head gained to a fluid as a function of its flow rate through the pump is defined as the pump characteristic curve. InfoSewer represents pumps as links of negligible length which are directly connected to wet-wells. The principal input parameters for a pump are:

- The wet-well ID representing the inlet side of the pump
- The manhole junction chamber ID representing the discharge side of the pump
- The additional number of parallel (identical) pumps with the same characteristics (optional)

InfoSewer uses different types of pump curves depending on the number of head-flow data points supplied. Three different types of pumps can be modeled as follows:

- **Fixed Capacity** representing a fixed pump flow. The pump flow is set to the fixed capacity and is independent of the flow entering the wet-well. It is assumed that the flow will transfer at the fixed rate, independent of the head requirements.
- **Single-Point Curve** - A single-point pump curve is defined by a single head-flow data point describing the desired pump operating point (design point). InfoSewer assumes that the cutoff head (at zero flow) is 133% of the design head and the maximum flow (at zero head) is twice the design flow. It then treats the curve as a three-point curve.
- **Three-Point Curve** - A three-point pump curve is defined by three points of operating data. InfoSewer will fit a continuous function of the form:

$$H_G = H_O + \alpha \cdot Q^{\beta}$$

where

H_G	=	head gain imparted by the pump, ft (m)
H_0	=	cutoff head, ft (m)
Q	=	flow through the pump (in flow units)
α	=	a resistance coefficient
β	=	a flow exponent

By supplying InfoSewer with the cutoff head H_O and two other points [(H_1, Q_1), (H_2, Q_2)], the program is able to estimate values for a and b from:

$$\alpha = \frac{H_0 - H_1}{{Q_1}^{\beta}}$$

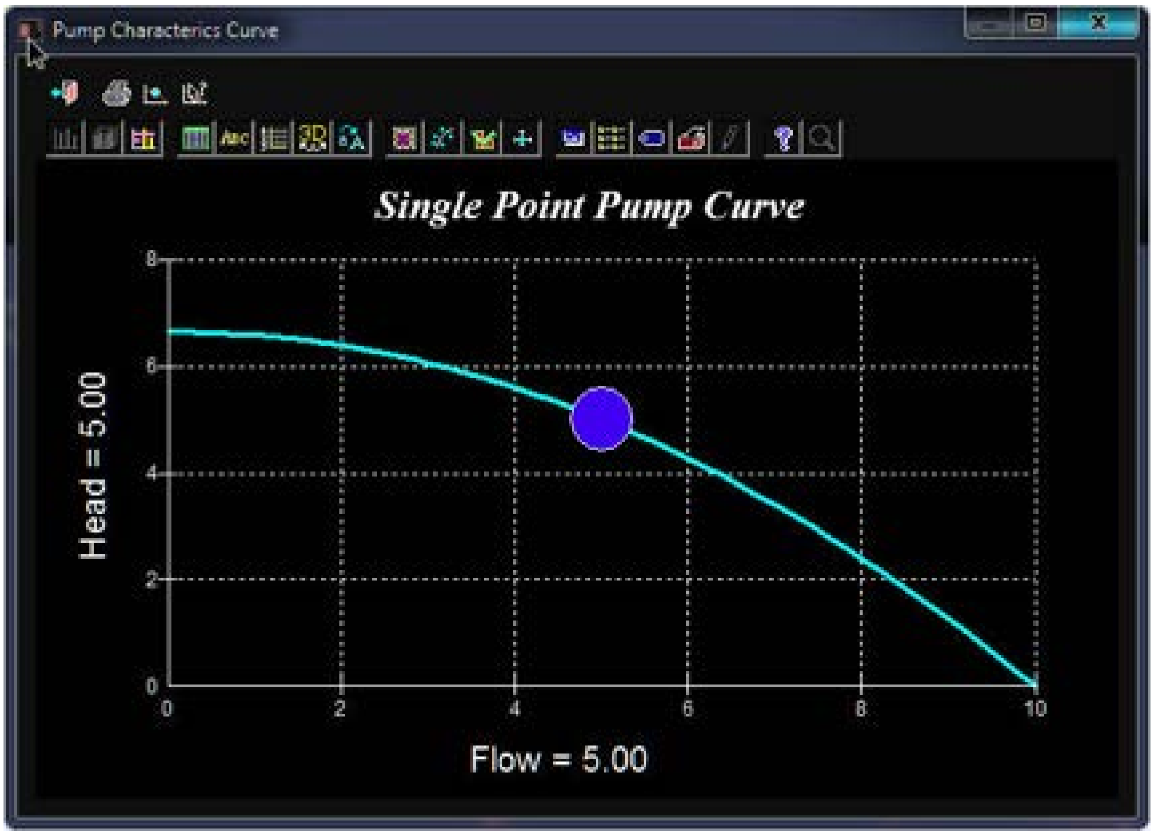
$$\beta = \frac{\log(\frac{H_0 - H_2}{H_0 - H_1})}{log\,\frac{Q_2}{Q_1}}$$

The computed results for pumps are the flow pumped and the head gained.

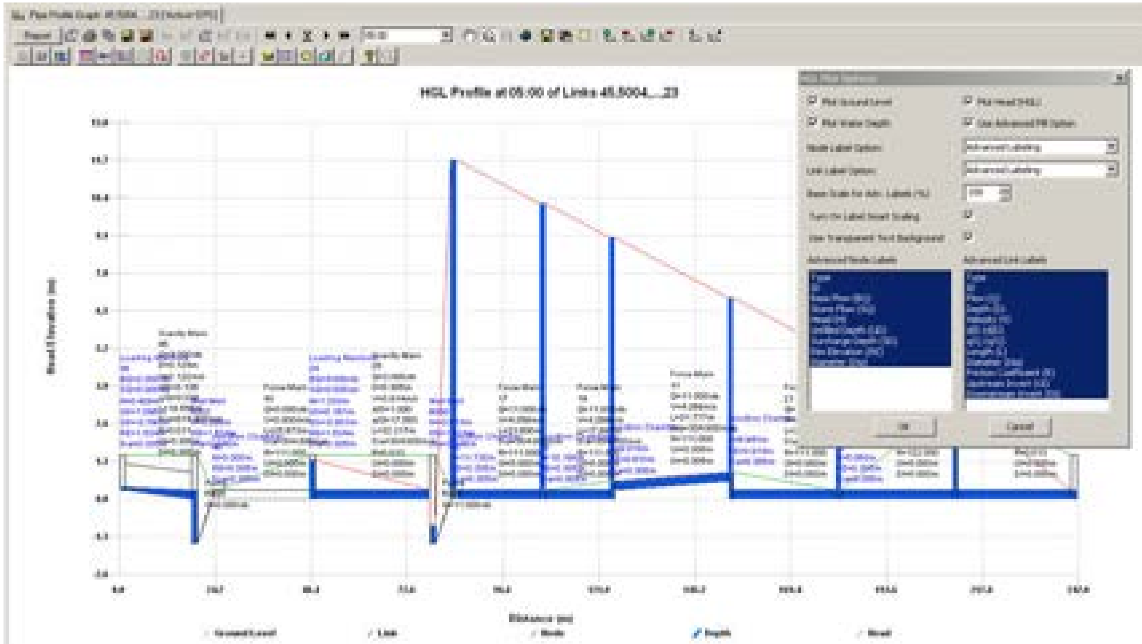
For variable speed pumps, the pump curve shifts as the speed changes. The relationships between flow (*Q*) and head (*H*) at speeds n_1 and n_2 are

$$\frac{Q_1}{Q_2} = \frac{n_1}{n_2}$$

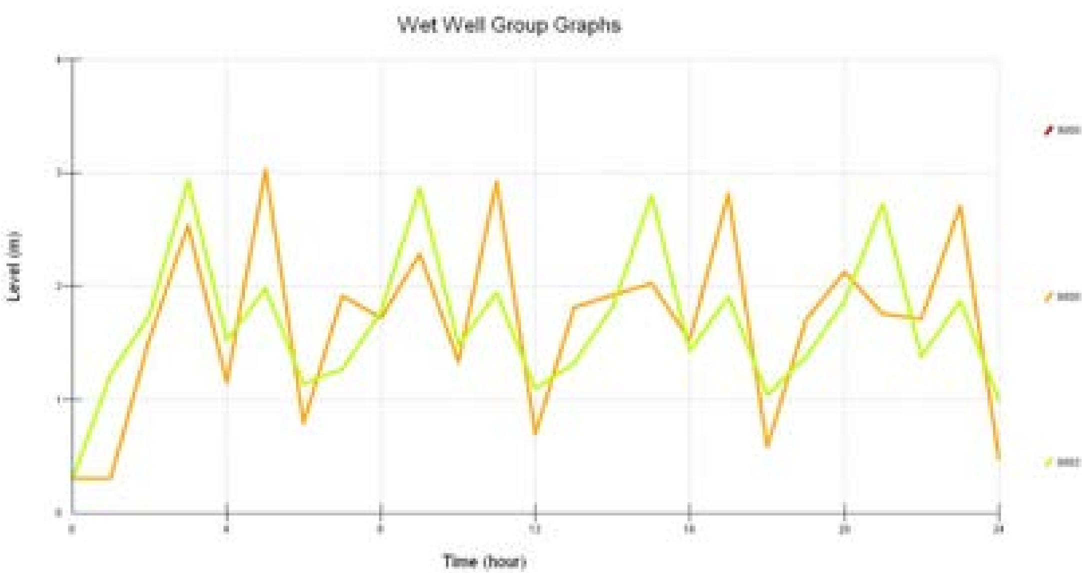
$$\frac{H_1}{H_2} = \left\{\frac{n_1}{n_2}\right\}^2$$



.



By definition, the original pump curve supplied to the program has a relative speed setting of 1. If the pump speed doubles, then the relative setting would be 2; if run at half speed, the relative setting is 0.5 and so on. The figure below illustrates how changing a pump's speed setting affects its characteristic curve.



If n denotes the pump speed ratio (n_1/n_2), then the pump characteristic curve becomes:

$$H_G = H_O n^2 - \frac{\alpha}{n^{\beta-2}} Q^\beta$$

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

5.2 NON-PHYSICAL COMPONENTS

In addition to physical components, InfoSewer employs four types of informational objects to describe the behavior and operational aspects of a sewer collection system. The informational objects are loads, curves, patterns, and controls.

Loads

Sanitary sewer system flow has two main components: sanitary or dry-weather loads and wet-weather loads. These loads are based on knowledge of the area land use patterns, wastewater generation characteristics, industries, inflow and infiltration characteristics, external flows, etc.

Sanitary or dry-weather flow results from human activity and is defined as the flow that exists in the sewer collection system during rainless periods. This flow is composed of domestic, commercial, industrial, and institutional waste. The sanitary loads are the basic data required for any hydraulic computation.

Wet-weather flow is related to rainfall activity and consists of groundwater infiltration (extraneous flow entering the sewer system because of poor construction, corrosion of the pipe, ground movement or structural failure through joints, porous walls or breaks) and structure inflow (extraneous flow entering the sewer system through manhole covers, basement drains and other sources).

For steady-state modeling, manhole loads can be either unpeaked or peaked as follows:

- **Unpeakable Flow type** - The corresponding load for each manhole is modeled as a direct flow into the sewer system.

$$Q_{Unpeakable} = Q_{Base}$$

where Q_{base} represents the average base flow (in flow units).

- **Peakable Base Flow** - InfoSewer uses a general form of the Federov's formula as follows:

$$Q_{Peakable} = K \bullet Q_{Base}^{p}$$

where *K* and *r* are peaking factor parameters.

Default values are *K* = 2.4 and *r* = 0.89. Values of *K* and *r* can be modified.

- **Peakable Coverage Flow** - InfoSewer uses the following formula which can describe both the Harman and Babbitt equations:

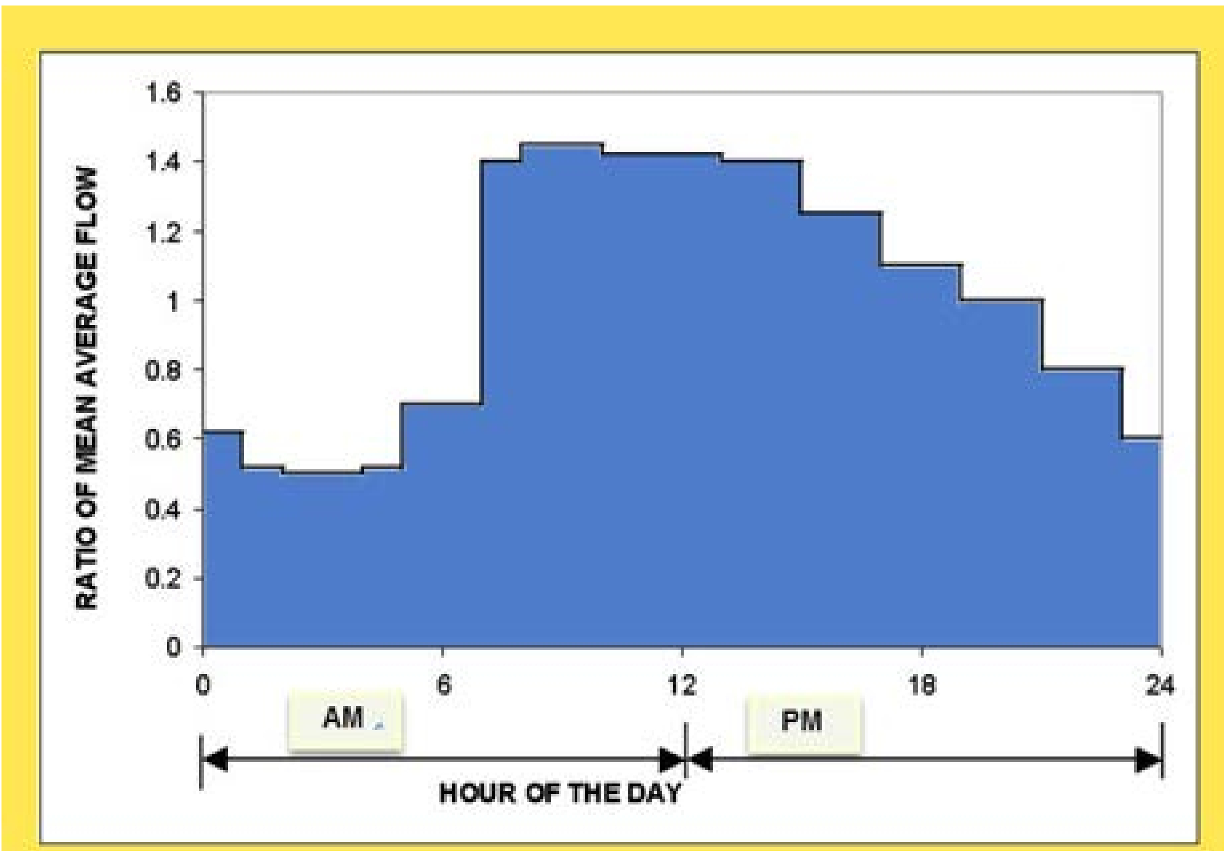
$$Q_{Peakable} = Q_{Base} \cdot \left[\frac{a}{b + \left(\frac{Population \times e}{1000} \right)^{c}} + d \right]$$

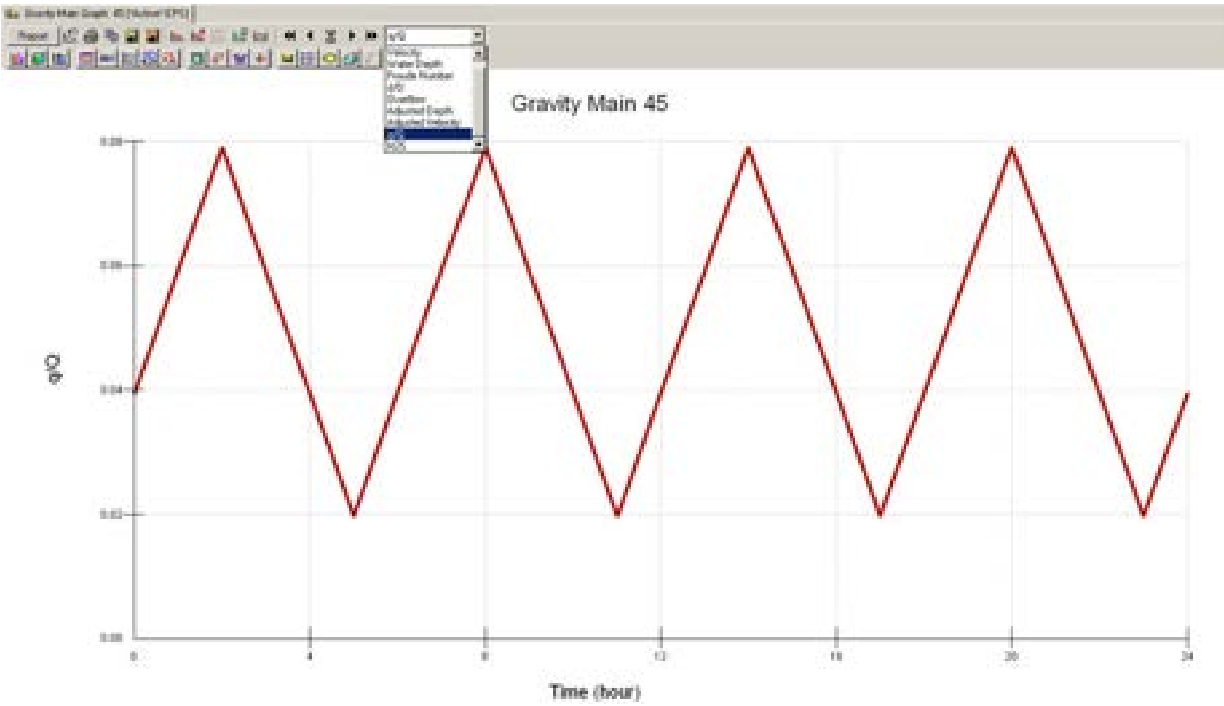
where *P* represents the population and *a*, *b*, *c*, *d* and *e* are peaking parameters. The default values for these parameters are: *a* = 5; *b* = 0; *c* = 0.2 , *d* = 0, and *e* = 1 which represents Babbitt equation (Babbitt and Baumann 1958). For the Harman equation (Babbitt and Baumann 1958): *a* = 14; *b* = 4; *c* = 0.5, *d* = 1 and *e* = 1.

$$Q_{Harman} = Q_{Base} \cdot \left[\frac{14}{4 + \left(\frac{Population \times 1}{1000} \right)^{1/2}} + 1 \right]$$

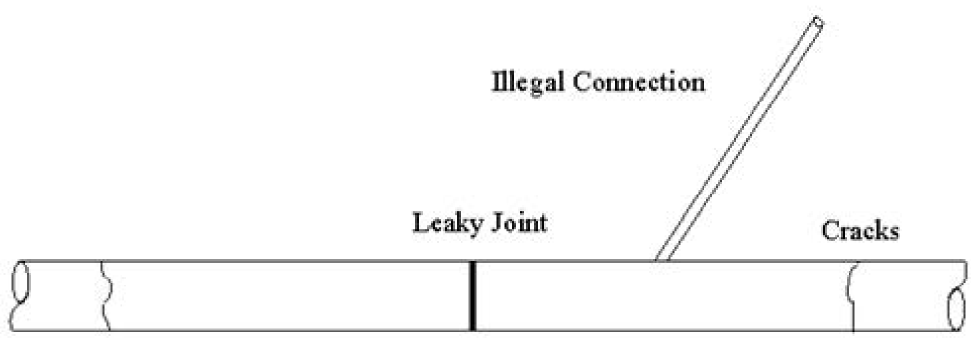
$$Q_{Babbitt} = Q_{Base} \cdot \left[\frac{5}{0 + \left(\frac{Population \times 1}{1000} \right)^{0.2}} + 0 \right]$$

For an extended period simulation, no peaking formula is used, instead, the multiplication factors from the diurnal pattern are used to adjust (multiply) all types of loads before they are aggregated. An example peaking-factor pattern is shown below.





Infiltration and inflow affect the operation of a sanitary sewer system and pumping, treatment, and overflow regulators facilities.



Infiltration occurs in gravity pipes while inflow occurs at manholes and wet wells. Infiltration loads refer to the volume of groundwater entering the sewer system from the soil through defective joints, broken or cracked pipes, improper connections, or manhole walls. Accurately determining infiltration is generally difficult as these loads depend on soil type, soil moisture conditions, system size and integrity, water table level, and the number of illegal connections. They are normally computed by subtracting base flow from total metered flow during dry weather or by compiling flow isolation measurements. Infiltration can be defined as proportional to the pipe length; proportional to the pipe length and to the pipe diameter; proportional to the pipe surface area (pipe length multiplied by its perimeter); proportional to the number of defects in the pipe (count-based); or as a pattern load/hydrograph (flow vs. time).

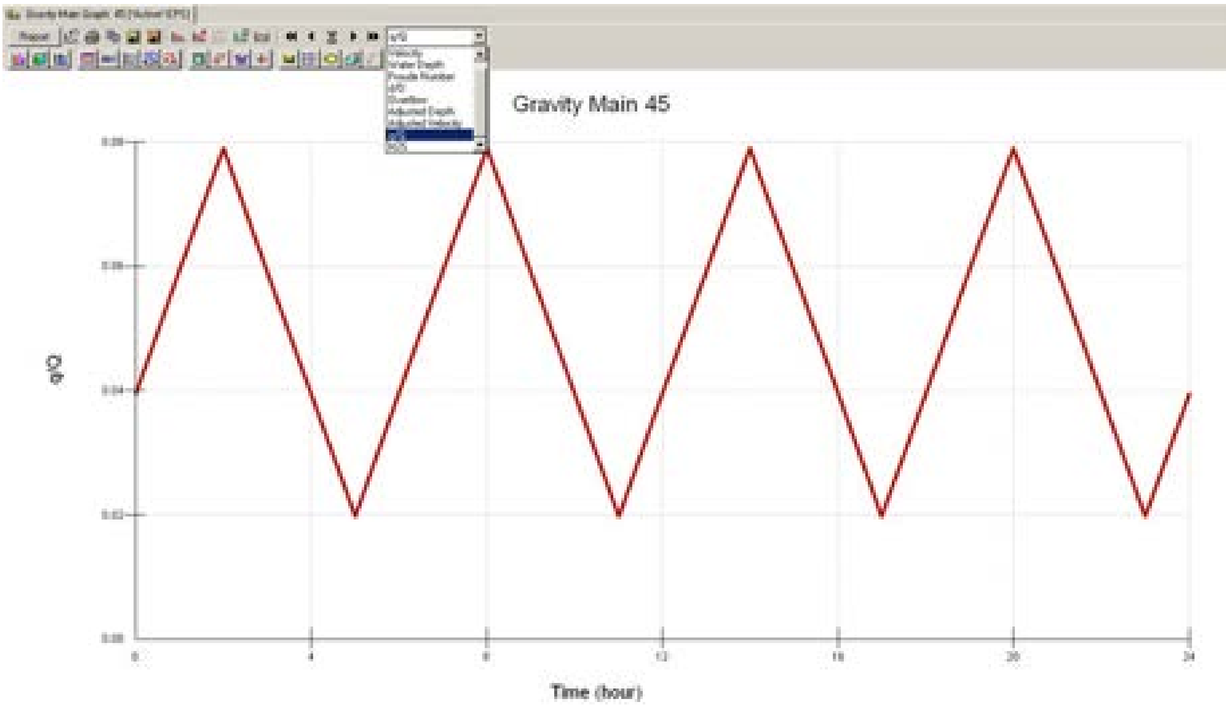
Inflow loads refer to stormwater or other drainage water and wastes (extraneous water) entering the sewer system through manhole covers. Inflow is measured during wet weather conditions and is determined by subtracting base flow and infiltration from data recorded during wet weather conditions. Inflows can be specified as pattern loads/hydrographs (flow vs time) for any manhole.

Patterns

Patterns are used to represent temporal variations within the system. They consist of a collection of multipliers (multiplication factors) that are applied to a base load to allow it to vary over time during an extended period simulation. The time interval used in all patterns is a fixed value set by the user. Although all patterns must utilize the same time interval, each can have a different number of periods. If the duration of a pattern is less than the total duration of the simulation, then the pattern will repeat itself and will wrap around to its first period again.

Two options are available for representing a pattern: stepwise or continuous (linear). A stepwise pattern is one that assumes a constant multiplication factor for each pattern time period. Within each time period a quantity remains at a constant level equal to the product of its nominal value and the pattern's multiplier for that time period. A continuous (linear) pattern is one that linearly interpolates for the multiplication factors between two adjacent time periods.

Different patterns can be applied to individual manholes or groups of manholes to accurately represent actual loading categories (e.g., low density residential, commercial, and industrial).



As an example of how patterns work consider a manhole with an average load of 2.0 CFS. Assume that the pattern time interval has been set to 4 hours and with the following multipliers:

Period	1	2	3	4	5	6
Multiplier	0.5	0.8	1.0	1.2	0.9	0.7

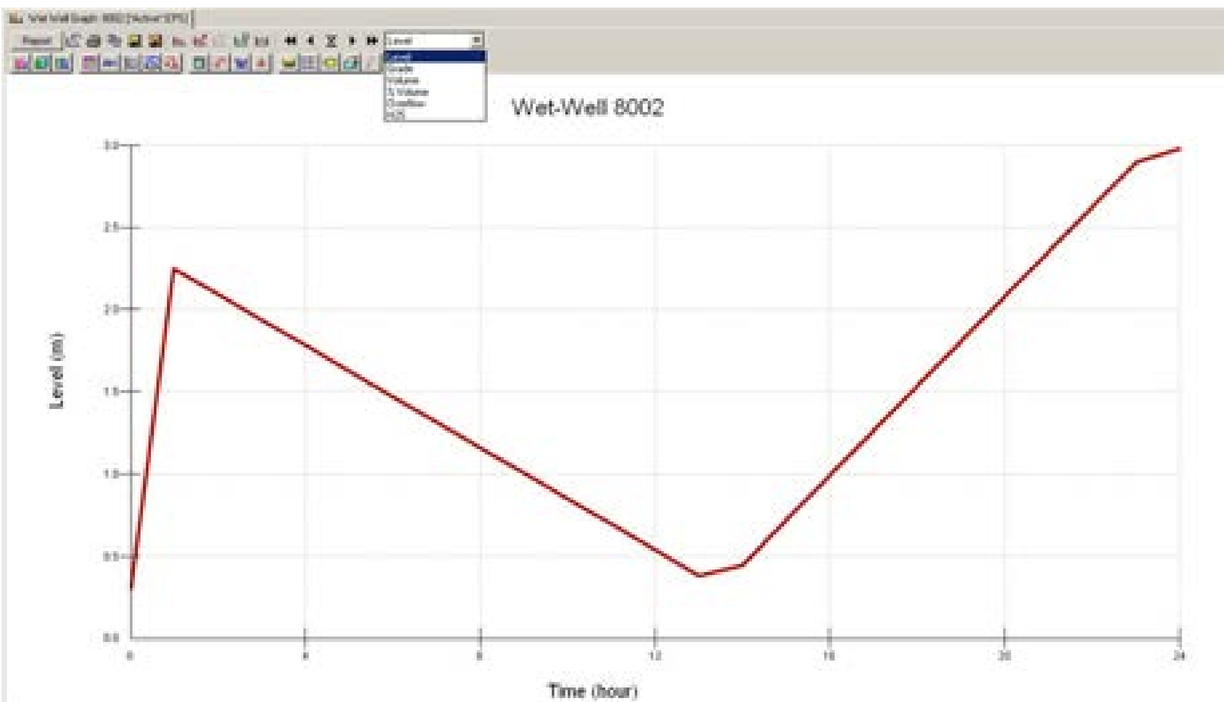
Then during the simulation, the actual load collected for this manhole will be as follows:

Hours	0-4	4-8	8-12	12-16	16-20	20-24
Load	1.0	1.6	2.0	2.4	1.8	1.4

Curves

Curves are objects that contain data pairs representing a relationship between two quantities. Two or more objects can share the same curve. An InfoSewer Pro model can utilize the following types of curves:

- Volume curve
- Peaking curve
- Flow split curve
- Design and analysis criteria curves
- Replacement and duplication design cost curves



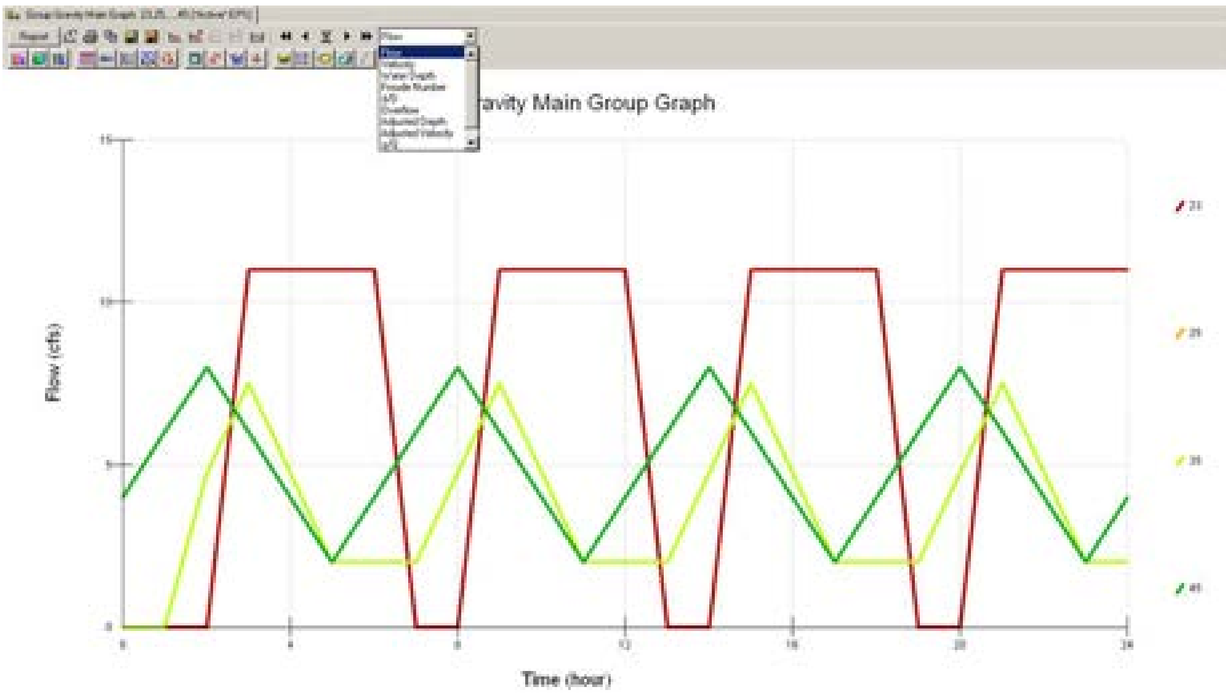
VOLUME CURVE

A volume curve determines how the wet-well volume (Y in cubic feet or cubic meters) varies as a function of the wastewater level (X in feet or meters). It is used when it is necessary to accurately represent a wet-well whose cross-sectional area varies with height (e.g., non-circular wet-wells). The lower and upper wastewater levels supplied for the curve must contain the lower and upper levels between which the wet-well operates. A valid volume curve must have increasing volume with increasing water height.

	ID	From ID	To ID	Diameter (in)	Length (ft)	Slope	Flow (cfs)	Flow Type	Velocity (ft/s)	Water Depth (ft)	Froude Number	Overflow (MG)	Backwater Adjustment	Adjusted Depth (ft)	Adjusted Velocity (ft/s)	d/D	q/Q	H2S (mg/l)
1	23	22	14	12.000	104.817	0.000	22.000	Pressurized	28.011	1.000	4.936	0.000	No	1.000	28.011	1.000	194.752	0.000
2	25	24	9000	60.000	105.700	0.000	7.500	Free Surface	0.510	3.503	0.050	0.000	No	3.503	0.510	0.701	0.838	0.927
3	35	24	8000	60.000	170.772	0.000	7.500	Free Surface	0.510	3.503	0.050	0.000	No	3.503	0.510	0.701	0.838	0.927
4	45	36	9002	12.000	60.879	0.000	4.000	Pressurized	5.093	1.000	0.898	0.000	Yes	1.000	5.093	1.000	29.962	0.961

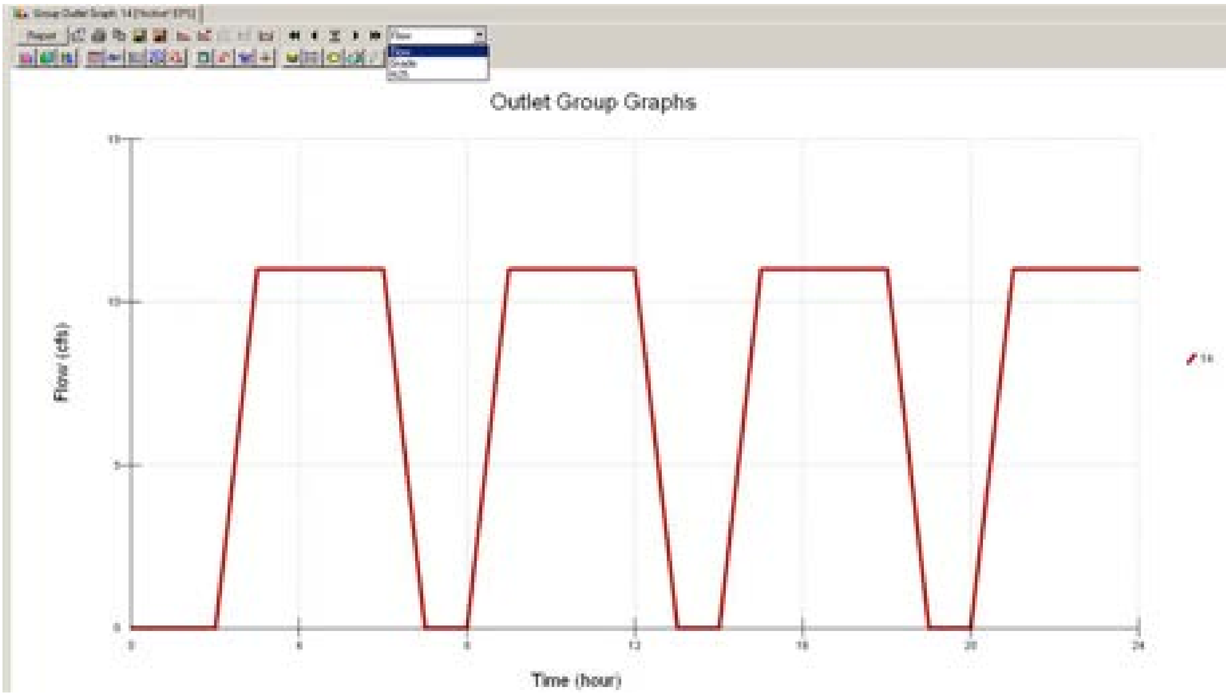
PEAKING CURVE

The peaking curve represents the variation of peak flows (Y-axis) as a function of base flows (X-axis) and is an alternative approach to compute flow data (loads) for peak conditions. Any peaking curve can be specified to estimate base flow peaks and model peak flow - base flow relationships.



FLOW SPLIT CURVE

A flow split curve determines how the flow split (Y in percentage or flow units depending on the desired split method) varies as a function of the total flow in the manhole (X in flow units). If the splitting method is the Variable Flow Split Percentage method then the Y data are specified in percentage. If the Inflow-Outflow Flow Split method is used then the Y data are specified in flow units. The data points must be defined in an increasing order of flow.



CRITERIA CURVES

Criteria curves are entered as a series of pipe diameters (X in inches or millimeters) along with their acceptable depth-to-diameter (d/D) ratios (Y unitless). The analysis criteria curve is used for determining the d/D ratios of existing pipe capacities while the design criteria curve is used for designing relief or replacement pipes when their capacity as determined by the analysis criteria has been exceeded. The data points specified for the above curves must be defined in an increasing order. For example, existing pipes up to 15 inch in diameter are allowed to flow only half full, up to 21 inch pipe can flow three-quarters full, and all other pipes are allowed to flow at full capacity.

DESIGN COST CURVES

A design cost curve determines how the design unit costs (Y in cost currency per unit of length) varies as a function of the diameter of the pipe to be designed (X in inches or millimeters). The user specifies one curve for each type of improvements, i.e., the construction of a new pipe (replacement method) and the placement of a parallel/relief pipe (duplication method). The data points for the above curves must be defined in an increasing order. Only those pipe diameters defined will be considered as candidate design sizes. Each pipe can also have specific design cost tables for replacement or duplication. The cost currency is user-specified.

DESIGN TABLE

InfoSewer supports modeling of circular and non-circular conduits for all the hydraulic simulations performed by the model (i.e., steady state analysis, design, and dynamic simulations). Unlike circular pipes that can be fully described in terms of a single input parameter (i.e., diameter), geometry of non-circular conduits is a function of multiple variables such as channel depth, channel width, and side slopes. As a result, the carrying capacity and the costs associated with replacing and duplicating non-circular pipes cannot be defined in terms of a single input parameter, thus limiting application of criteria curves and cost curves to circular pipes only.

For non-circular channels, the model provides a design table through which the user may supply inputs related to conduit sizes, flow depth to channel depth ratio for analysis as well as design criteria, replacement cost, and duplication costs for non-circular channels. The model calculates conveyance factor of the channel (a measure of channel's carrying capacity) and associates this factor with their respective criteria and cost values. As described earlier, InfoSewer uses Manning equation to determine pipe flow in gravity mains. For Manning's equation, the conveyance factor is given as:

$$K_f = AR^{2/3}$$

where

K_f = conveyance factor, ft8/3 (m8/3)

R = hydraulic radius, i.e., the flow area divided by the wetted perimeter, ft (m)

A = flow area, ft^2 (m^2)

Controls

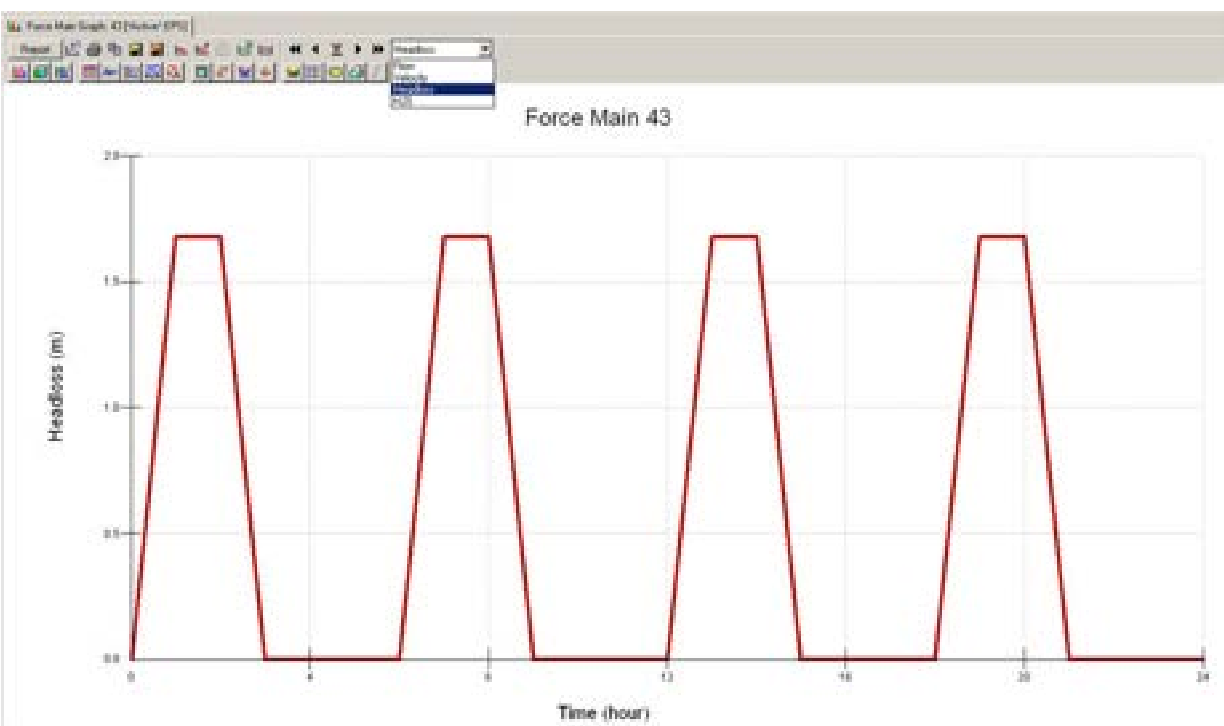
InfoSewer provides comprehensive operational control schemes to accurately simulate the dynamic hydraulic behavior of a sanitary sewer collection system. During an EPS, controls describe the on-off status and relative speed setting of selected pumps as a function of the flow levels or volumes of wet-wells, or to match a targeted pump discharge flow.

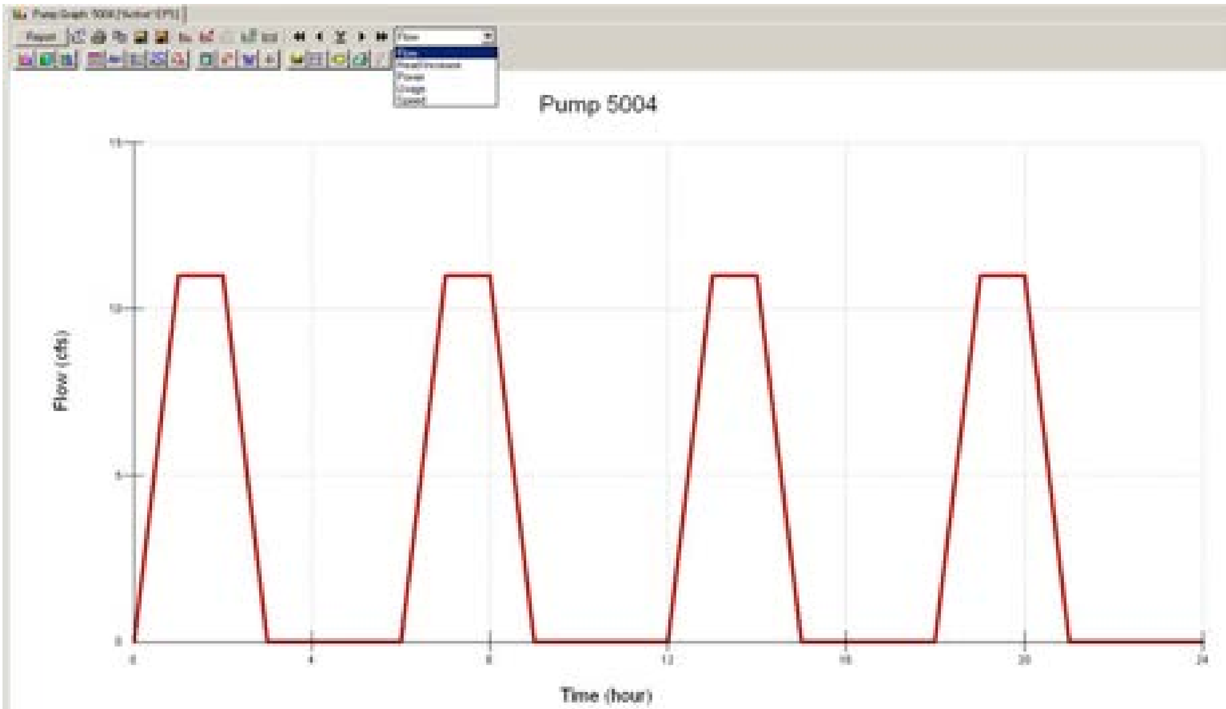
Level controls are stated in terms of the height of wastewater above the wet-well bottom elevation. The default “ON Setting” value is the top wastewater level/volume of the wet-well while the default “OFF Setting” is the bottom wastewater level/volume of the wet-well.

An example of level control is given below:

IF (level in wet-well WW1 drops below 2 feet) THEN (turn OFF pump P1)

The default pump speed setting is one (pump speed ratio of 1). The initial pump status is overwritten by the operational controls during an extended period simulation.





Pump 5004
Flow (cfs)
Time (hour)

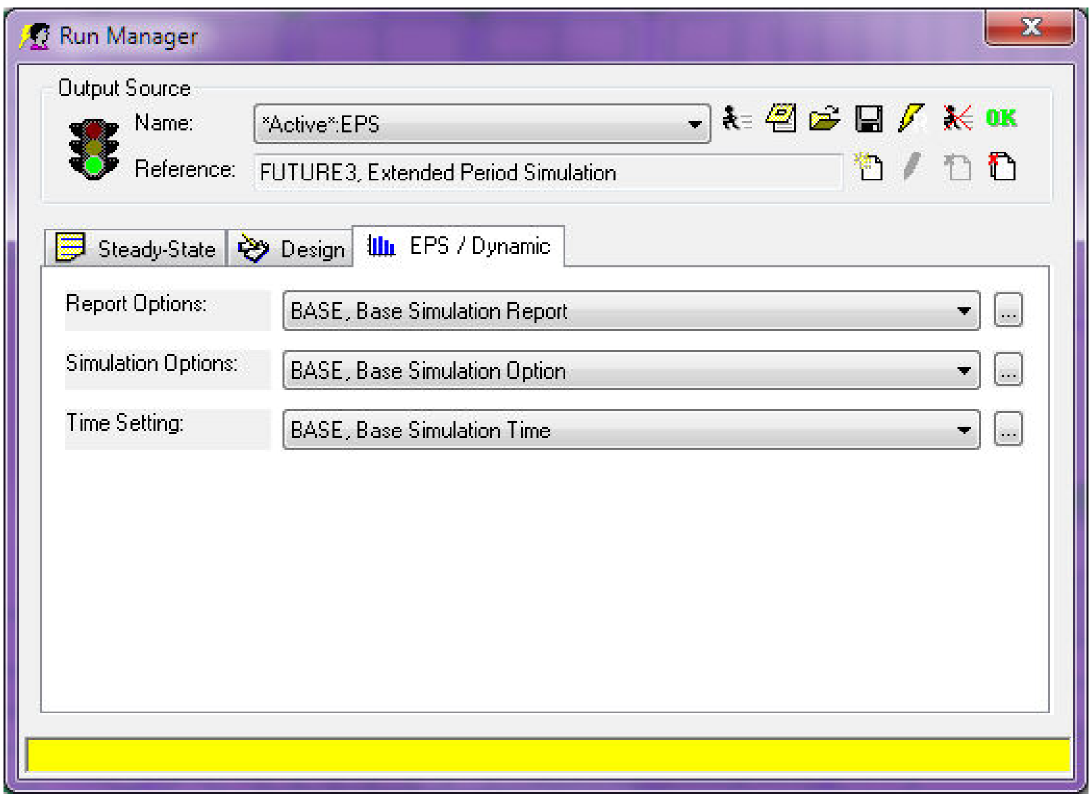
InfoSewer™

Modeling Made Easy

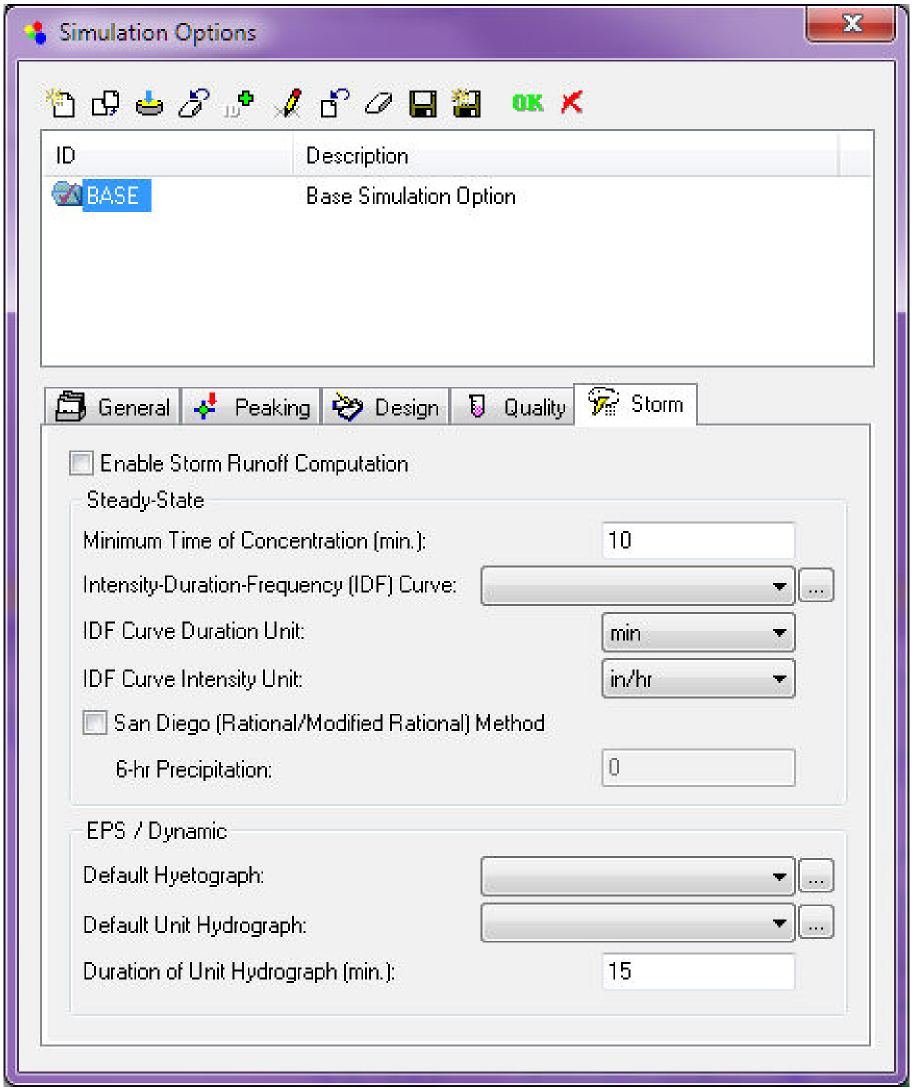
InfoSewer™
Modeling Made Easy

Three Types of Simulation Runs in InfoSewer

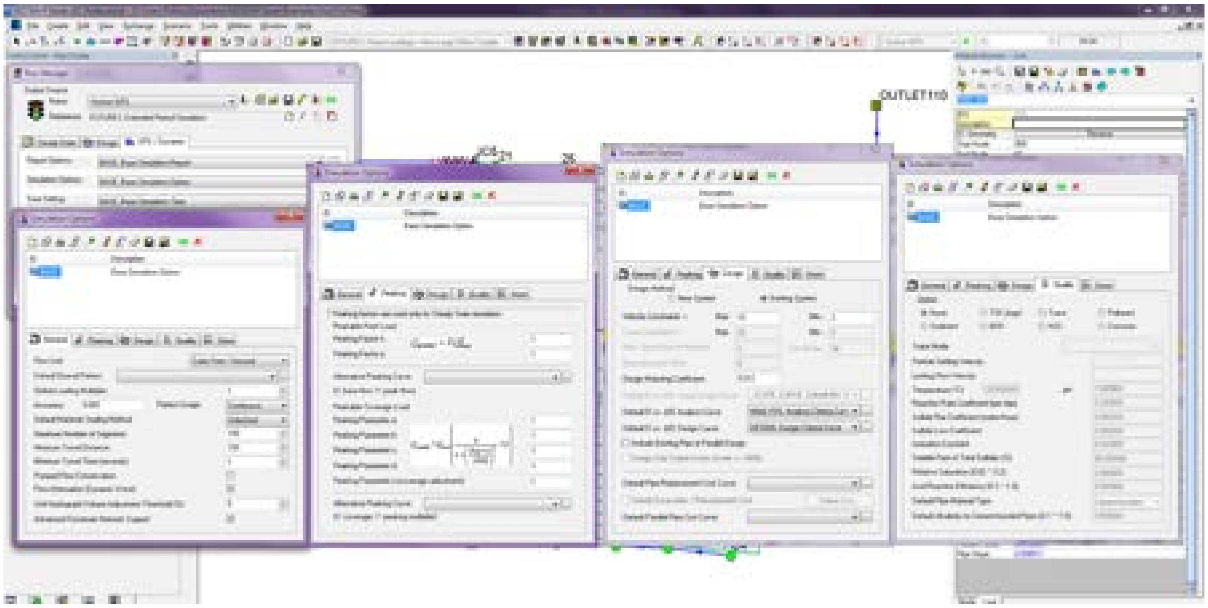
Three different types of hydraulic analyses can be carried out by InfoSewer and are explained below in the following sections. Here are the three **Run Manage**r Tabs



InfoSewer Pro Only has the Storm Tab, if you do not have the Pro version you will not be able to simulate RDII or Stormwater Runoff



InfoSewer and InfoSewer Pro allow you to use the General, Peaking, Quality and Design Tabs


OUTLET110

Steady State InfoSewer

The purpose of a sanitary sewer system is to convey wastewater from various origins at various rates of flow. The maximum and minimum flow rates in a single day can vary greatly. The system should be able to carry the maximum rate of flow without backing up to any significant degree and within the acceptable velocity limit. The system should also be able to convey the minimum flow without deposition of suspended solids. InfoSewer allows you to effectively simulate the system's hydraulic behavior at any specific time period and analyze it under various conditions. Hydraulic calculations are based on the assumptions of one-dimensional, incompressible, steady flow with constant rate of flow between concentrated inflows or outflows. The calculations deal primarily with change in depth and velocity of flow along the sewer (ASCE 1982).

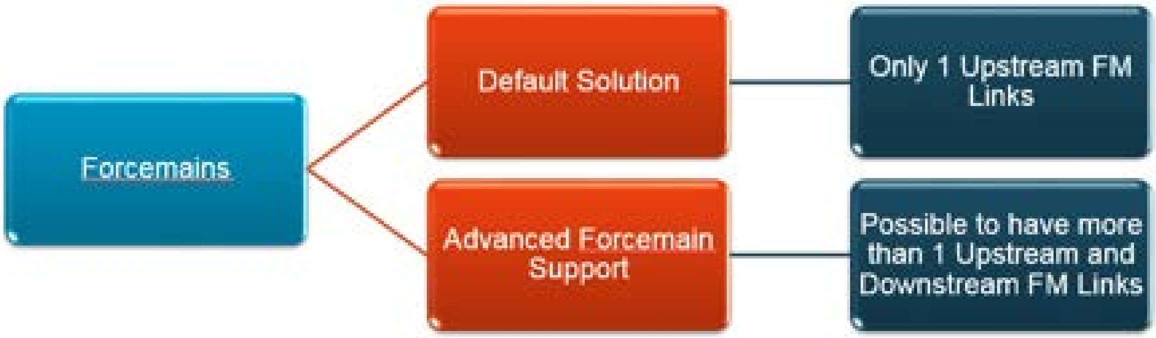
For steady-state analysis, all flows are assumed to accumulate in the system and discharge only at the outlets. This means that even if a pipe has a flow beyond its maximum capacity, the flow is still carried downstream including through pumps and force mains. The transition between gravity flow and pressurized flow is also ensured by assuming that all flows are transported through each force main, subject to the upstream hydraulic control.

Peaking factors and the peaking Tab are a key difference between Steady State and EPS simulations.

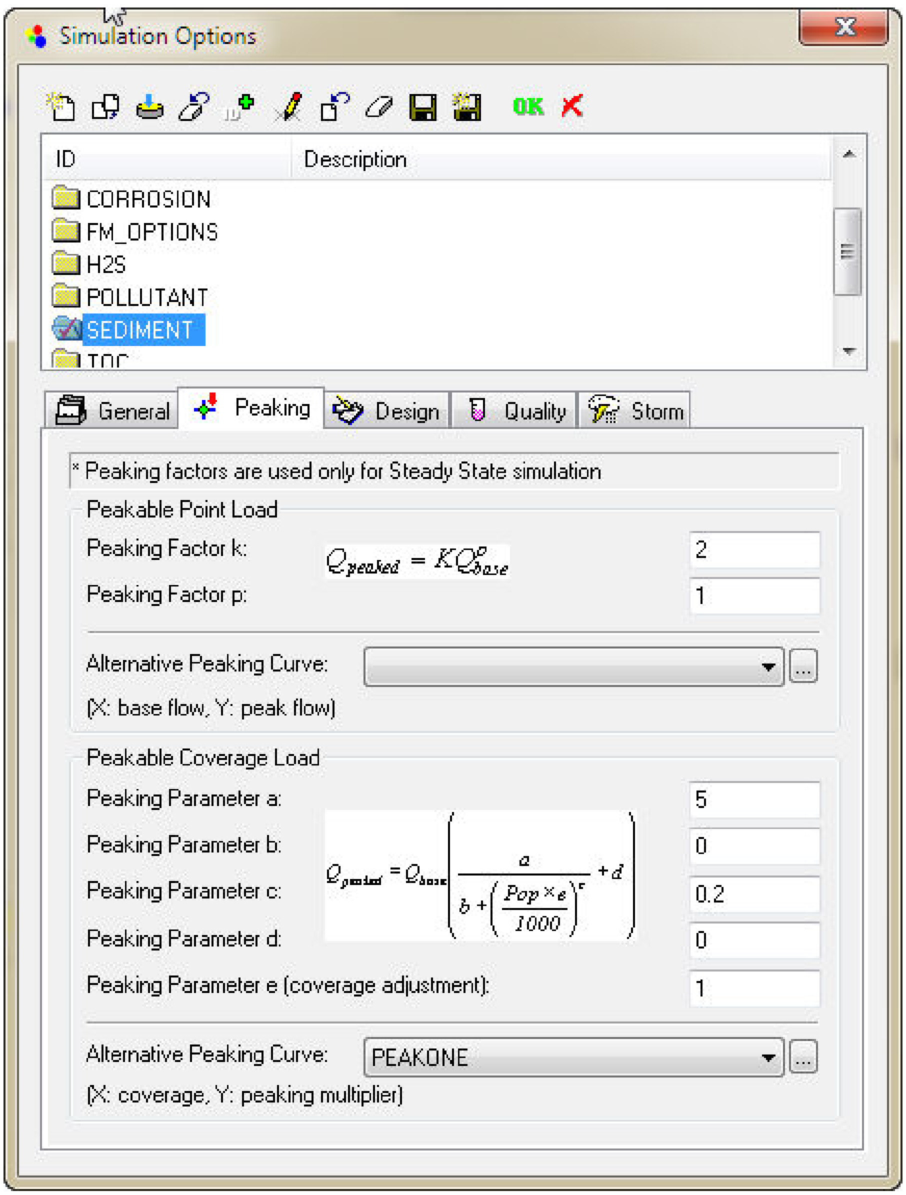
Another key difference is that complicated upstream and downstream Force Mains are not allowed in the Steady States solution. Error 1352 is a typical error message

Error 1352: invalid split-flow configuration for given force-main without advanced forcemain network support

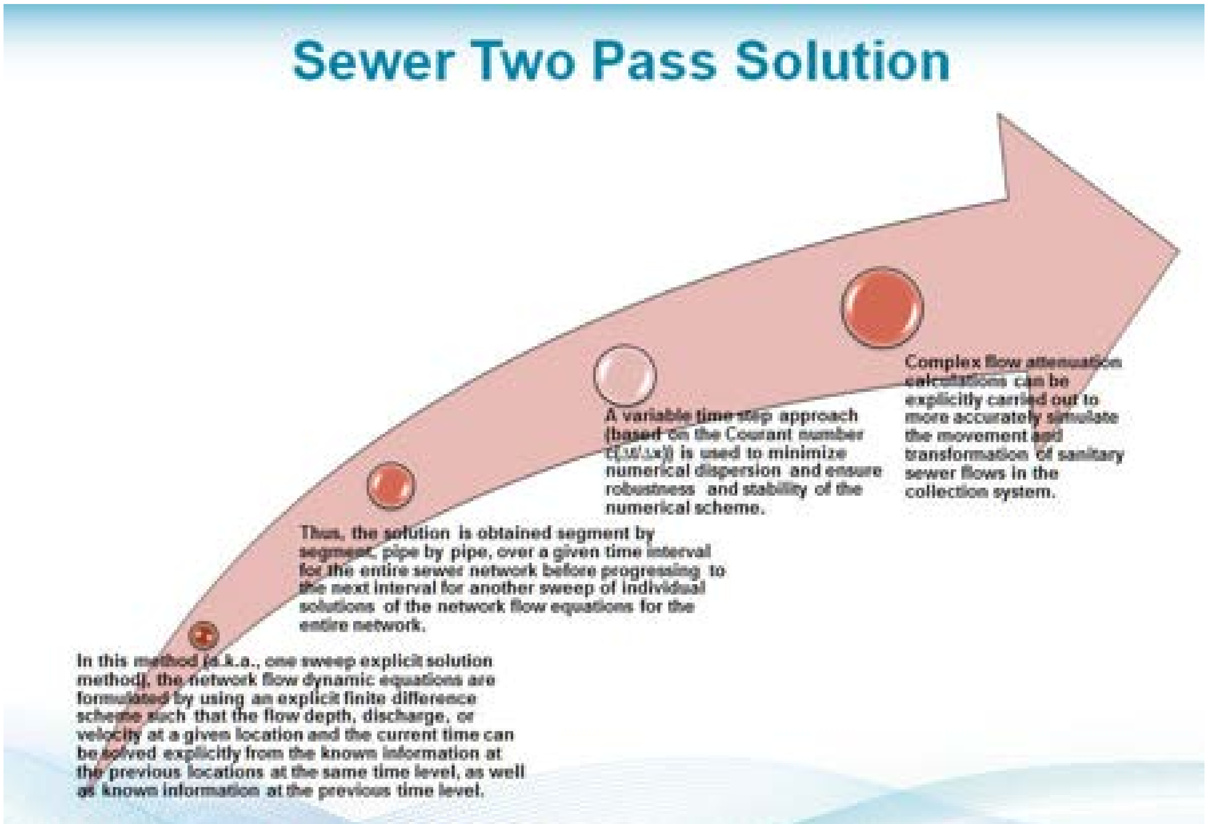
65 10 30 0.000000 0.000000 182.051359 6.000000 111.000000 0

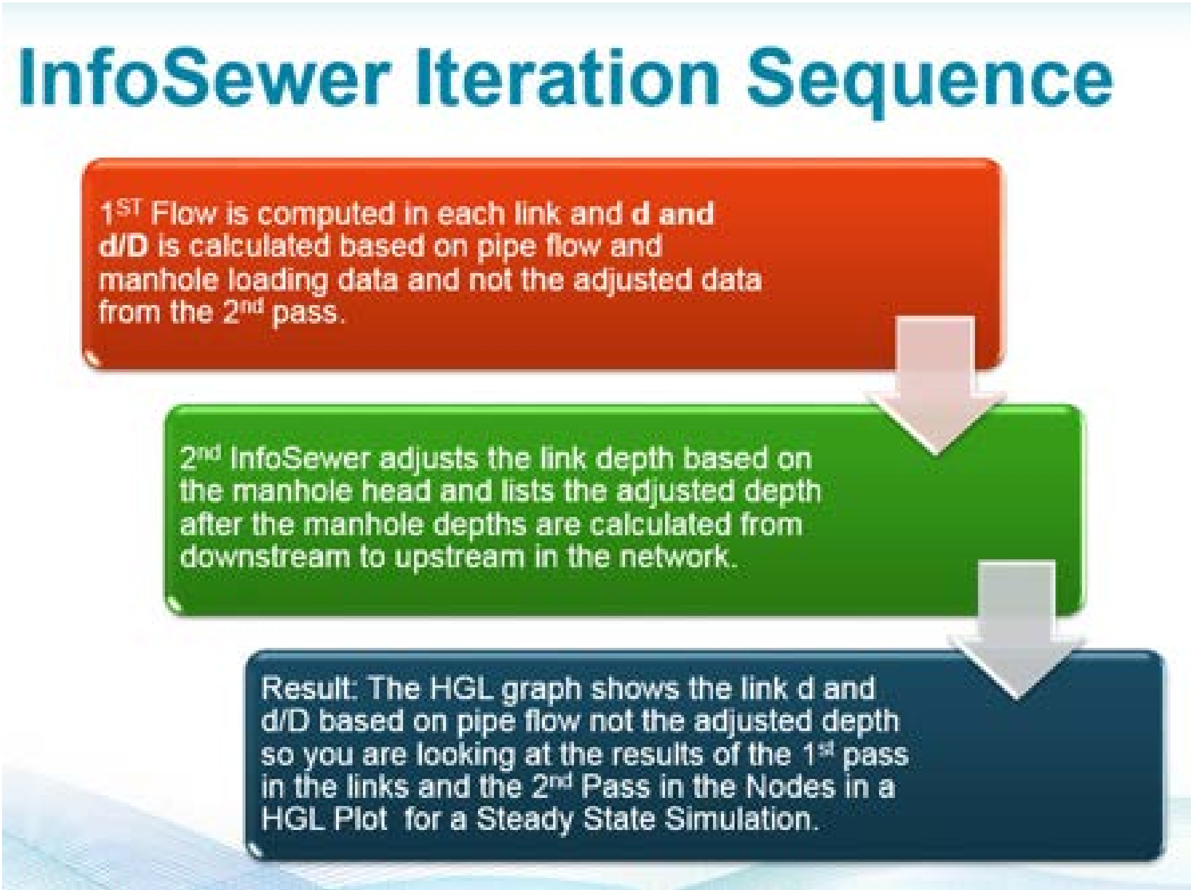


Forcemains
Default Solution
Advanced Forcemain Support
Only 1 Upstream FM Links
Possible to have more than 1 Upstream and Downstream FM Links

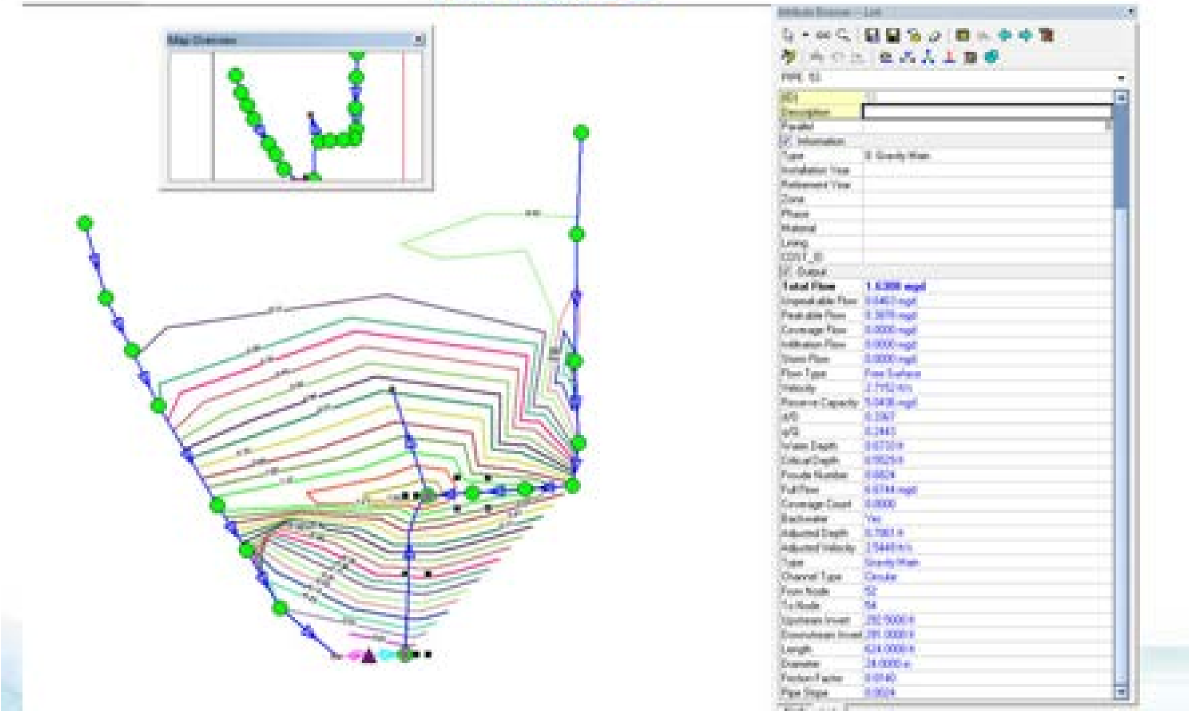


Simulation Options
OK
ID
Description
CORROSION
FM_OPTIONS
H2S
POLLUTANT
SEDIMENT
TOC
General
Peaking
Design
Quality
Storm
* Peaking factors are used only for Steady State simulation
Peakable Point Load
Peaking Factor k:
Peaking Factor p:
2
1
Alternative Peaking Curve:
(X: base flow, Y: peak flow)
Peakable Coverage Load
Peaking Parameter a:
Peaking Parameter b:
Peaking Parameter c:
Peaking Parameter d:
Peaking Parameter e (coverage adjustment):
5
0
0.2
0
1
Alternative Peaking Curve:
PEAKONE
(X: coverage, Y: peaking multiplier)

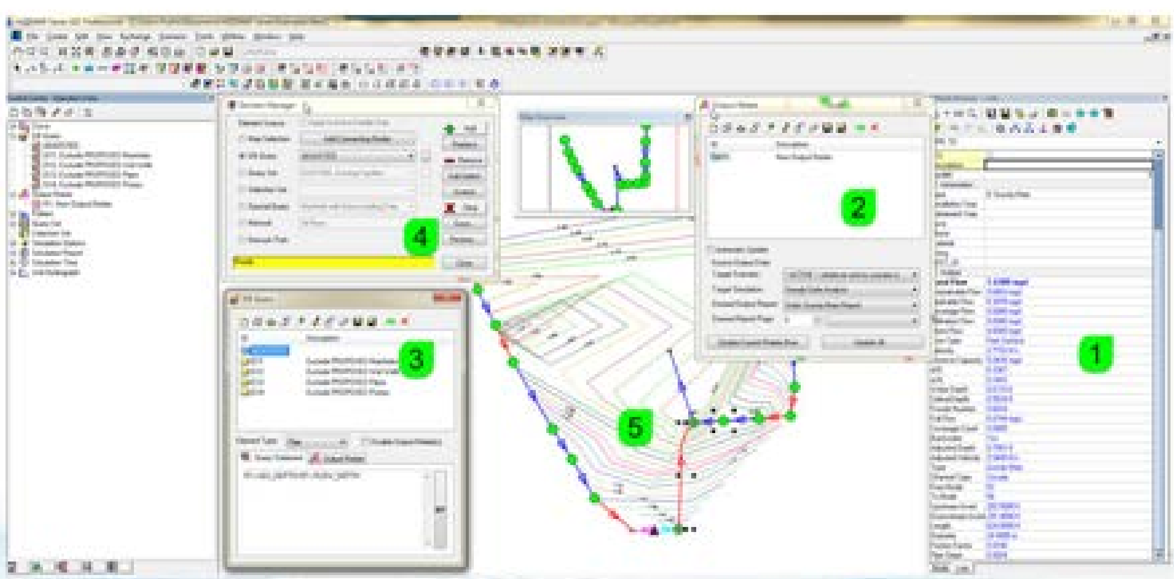

Sewer Two Pass Solution
In this method (a.k.a., one sweep explicit solution method), the network flow dynamic equations are formulated by using an explicit finite difference scheme such that the flow depth, discharge, or velocity at a given location and the current time can be solved explicitly from the known information at the previous locations at the same time level, as well as known information at the previous time level.
Thus, the solution is obtained segment by segment, pipe by pipe, over a given time interval for the entire sewer network before progressing to the next interval for another sweep of individual solutions of the network flow equations for the entire network.
A variable time step approach (based on the Courant number C(Δt/Δx)) is used to minimize numerical dispersion and ensure robustness and stability of the numerical scheme.
Complex flow attenuation calculations can be explicitly carried out to more accurately simulate the movement and transformation of sanitary sewer flows in the collection system.


InfoSewer Iteration Sequence
1ST Flow is computed in each link and d and d/D is calculated based on pipe flow and manhole loading data and not the adjusted data from the 2nd pass.
2nd InfoSewer adjusts the link depth based on the manhole head and lists the adjusted depth after the manhole depths are calculated from downstream to upstream in the network.
Result: The HGL graph shows the link d and d/D based on pipe flow not the adjusted depth so you are looking at the results of the 1st pass in the links and the 2nd Pass in the Nodes in a HGL Plot for a Steady State Simulation.

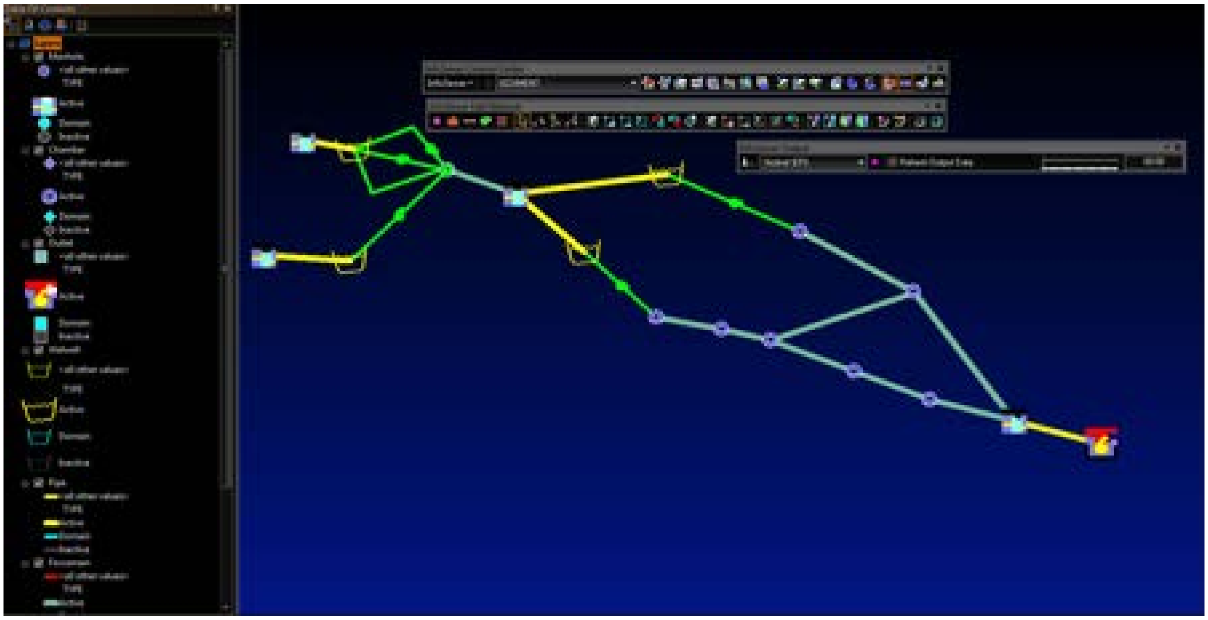
Adjusted Depth and Velocity are 2nd Pass Variables



Adjusted Depth and Velocity are 2nd Pass Variables in the Steady State and EPS Solutions



Flooding at manholes and wet-wells in InfoSewer is not modeled during an extended period dynamic simulation. Instead, the



Design in InfoSewer

STEADY STATE DESIGN

A sanitary sewer collection system has basically two main functions: to convey the designed peak discharge and to transport solids so that deposits are kept to a minimum. It is imperative, therefore, that the sanitary sewer has adequate capacity for the peak flow and that it functions at minimum flows without excessive maintenance and generation of odors. InfoSewer uses an efficient methodology that allows you to accurately design portions of the sewer network, from a single pipe to the entire system. In this explicit design method each sewer pipe is designed independently without consideration of the flows in other pipes. This can be done because to design a sewer, only the peak discharge is required. The method is attractive because it does not require re-computation of the flow in upstream pipes.

InfoSewer gives you considerable flexibility to design a new sewer collection system or an existing sewer system.

EXISTING SYSTEM DESIGN

Increased sewer flows and/or aging of sewer pipes may call for the design existing sewer systems. InfoSewer designs existing pipes considering flow capacity constraints while minimizing cost. The model analyzes capacity of existing sewer systems and designs pipes that are transporting flow in excess of the desired capacity.

There are two criteria available for the design capacity calculations:

- Analysis criteria which are used to determine the capacity of existing pipes.
- Design criteria which are used to determine the size of new replacement pipes and new parallel pipes for those pipes observed to have flow depths above their analysis criteria.

InfoSewer/Pro will determine the optimal replacement and parallel (relief) pipe sizes where pipe capacity is exceeded (i.e., when existing pipes have depth-to-diameter ratios or flow depth-to-channel depth ratios exceeding user specified limits derived from the analysis criteria curve). The design cost for replacing and duplicating (paralleling) a pipe will also be calculated. A different Manning coefficient is then used when designing those pipes exceeding their analysis capacity. The replacement and the duplication channels will have the same shape as the existing channel (i.e., if an existing closed rectangular channel is found to be deficient, the replacement and/or the relief channel will also be a closed rectangular channel).

In addition, InfoSewer makes sure that flow velocity in the designed pipes meet a user-specified minimum (e.g., not to be less than 2 ft/s to prevent or minimize permanent deposition in the pipes) and maximum (e.g., not to exceed 10 ft/s to prevent the occurrence of scour or other undesirable effects of high velocity flow) design velocities.

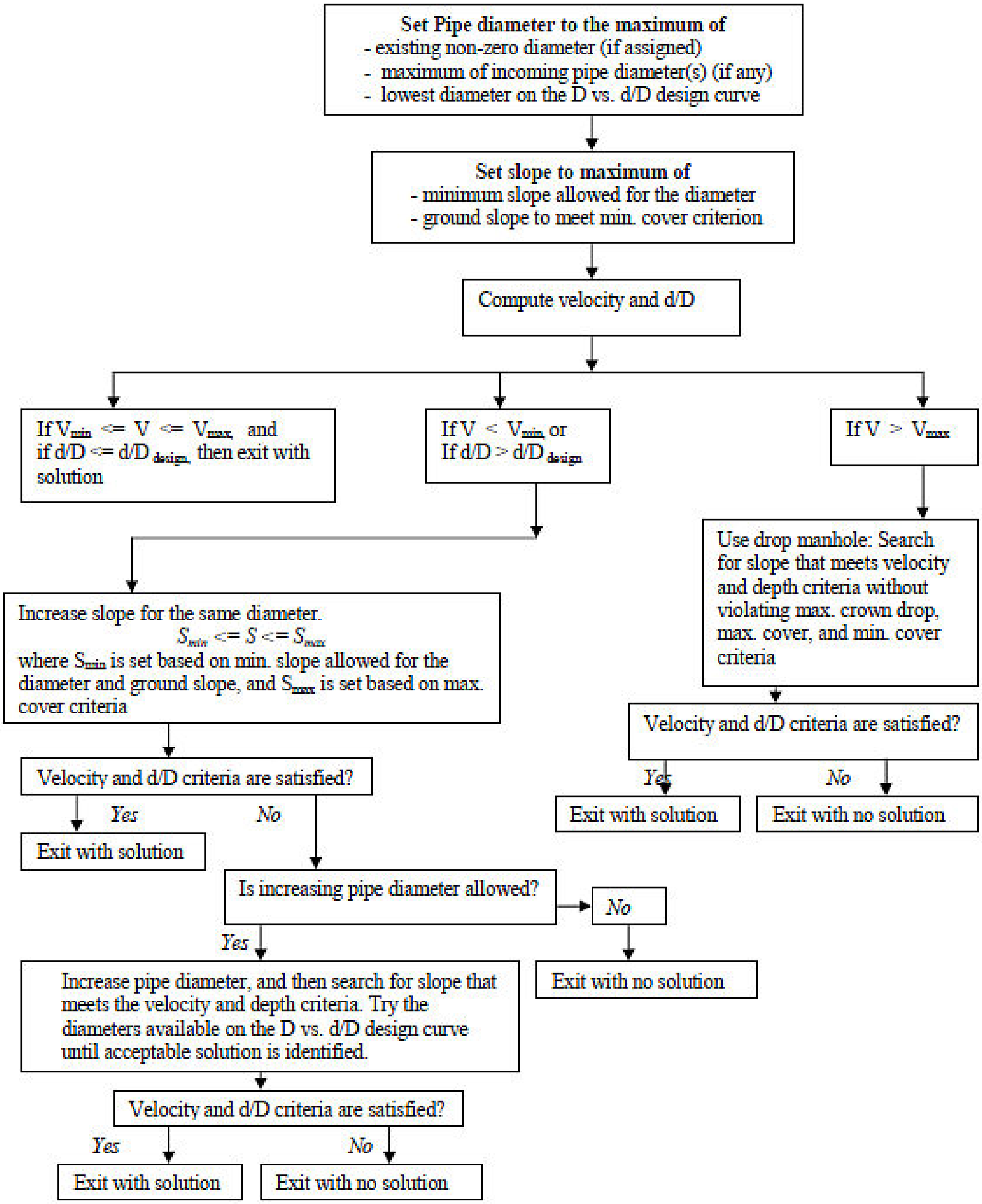
NEW SYSTEM DESIGN

InfoSewer can design a new sewer collection system given manhole locations, pipe lengths, sewer loads, topographic data of the sewershed, and design criteria. For new sewer design, the objective is to determine size and slope of a conduit that carries a given flow while maintaining velocity and cover depth within a desired range. The model calculates design flows, which may include both dry weather flows and wet weather flows, and determines pipe size and slope at minimum cost considering various design criteria including depth to diameter ratio, velocity, and cover depth.

The design is carried out for one pipe at a time and it progresses downwards along the flow direction. Trial pipe diameter is assigned to the pipe considering diameter of upstream pipes and a minimum diameter the pipe is allowed to take, and slope is selected considering ground slope, minimum cover required for the location, and a minimum slope allowed for the pipe size. Then flow velocity and flow depth are calculated for the pipe and are compared with the design criteria. If the pipe fails to meet one or more design criteria, pipe slope that satisfies all design criteria for the same pipe size would be searched for. If the pipe fails to meet all design criteria by changing slope alone, the model offers the option to change pipe size to the next large size defined. Drop manhole is used if pipe velocity exceeds the maximum allowed velocity.

A pipe is not allowed to take a diameter less than size of pipes on its upstream end. Crown elevation of a downstream pipe is set to crown elevation of upstream pipe unless the manhole is drop manhole. If more than one pipe is entering a manhole, the downstream pipe takes the lowest of the upstream crown elevations. Drop manholes are used if velocity exceeds the maximum allowed velocity, for example, due to steep terrains. An attempt is made to set cover depths to the minimum to reduce cost of excavation and reinstatement. The following flow chart briefly describes the design procedure.

Design results include pipe diameter, pipe slope, to invert elevation and from invert elevation of pipes, invert elevation of manholes, cover depth at upstream end and downstream ends of a pipe, and cost estimates. Design results could be directly applied to the model database. In addition, the model issues design status information for every pipe indicating whether the pipe is successfully designed or failed. Pipes located to the downstream of a failed pipe will not be designed. Design results could be easily reviewed using pipe profile plot, report, or color coding, for example based on, design status.



A more Step by Step Approach:

1. You can use EPS as the basis of your design **Step 1**

2. In the Design Tab for your existing System pick the Existing System Option **Step 2**

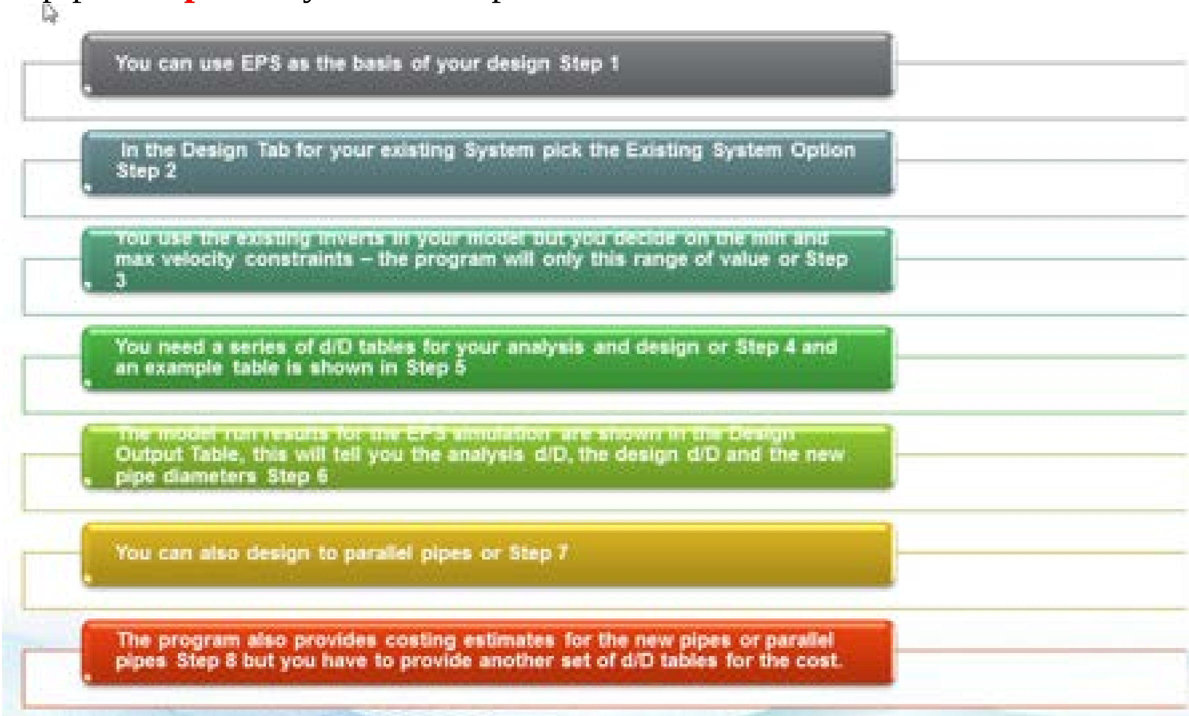
3. You use the existing inverts in your model but you decide on the min and max velocity constraints – the program will only this range of value or Step 3

4. You need a series of d/D tables for your analysis and design or **Step 4** and an example table is shown in Step 5

5. The model run results for the EPS simulation are shown in the Design Output Table, this will tell you the analysis d/D, the design d/D and the new pipe diameters Step 6

6. You can also design to parallel pipes or **Step 7**

7. The program also provides costing estimates for the new pipes or parallel pipes **Step 8** but you have to provide another set of d/D tables for the cost



EXISTING SYSTEM DESIGN

Increased sewer flows and/or aging of sewer pipes may call for the design existing sewer systems. InfoSewer designs existing pipes considering flow capacity constraints while minimizing cost. The model analyzes capacity of existing sewer systems and designs pipes that are transporting flow in excess of the desired capacity.

There are two criteria available for the design capacity calculations:

• Analysis criteria which are used to determine the capacity of existing pipes.

• Design criteria which are used to determine the size of new replacement pipes and new parallel pipes for those pipes observed to have flow depths above their analysis criteria.

InfoSewer will determine the optimal replacement and parallel (relief) pipe sizes where pipe capacity is exceeded (i.e., when existing pipes have depth-to-diameter ratios or flow depth-to-channel depth ratios exceeding user specified limits derived from the analysis criteria curve). The design cost for replacing and duplicating (paralleling) a pipe will also be calculated. A different Manning coefficient is then used when designing those pipes exceeding their analysis capacity. The replacement and the duplication channels will have the same shape as the existing channel (i.e., if an existing closed rectangular channel is found to be deficient, the replacement and/or the relief channel will also be a closed rectangular channel).

In addition, InfoSewer makes sure that flow velocity in the designed pipes meet a user-specified minimum (e.g., not to be less than 2 ft/s to prevent or minimize permanent deposition in the pipes) and maximum (e.g., not to exceed 10 ft/s to prevent the occurrence of scour or other undesirable effects of high velocity flow) design velocities.

A more Step by Step Approach:

1. You can use EPS as the basis of your design Step 1

2. In the Design Tab for your existing System pick the Existing System Option Step 2

3. You use the existing inverts in your model but you decide on the min and max velocity constraints – the program will only this range of value or Step 3

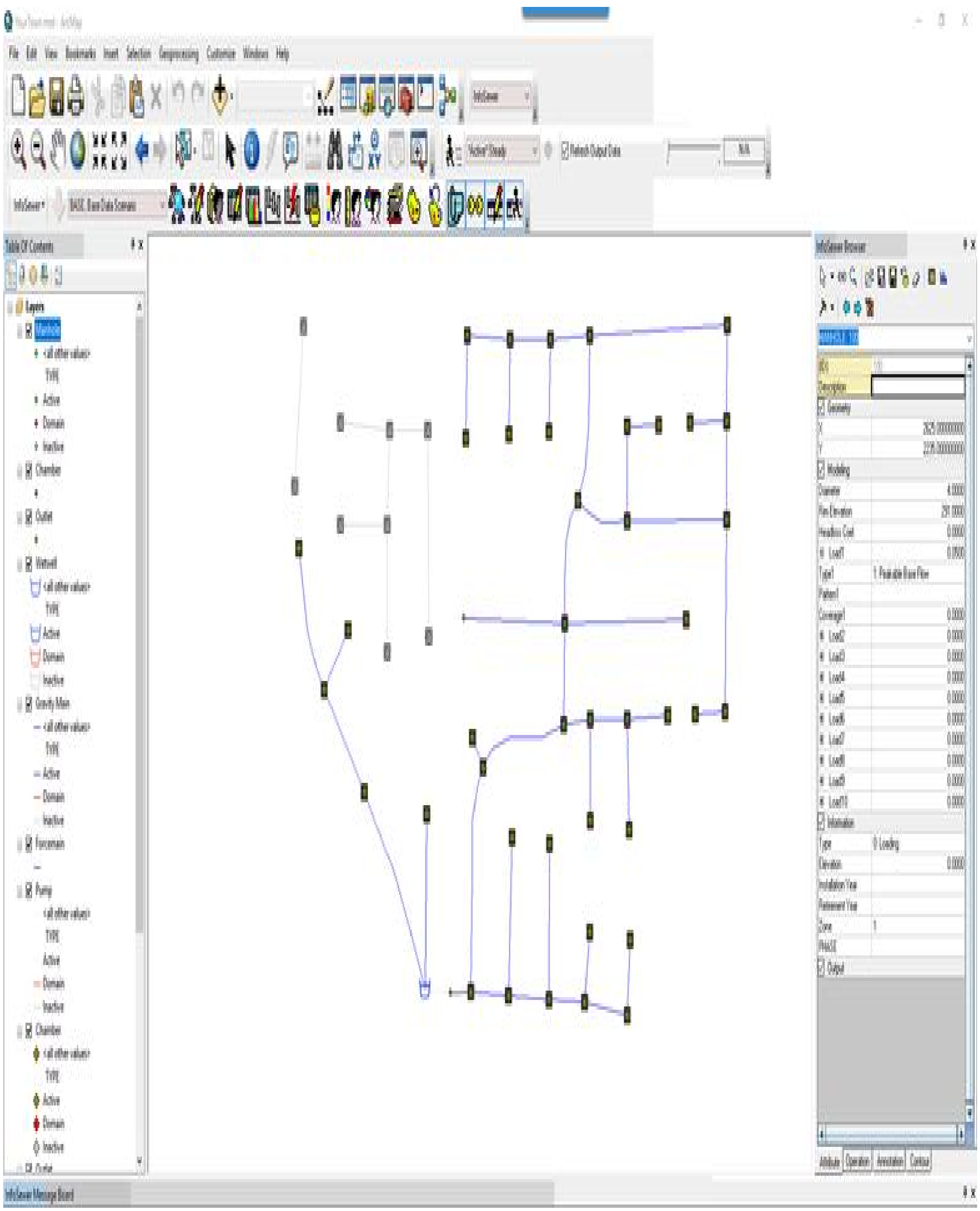
4. You need a series of d/D tables for your analysis and design or Step 4 and an example table is shown in Step 5

5. The model run results for the EPS simulation are shown in the Design Output Table, this will tell you the analysis d/D, the design d/D and the new pipe diameters Step 6

6. You can also design to parallel pipes or Step 7

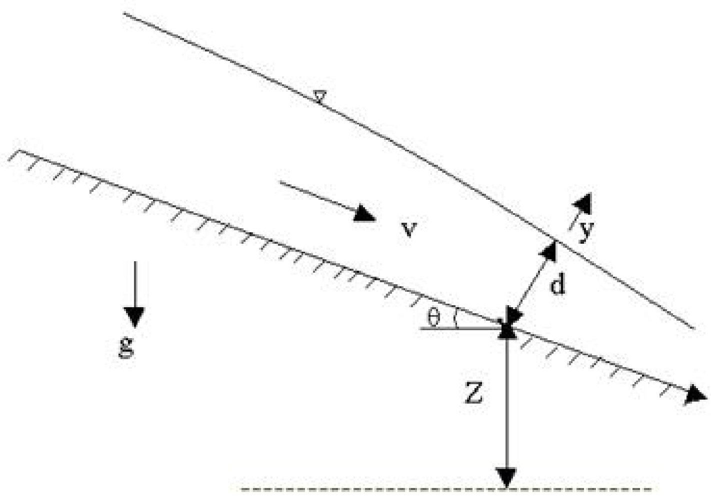
7. The program also provides costing estimates for the new pipes or parallel pipes Step 8 but you have to provide another set of d/D tables for the cost

8. Would it help if I sent you a small example model?



EXTENDED PERIOD DYNAMIC Simulation (UNSTEADY FLOW)

InfoSewer tracks the movement of wastewater flowing through the network over an extended period of time under varying wastewater loading and operating conditions. The extended period simulation (EPS) model implemented in InfoSewer is unsteady model and is predicated on solving a simplified form of the 1D Saint-Venant equations neglecting local acceleration.



The Saint-Venant equations or full dynamic wave equations for open channel flow routing consist of the conservation of momentum equation and the equation of continuity. The momentum equation is:

$$\frac{1}{gA}\cdot\frac{\partial Q}{\partial t}+\frac{1}{gA}\cdot\frac{\partial}{\partial x}\left(\frac{\beta Q^2}{A}\right)+\ cos\emptyset\frac{\partial d}{\partial x}-S_o+S_f=0$$

The continuity (mass conservation) equation is:

$$\frac{\partial A}{\partial t}+\frac{\partial Q}{\partial x}=0$$

where

x = distance along the pipe (longitudinal direction of sewer)

A = flow cross sectional area normal to x

y = coordinate direction normal to x on a vertical plane

d = depth of flow of the cross section, measured along y direction

Q = discharge through A

V = cross sectional average velocity along x direction

S_0 = pipe slope, equal to sin θ

θ = angle between sewer bottom and horizontal plane

S_f = friction slope

g = gravitational acceleration

t = time

β = Boussinesq momentum flux correction coefficient for velocity distribution

These complete unsteady flow equations (momentum together with continuity) along with appropriate initial and boundary conditions are rather tedious and computationally expensive to solve, especially for large sewer collections systems. As a result, acceptable simplifications and improved solution methods have been proposed including *non-inertial*, *kinematic wave* and *dynamic wave* simplifications. Hydraulically, the dynamic wave approach is the most accurate model among the approximations. The Muskingum-Cunge explicit diffusion wave dynamic flow routing model, obtained by neglecting local acceleration term in the momentum equation, is the most commonly used dynamic wave model.

In InfoSewer, unsteady open channel (free surface) flow is simulated using Muskingum-Cunge technique whereas pressurized flow in any pipe is modeled assuming the pipe is flowing full and the energy equation is applied to the entire pipe section.

Muskingum-Cunge:

$$\frac{\partial Q}{\partial t} + c \cdot \frac{\partial Q}{\partial x} + c\frac{\partial}{\partial t}\left(\frac{\alpha}{c^2}\frac{\partial Q}{\partial x}\right) = 0$$

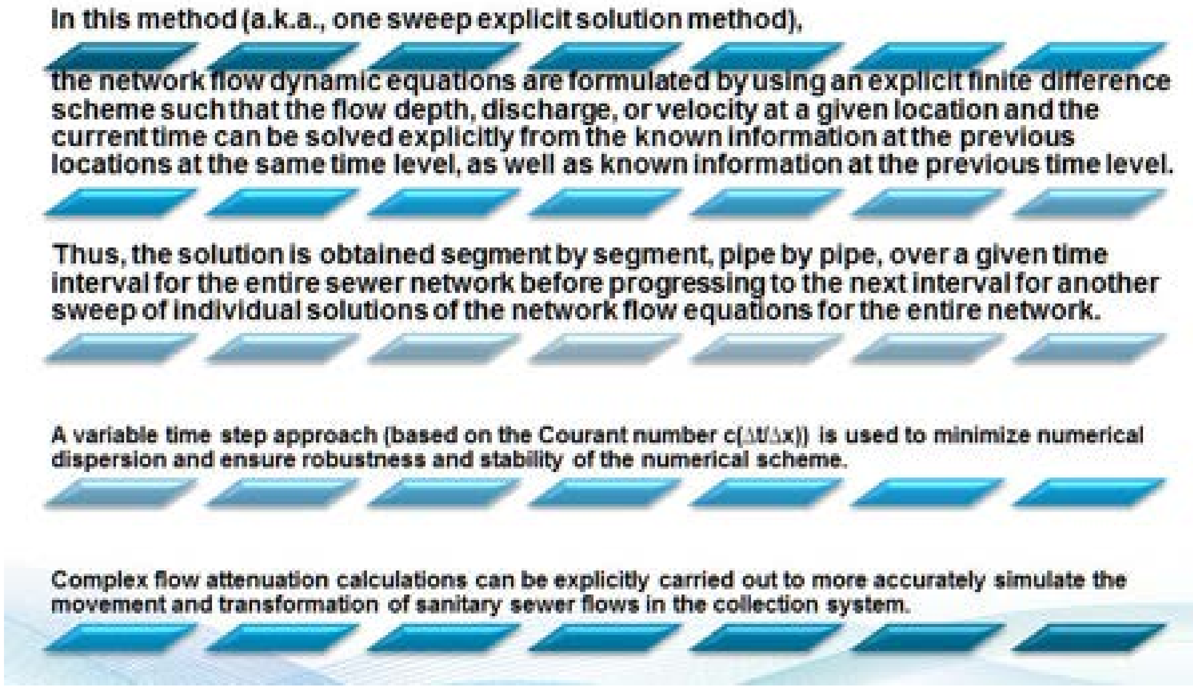
$$c = \frac{\partial Q}{\partial A}$$

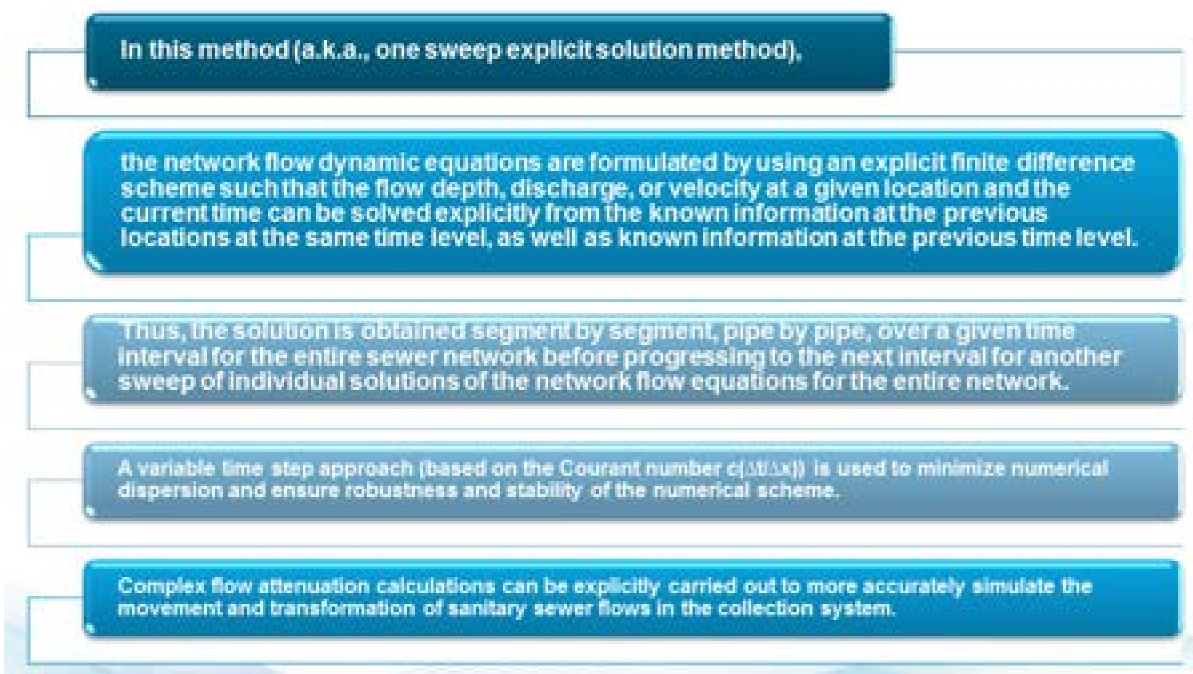
$$\alpha = \frac{Q}{2BS_0}[1 - \frac{BQ^2}{gA^3}[(1 - \frac{Ac}{Q})\ ]$$

Here c is the dynamic wave celerity and B is the top width at normal depth for discharge Q. This highly efficient and accurate flow routing algorithm is used by InfoSewer to track the spatial and temporal variation of flows throughout the collection system.

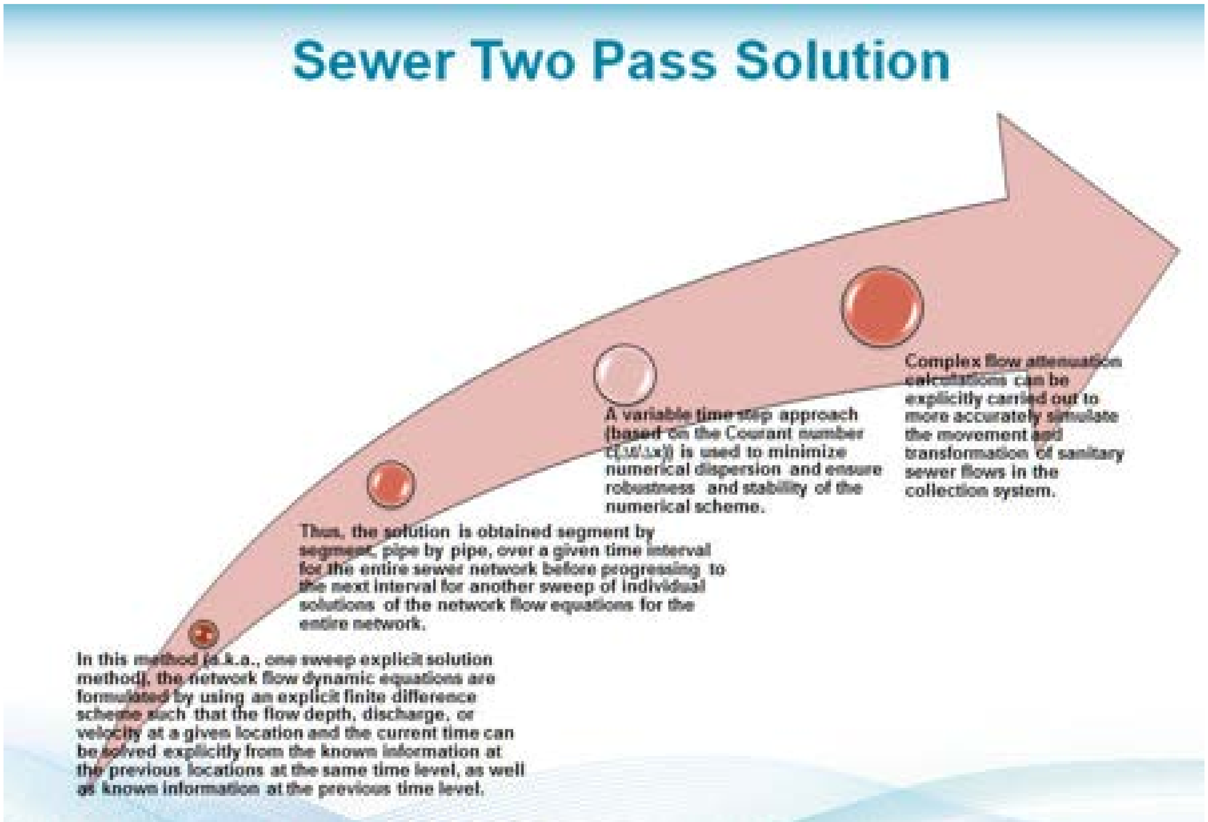
In this method (a.k.a., one sweep explicit solution method), the network flow dynamic equations are formulated by using an explicit finite difference scheme such that the flow depth, discharge, or velocity at a given location and the current time can be solved explicitly from the known information at the previous locations at the same time level, as well as known information at the previous time level. Thus, the solution is obtained segment by segment, pipe by pipe, over a given time interval for the entire sewer network before progressing to the next interval for another sweep of individual solutions of the network flow equations for the entire network. A variable time step approach (based on the Courant number $c(\Delta t/\Delta x)$) is used to minimize numerical dispersion and ensure robustness and stability of the numerical scheme. Complex flow attenuation calculations can be explicitly carried out to more accurately simulate the movement and transformation of sanitary sewer flows in the collection system.

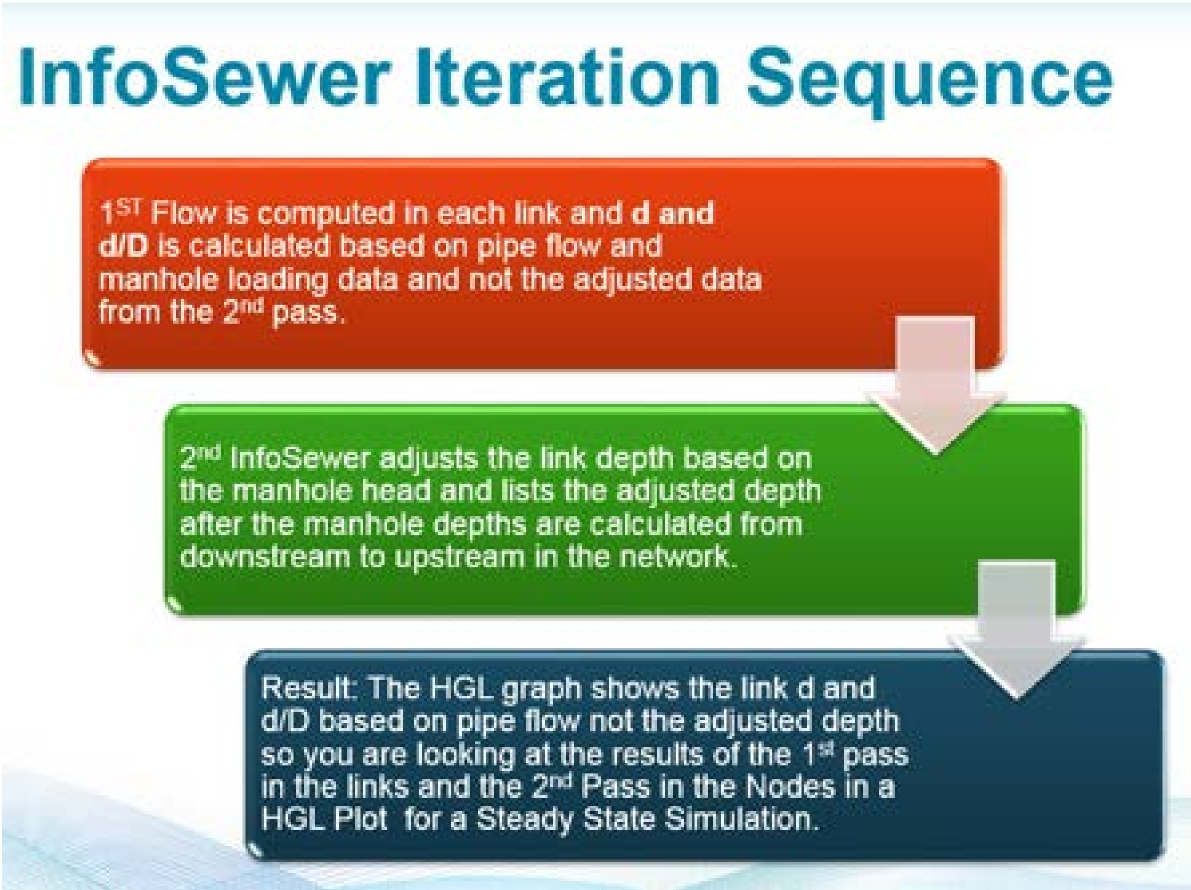
$$c\ \frac{\Delta x}{\Delta t} \leq 1$$



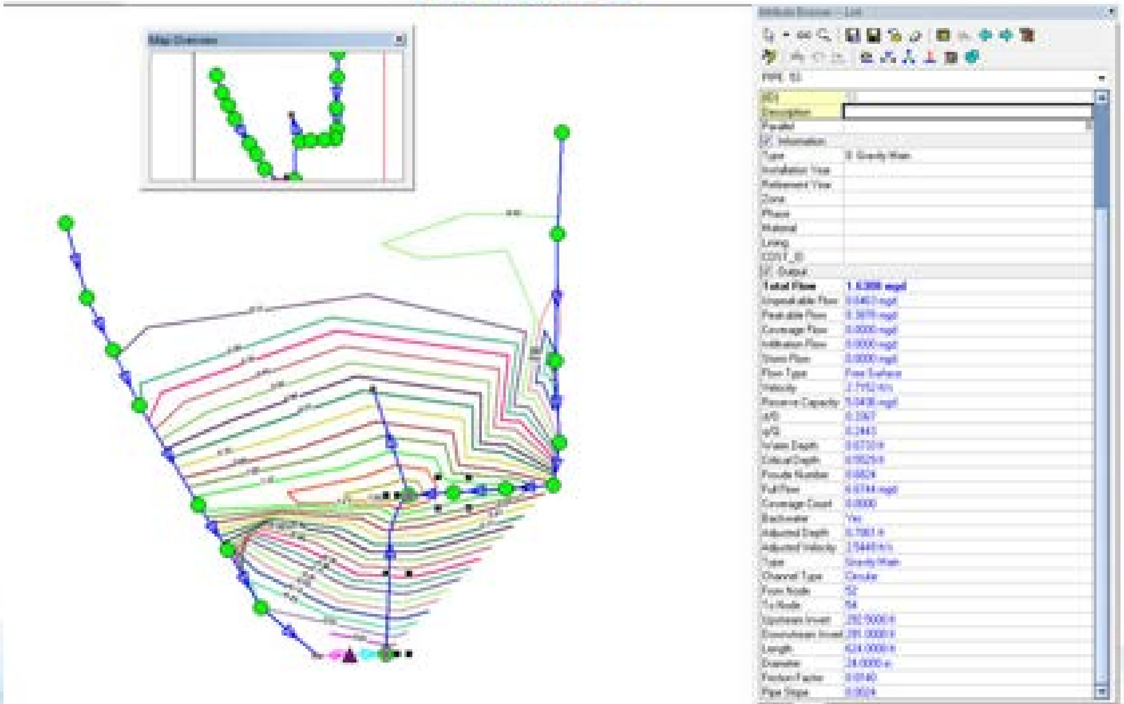


An excellent review and comparison between simulated and observed hydrographs of the various numerical methods for solving unsteady flow in simple and compound channels was presented by Chatila (Chatila 2003). In terms of overall performance, the Muskingum unsteady solution scheme compared favorably and proved to be a simple and reliable method avoiding complicated mathematical and numerical computations for the cases considered.

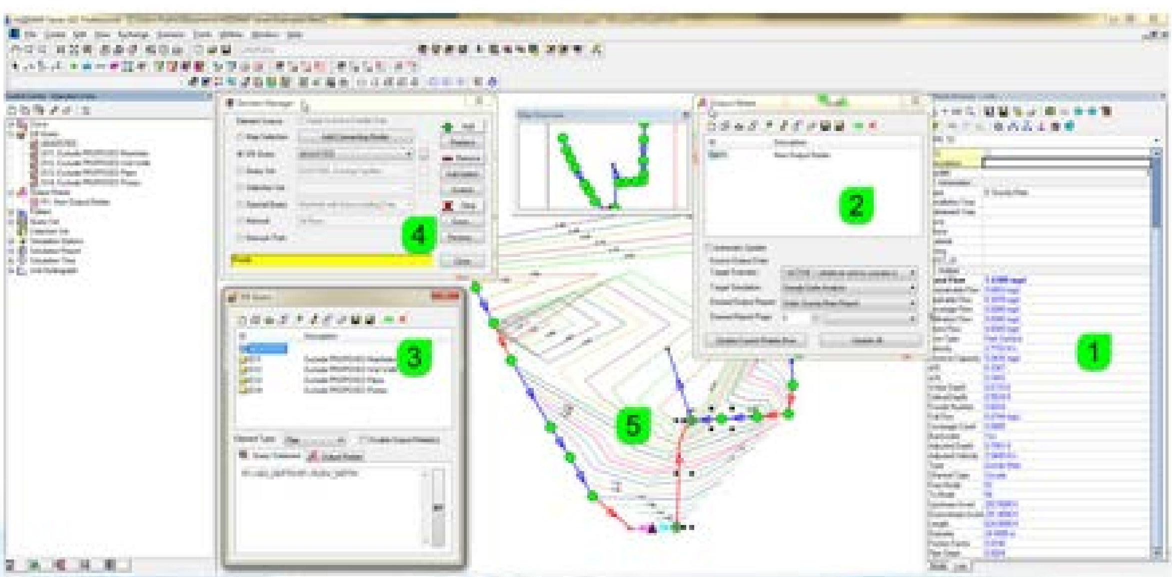



InfoSewer Iteration Sequence
1ST Flow is computed in each link and d and d/D is calculated based on pipe flow and manhole loading data and not the adjusted data from the 2nd pass.
2nd InfoSewer adjusts the link depth based on the manhole head and lists the adjusted depth after the manhole depths are calculated from downstream to upstream in the network.
Result: The HGL graph shows the link d and d/D based on pipe flow not the adjusted depth so you are looking at the results of the 1st pass in the links and the 2nd Pass in the Nodes in a HGL Plot for a Steady State Simulation.

Adjusted Depth and Velocity are 2nd Pass Variables



Adjusted Depth and Velocity are 2nd Pass Variables in the Steady State and EPS Solutions



Flooding at manholes and wet-wells in InfoSewer is not modeled during an extended period dynamic simulation. Instead, the flows at the flooded structures are conserved and are not lost by the occurrence of flooding at the manholes. In actual flooding situations, flows may be diverted away from the flooded structures and out of the sewer collection system. However, a surcharged pipe or manhole is generally an indication of poor hydraulic performance of the sewer system. InfoSewer i assumes that the downstream pipes of flooded manholes are flowing full.

Sanitary sewer systems are typically designed to flow less than full and have an upper pressure limit of 4 to 6 psi. Sewer systems operating under pressurized flow condition may run the risk of violating local, state, and federal health codes. The USEPA regulations would also be in violation if raw sewage were discharged into the ground, potentially affecting groundwater. For these reasons, pressurized flows in sanitary sewers not designed to sustain pressures can be dangerous and in some cases can present an unlawful activity.

SURCHARGE

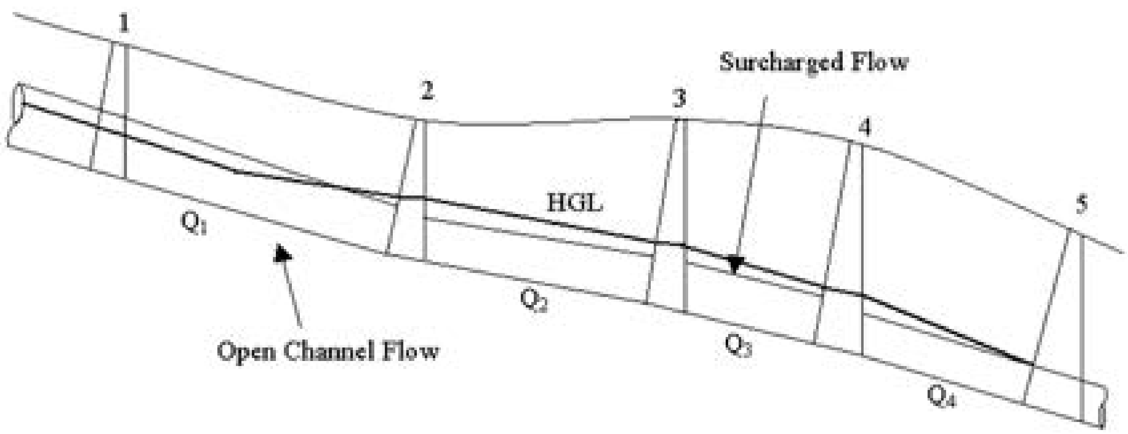
Sewer pipes can flow full with water under pressure, which is often known as surcharge flow. Surcharge flow occurs in under-designed pipes (or under extreme flows) when the flow rate Q exceeds the full pipe capacity Q_f .

Flow conditions are unstable at the transition between open-channel (free surface) flow and full pipe flow. A wave or surge can induce full flow in the pipe in the unstable range. Surcharge in sewer pipes is modeled in InfoSewer using energy and continuity principles. The energy equation between sections 1 and 2 in a pipe can be written as:

$$z_1 + d_1 + \frac{V_1^2}{2g} - H_L - z_2 - d_2 - \frac{V_2^2}{2g} = 0$$

Here z denotes the invert elevation; d represents the water depth; and HL designates the head loss between sections 1 and 2. The energy equation is used to determine the difference in hydraulic grade line elevation (which is added at the upstream manholes) needed to pass downstream flows under the surcharge condition.

The procedure for analyzing surcharge in sewer pipes is illustrated using the figure below as a reference.



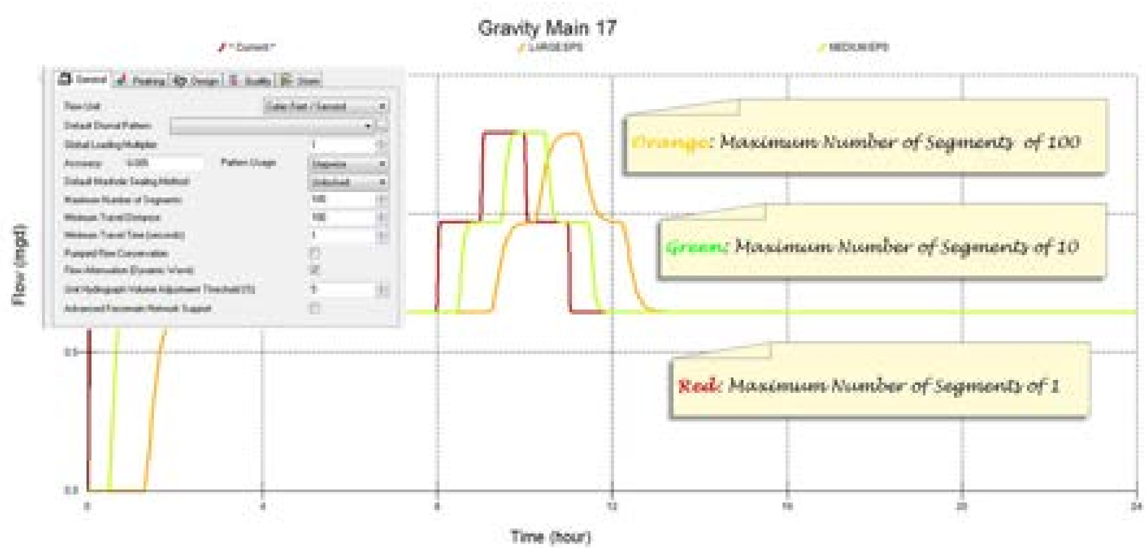
Assuming that pipe 4 (between manholes 4 and 5) is under-designed, Q4 will exceed its full flow capacity and the hydraulic grade line at manhole 4 will increase based on energy consideration to allow Q4to pass through pipe 4 (note that water always flows from higher to lower energy) as continuity must be satisfied. This forces the hydraulic grade line at manhole 3 to increase in order for Q3to pass through pipe 3. The procedure continues upstream until the slope of the energy grade line needed to transport the flow allows open-channel flow condition to occur in the pipe. The projected hydraulic grade line will then intersect the uniform water surface flow to complete the backwater curve.

The energy equation is also used to model the flow in siphons, which can occur in adverse pipes. InfoSewer assumes that the siphon flows full, with a continuous liquid column throughout it.

FLOW ATTENUATION

When a flow hydrograph is injected and propagates downstream in sewer pipes the bulk of the water will normally travel slower than its induced disturbance or wave. That is, if the water is injected with a tracer then the tracer lags behind the disturbance. The speed of the disturbance depends on parameters such as depth, width and flow velocity. This disturbance will tend to flatten, or spread out, the peak flow in the downstream direction along the sewer pipes.

Flow attenuation in a sewer system is defined as the process of reducing the peak flow rate by redistributing the same volume of flow over a longer period of time as a result of friction (resistance), internal storage and diffusion along the sewer pipes. InfoSewer uses the distributed Muskingum-Cunge flow routing method based on diffusion analogy, which is capable of accurately predicting hydrograph attenuation or peak flow damping effects (peak subsidence). The method is attractive since the routing parameters can be directly calculated as a function of pipe and flow properties, is applicable for a wide range of flow conditions, and does not require calibration or any iterative scheme. The Muskingum coefficients are derived from the pipe diameter, length, discharge, dynamic wave celerity, and slope of the flow. The magnitude of attenuation depends on parameters such as the peak discharge, the curvature of the hydrograph, and the width of flow. An example of flow attenuation process as a hydrograph is routed through a sewer system is illustrated in the figure below.



HYDROGRAPH AGGREGATION/FLOW ACCUMULATION

Proper aggregation of multiple hydrographs with distinct time steps is essential in a sewer collection system as the flows are routed in both time and space.

Aggregation normally occurs when laterals are merging around manholes and wet-wells. This can create offset of time-steps, which can affect accurate determination of flow peaks and volumes. InfoSewer utilizes a highly accurate dynamic hydrograph aggregation method that allows preservation of both flow peaks and flow volumes when multiple hydrographs with different time steps are added. The method is Lagrangian in nature and tracks the hydrograph ordinates as they are transported along the sewer pipes and mix together at manholes and wet-wells. A variable time step is used to minimize numerical dispersion, enhance stability, and maximize computational efficiency. See InfoSewer, H20Map Sewer EPS Mass Balance for how to interpret the Mass Balance Check and the options to lower the overall continuity error.

InfoSewer Water Quality Options

Introduction: InfoSewer is used worldwide by municipal engineers and planners to create detailed, accurate models of their sewer infrastructure systems. These models enable users to evaluate the effect of new developments, zoning changes, and other additional loads on system flows; pinpoint current and future problem areas; predict overflows and backups; and determine how to best restore needed capacity lost to infiltration and inflow with the least rehabilitation. In addition, users rely on these models to compute hydrogen sulfide generation and corrosion potential; analyze the rate of Biochemical Oxygen Demand (BOD) exertion; track sediment movement and deposition; calculate the amount of pollutant transported to the wastewater treatment plant; and assess pollutants' impacts on receiving waters. Extensive scenario management along with domain and facility management functionality makes the program capable of analyzing existing or proposed sewage collection systems. It has a Steady State, Design and EPS run modes and includes many of the same Arc Map tools that exist in InfoWater and InfoSWMM.

You can model 8 options in InfoSewer to simulate various aspects of Water Quality (Figure 1). If you make the base scenario no water quality you can have the same network, same loading but different aspects of water quality in seven child scenarios' (Figure 2). The parameters for each water quality option are shown in the Quality Tab of the Simulation Options Dialog.

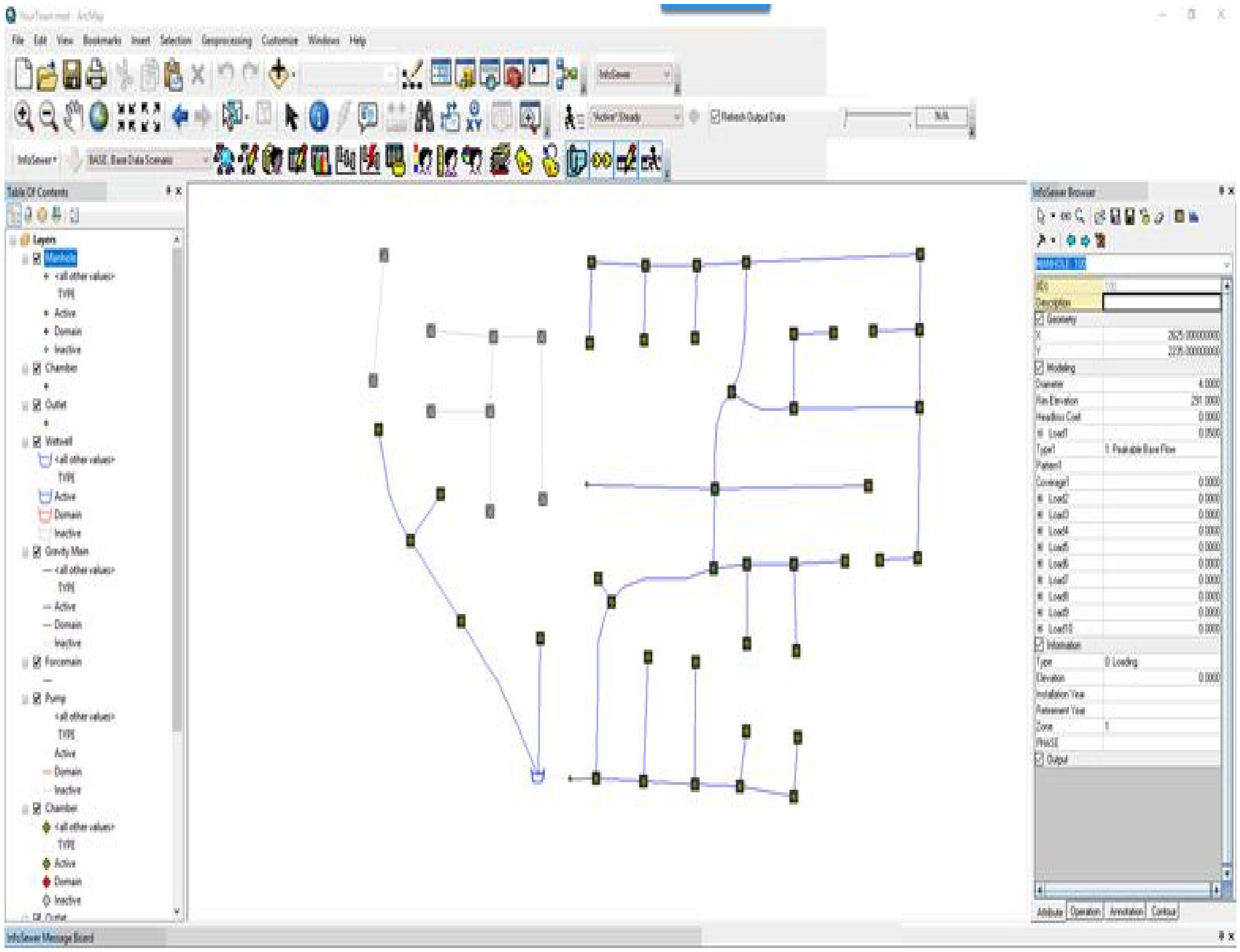


Figure 1. Water Quality Simulation Choices in InfoSewer Pro.

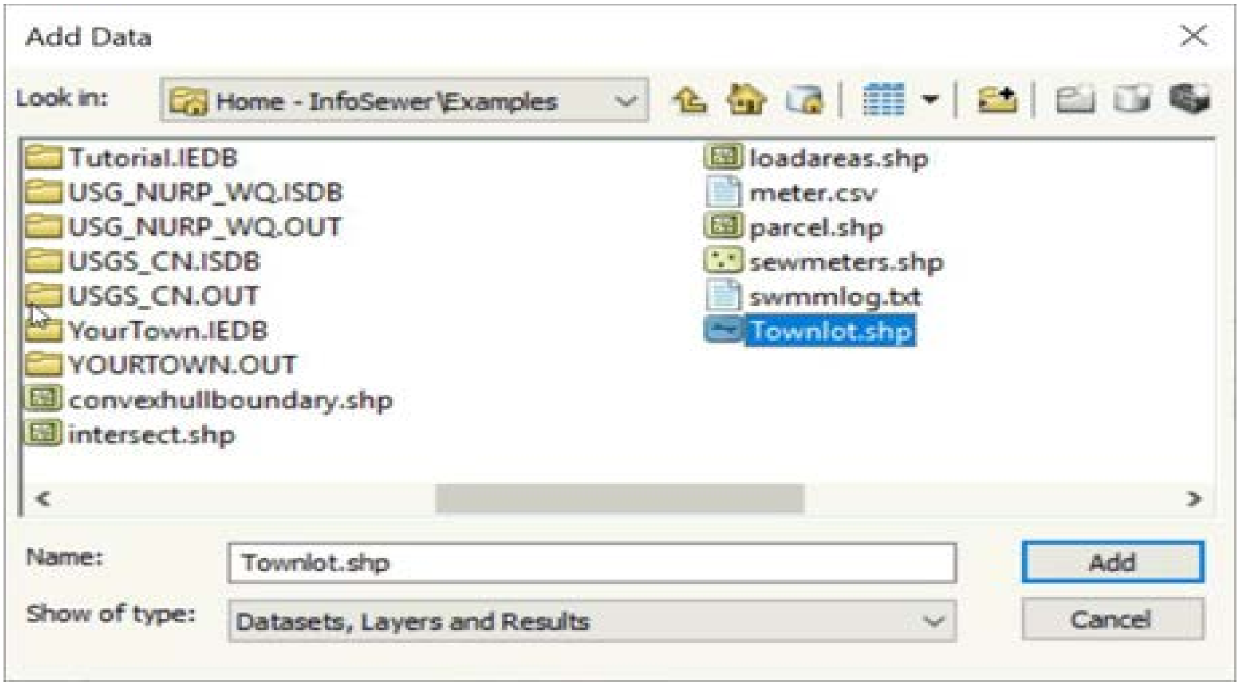


Figure 2. Water Quality Simulation Choices in the Scenario Explorer of InfoSewer

Biochemical Oxygen Demand

Biochemical Oxygen Demand (BOD) is the most widely used parameter of organic pollution in sanitary sewer systems. The ability to model BOD is of great importance in wastewater engineering to assist wastewater utilities in (1) estimating the quantity of oxygen required to biologically stabilize the organic matter present; (2) determining the size of wastewater treatment facilities; (3) evaluating the efficiency of the treatment process; and (4) ensuring compliance with wastewater discharge permits (Tchobanoglous 2003).

InfoSewer models the rate of BOD oxidation (exertion) throughout the collection system using first-order kinetics with the rate of oxygen utilization being proportional to the difference between the amount of oxygen used and the ultimate BOD as:

$$\frac{dBOD}{dt} = -kBOD$$

or

$$BOD = UBOD(1 - e^{-k\,t})$$

where

BOD = BOD exerted at time t, mg/L

UBOD = total or ultimate carbonaceous BOD, mg/L

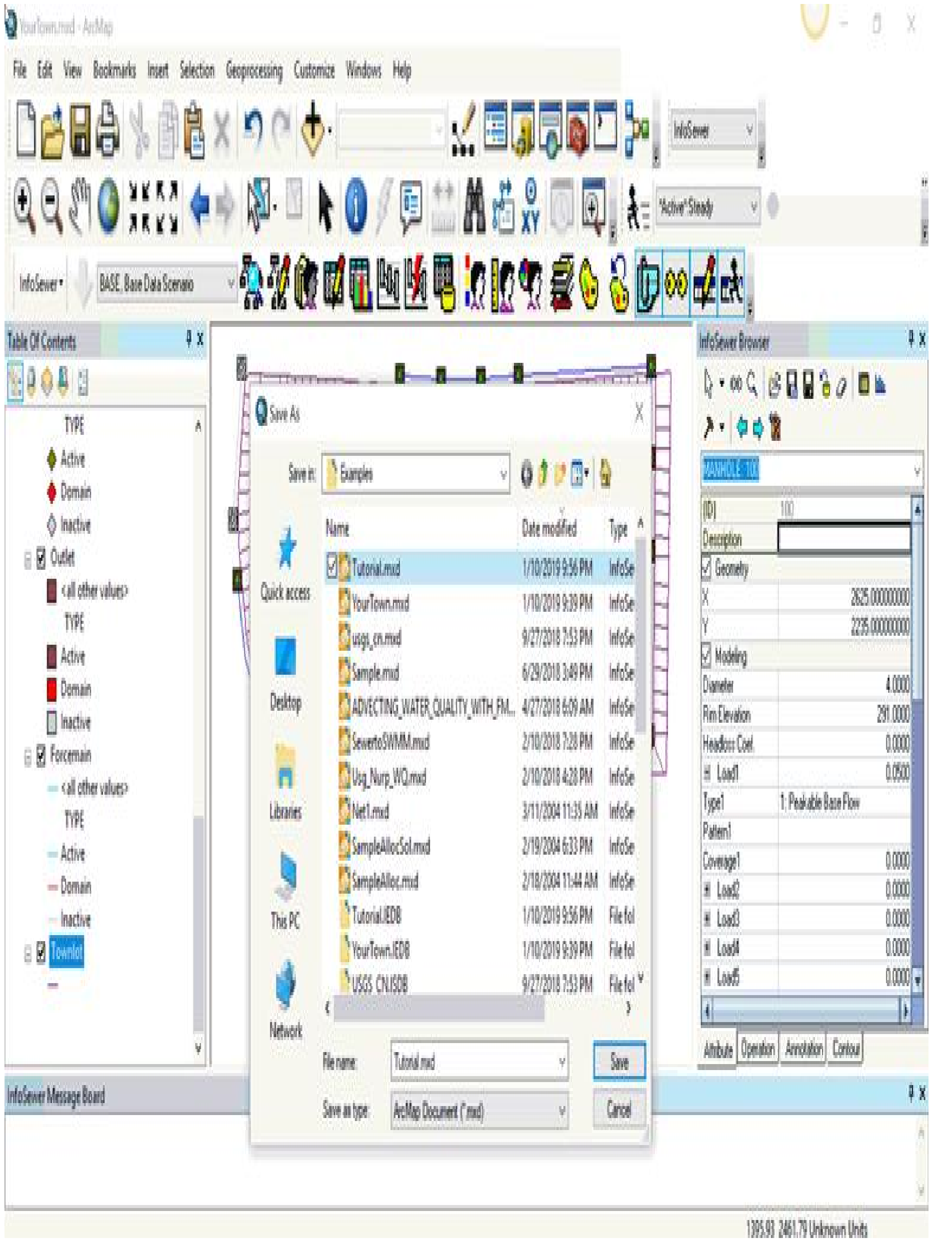
k = first-order reaction rate constant, 1/day

t = time, day

The value of the first-order reaction rate constant k varies with the type of waste and normally ranges from 0.05/day to 0.46/day or more, with a typical value of 0.23/day. This constant can also be expressed as a function of temperature as:

$$k = k_{20}\theta^{T-20}$$

where k_{20} is the first-order reaction rate constant at 20^{o}C (1/day) and T is the temperature (degree Celsius). The value of θ is 1.056 in the temperature range between 20^{o}C and 30^{o}C and 1.135 in the temperature range between 4^{o}C and 20^{o}C, with a typical value of 1.047.InfoSewer determines and uses appropriate values of k and θ based on the user-specified temperature. The default values used by InfoSewer for k, θ, and T are 0.23/day, 1.047, and 20^{o}C, respectively.



YourTown.mxd - ArcMap
File Edit View Bookmarks Insert Selection Geoprocessing Customize Windows Help
InfoSewer
BASE, Base Data Scenario
Table Of Contents
Save As
Save in: Examples
Name
Date modified
Type
Tutorial.mxd
YourTown.mxd
usgs_cn.mxd
Sample.mxd
SewertoSWMM.mxd
Net1.mxd
SampleAllocSol.mxd
SampleAlloc.mxd
Tutorial.IEDB
YourTown.IEDB
Quick access
Desktop
Libraries
This PC
Network
File name: Tutorial.mxd
Save as type: ArcMap Document (*.mxd)
Save
Cancel
InfoSewer Browser
MANHOLE: 100
Description
Geometry
Modeling
Diameter
Rim Elevation
Headloss Coef
Load1
Type1
1: Peakable Base Flow
Pattern1
Coverage1
Load2
Load3
Load4
Load5
Attribute
Operation
Annotation
Contour
InfoSewer Message Board
1395.93 2461.79 Unknown Units

Pollutant Transport

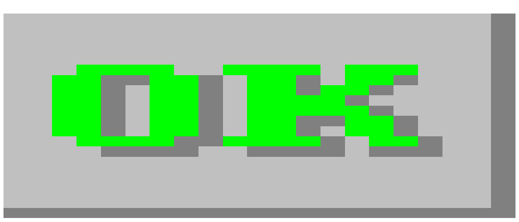
InfoSewer can effectively simulate the transport of dissolved pollutants throughout the sewer collection system. It tracks the movement of conservative constituents (e.g., chloride, bromide, sulfate, boron, sorbed trace metals) flowing through the network over time. The dynamic quality simulation model is predicated on conservation of mass coupled with reaction kinetics and consists essentially of three processes: advection in pipes, mixing at sewer manholes and wet-wells, and kinetic reaction mechanism. Longitudinal dispersion due to concentration gradient is neglected which means that there is no intermixing of mass between adjacent sewage parcels traveling down a pipe. Advective transport within a sewer pipe is represented with the following equation:

$$\frac{\partial C_i}{\partial t} = -u_i \frac{\partial C_i}{\partial x} + r(C_i)$$

where Ci is the concentration (mass/volume) in pipe i as a function of distance x and time t; ui is the flow velocity (length/time) in pipe I; and r is the rate of reaction (mass/volume/time) as a function of concentration.

For conservative (inert) pollutants the rate of reaction (r) is set to zero. For pumps, instantaneous substance advection is assumed.

At sewer manholes and wet-wells, the mixing of fluid is taken to be complete and instantaneous. Thus the concentration of a substance in sewage leaving the manhole or wet-well is simply the flow-weighted sum of the concentrations from the incoming pipe(s) and is described by the following equation:



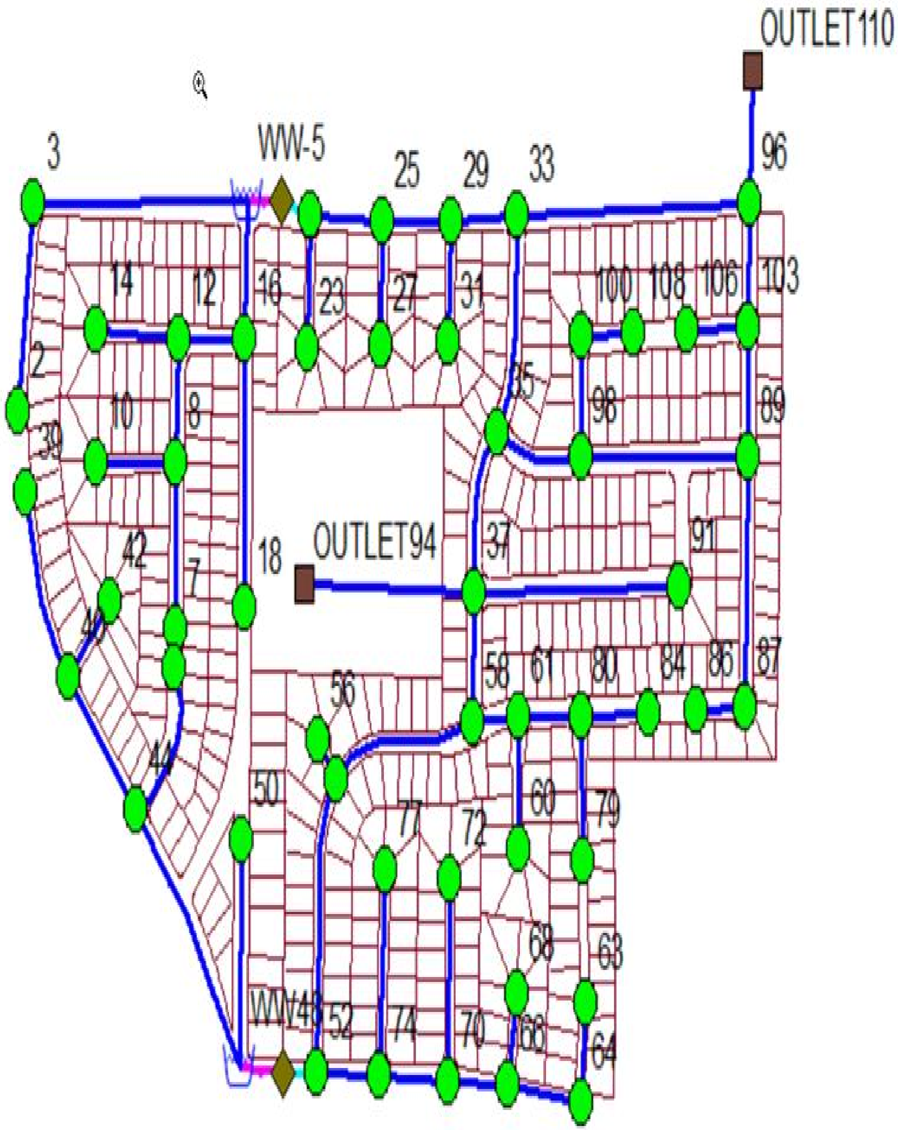
where Qin is the incoming flow (volume/time); Cin is the pollutant concentration of the incoming flow; and Cout is the concentration of the pollutant leaving the manhole.

Under completely mixed conditions, the concentration throughout the wet-well is a blend of the current contents and that of any entering sewage and is represented by the following equation:

$$\frac{\partial(V_{ww} C_{ww})}{\partial t} = \sum Q_{in} C_{in} - \sum Q_{out} C_{ww}$$

where V_{WW} is the wet-well volume at time t, Cww is the wet-well concentration; and the remaining terms are as defined above.

The ability to model pollutant transport in sewer collection systems is useful in determining the load of pollutants that is transported to the wastewater treatment plant and assessing impact on the receiving waters.

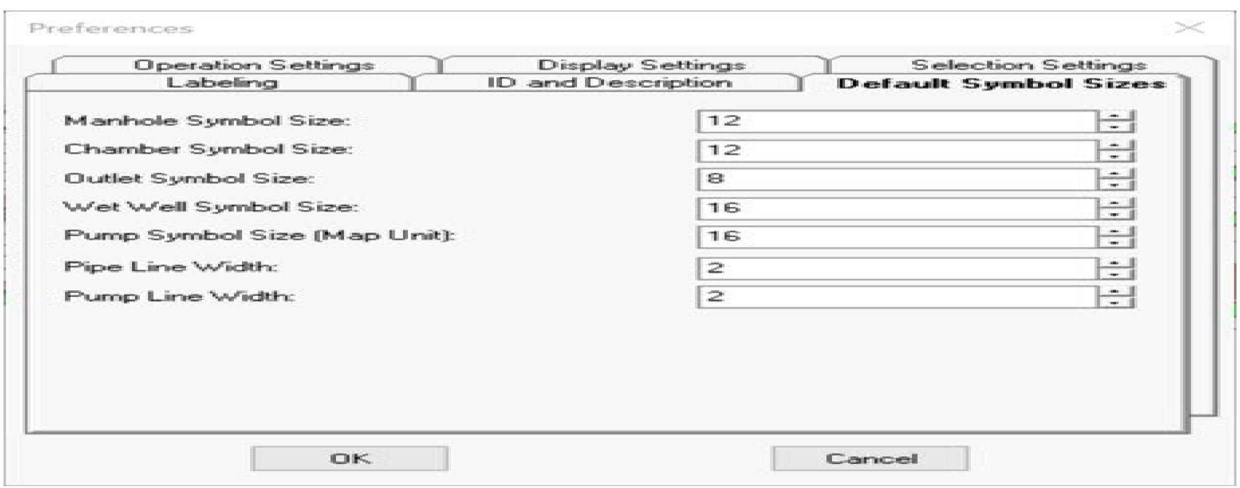


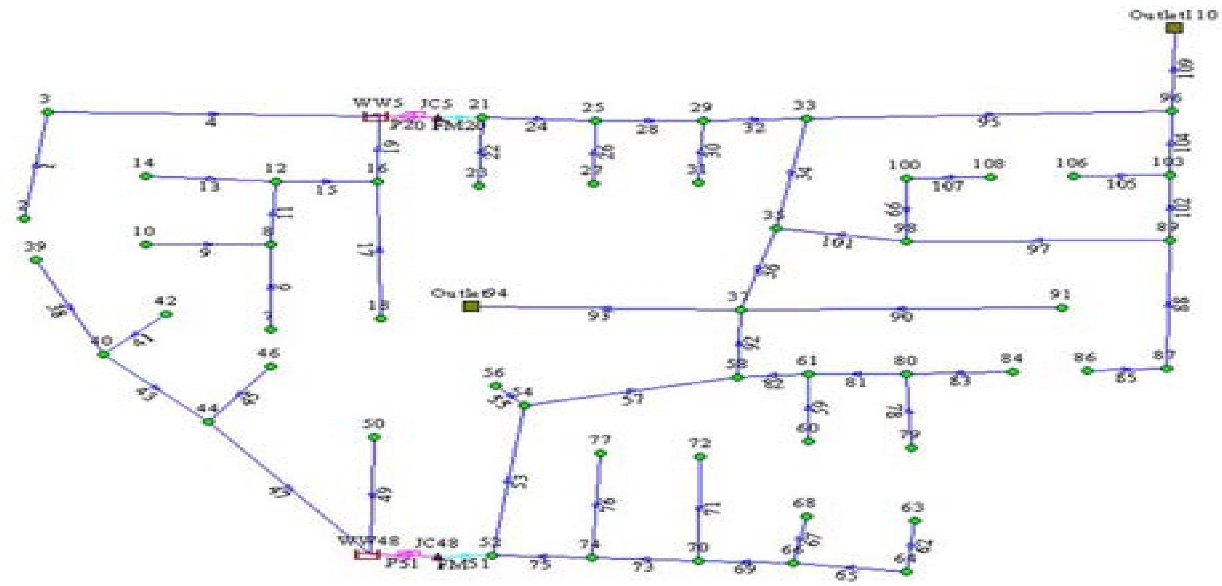
OUTLET110
WW-5
3
25
29
33
96
14
12
16
23
27
31
100
108
106
103
2
10
8
35
98
89
39
42
7
18
OUTLET94
37
91
40
58
61
80
84
86
87
56
44
50
60
79
77
72
68
63
WW48
52
74
70
66
64

Time of Concentration or Water Age

InfoSewer can model the changes in the age of sewage flow (time of concentration) throughout a collection system. Time of concentration is the time spent by a sewage flow parcel in the network (i.e., the time of flow in the sewerage system). This parameter is useful to address important water quality and safety issues such as generation of sulfide that may occur in a sanitary sewer system (which manifest itself in corrosion and odor issues).

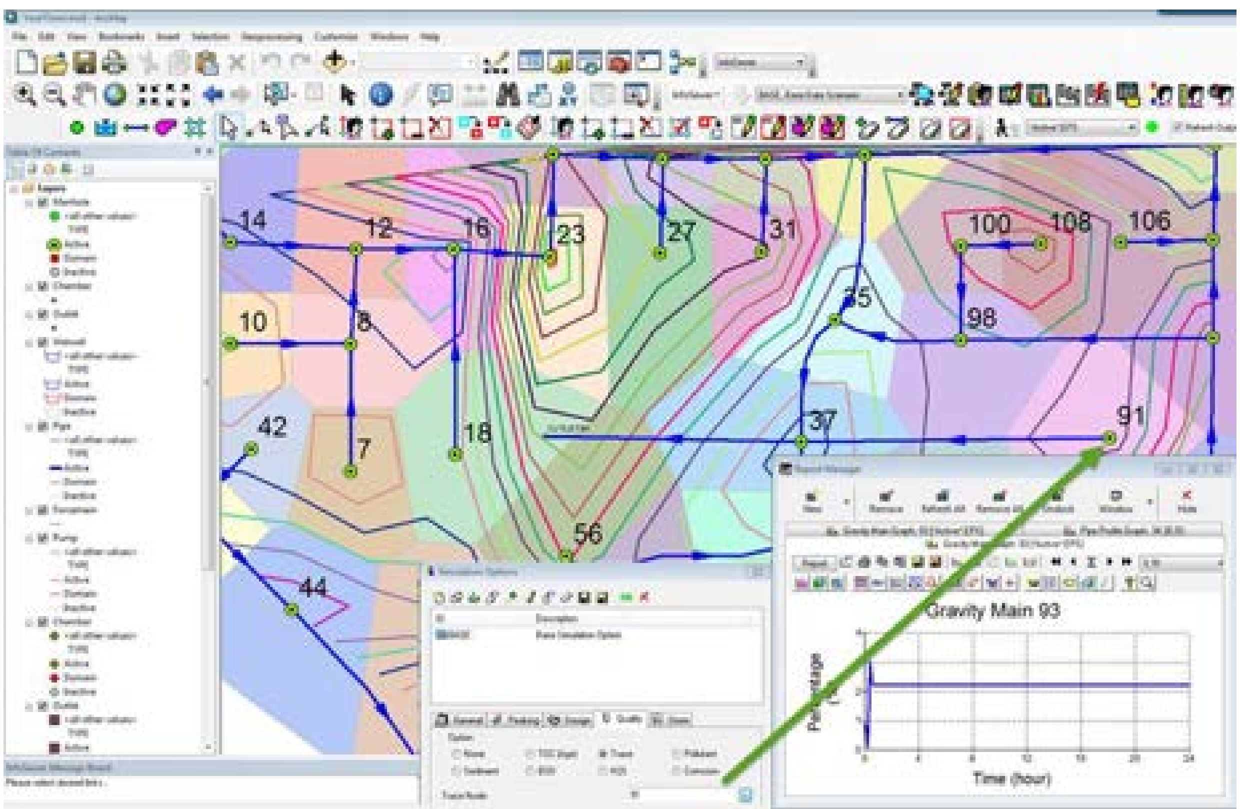
In InfoSewer Pro, new sewage entering the network from loading manholes enters with age of zero. As this sewage moves through the collection system it splits apart and blends together with sewage flow parcels of varying age at manholes and wet-wells. InfoSewer provides automatic modeling of sewage age. Internally, it treats age as a reactive constituent whose growth follows zero-order kinetics with a rate constant (r =1.0) equal to 1 (i.e., each second the sewage becomes a second older). Time of concentration for a manhole is thus calculated as a flow-weighted average sewage age value of flows entering the manhole. Travel time of a sewage flow parcel through a sewer pipe is computed based on flow velocity and pipe length. For the pipes leaving a manhole, the average sewage age is increased by the travel time to the next downstream manhole.





Source Tracing

InfoSewer can also perform sophisticated source tracing calculations. Source tracing tracks over time what percent of sewage reaching any pipe or manhole in the network had its origin at a particular source node. The source node can be any manhole in the network, including wet-wells. Source tracing is very useful in sewer collection systems, and could be used for (1) tracking changes in sewage flow contribution (and associated constituents) over space and time; (2) predicting impact of industrial and commercial waste discharges on performance of wastewater treatment plants; (3) determining contaminant level that causes a wastewater treatment plant to be in violation of its discharge permits; (4) and developing appropriate user charges based on wasteloads and level of contaminant. Internally, InfoSewer treats the source node as a constant source of a non-reacting constituent that enters the network with a concentration of 100.



H2S Modeling

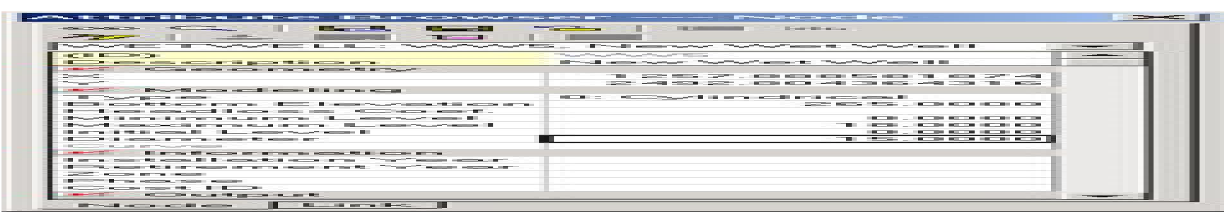
Hydrogen sulfide is the most common odorous gas found in municipal wastewater collection and treatment systems. Colorless, emitting a characteristic odor of rotten eggs, the gas is extremely toxic and can lead to significant corrosion problems, pipeline collapses, and even loss of human life. InfoSewer allows users to readily model and analyze entire sewer collection systems for sulfide generation and corrosion potential under varying conditions anticipated throughout the life of their systems. It enables them to pinpoint odor and corrosion problems, develop effective monitoring programs, alert plant operators and sewer maintenance workers to potential danger and the need to observe safety practices, and implement the most effective control system. (The most common methods for control of hydrogen sulfide are ventilation and scrubbing, and chemical injection.) Users can evaluate alternative pipeline profiles to minimize turbulence, low velocities, long retention times and other hydraulic conditions that promote sulfide buildup. They can also analyze the impact of diversions, future flows, and changes in wastewater characteristics before potentially costly decisions are made.

Application Dependent - InfoSewer Suite.

Corrosion Prediction

Corrosion is one of the primary reasons that sewer systems lose their structural integrity. Corroded sewer pipes may allow greater inflow and infiltration into the collection system, further deteriorating reliability of the network by causing undesirable conditions such as surcharges and overflows, ultimately requiring premature replacement of the pipes. Corrosion of unprotected concrete or metal surfaces is primarily due to the production of sulfuric acid in sewer systems through oxidation of hydrogen sulfide gas by bacterial action on the exposed surfaces under aerobic conditions. This type of corrosion is commonly referred to as microbially induced corrosion. Corrosion in InfoSewer suite helps wastewater engineers to predict the rates of corrosion of the sewer pipes in their collection systems under varying environmental and hydraulic conditions. It enables them to pinpoint corrosion problems, prioritize repairs, specify corrosion resistant materials or select other forms of corrosion protection (e.g., protective linings).

As previously described in the hydrogen sulfide section, some of the soluble H2S available in pipes may escape into the sewer atmosphere and transferred to the pipe walls above the wastewater surface. It will then be taken up as it comes in contact with the damp surfaces of pipes. The H2S retained in this dampness is then converted to sulfuric acid by aerobic bacteria, as described in the following reaction:



Next, some or all of the produced sulfuric acid reacts with the pipe material causing corrosion, mainly at the inside pipe wall above the wastewater flow line. The amount of sulfuric acid that reacts with the pipe material depends on the rate of production of the acid, which in turn depends on moisture, the presence of oxygen, and the mass emission of sulfide gas. If the rate of acid production is slow, almost all of the acid will react with the pipe material. If the rate of production is rapid, much of the acid will not be able to diffuse through the material. Consequently, it will be carried down the walls of the pipe and into the flowing wastewater stream where the sulfuric acid reacts with alkalinity producing sulfate ion.

In addition to the concentration of acid present, the corrosive effect of sulfuric acid varies according to the type of pipe material used and the ambient temperature. Cementitious pipes, including ferrous pipes with mortar lining, experience a reaction that converts the surface material into a pasty mass, which is primarily a calcium sulfate ($CaSO_4$), commonly referred to as gypsum. This pasty mass may fall away and expose new surfaces to corrosive attack. Ferrous pipe materials may experience surface reaction in which a portion of the material is dissolved and a portion is converted to iron sulfide, yielding a hard bulky mass that forms on the exposed surface. A warm and humid environment creates good condition for microbial induced corrosion.

The rate of pipe corrosion depends upon the rate of sulfuric acid production, the amount of the produced sulfuric acid that reacts with the pipe material, and the alkalinity of the pipe material. Sulfuric acid production is related to the amount of hydrogen sulfide gas that escapes to the sewer atmosphere, which in turn depends on a number of environmental and hydraulic conditions previously described in relation to hydrogen sulfide buildup, partitioning, and release. Corrosion PredictorTM uses the following equations to estimate rate of corrosion for cementitious (cement-bonded) materials and ferrous materials in gravity sewers. In force mains, where the lines flow full, there is generally no internal corrosion since generation of sulfuric acid is prevented.

For cement-bonded materials (Metcalf & Eddy 1981),

where

C = average rate of penetration, mm/yr

k = coefficient of efficiency for acid reaction considering the estimated fraction of acid remaining on the wall May be as low as 0.3 and approaches 1.0 for complete acid reaction

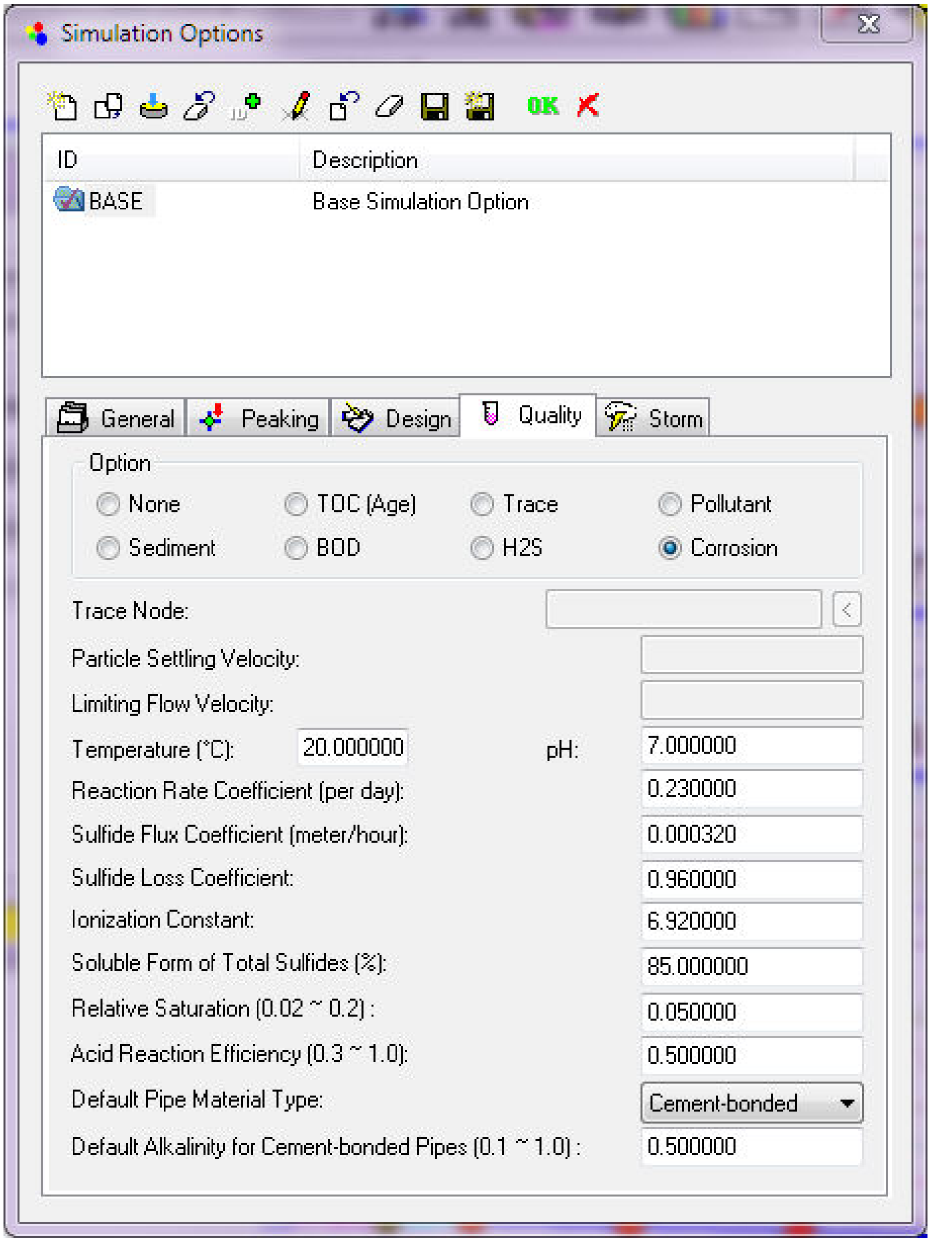
ϕ_{sw} = flux of H2S to the pipe wall, g/m2.hr.

A = alkalinity of the cement-bonded material, expressed as CaCO3 equivalents. Approximately 0.18 to 0.23 for granitic aggregate concrete, 0.9 for calcareous aggregate, 0.4 for mortar linings, and 0.5 for asbestos cement.

For ferrous materials (Metcalf & Eddy 1981),

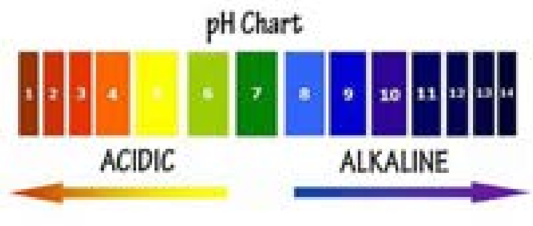
$$C = 2.04 k \phi_{sw}$$

where C, k, and ϕ_{sw} are consistent with the definitions given above. The default values used for k and A are 0.5 and 0.5, respectively.


Simulation Options
OK
ID
Description
BASE
Base Simulation Option
General
Peaking
Design
Quality
Storm
Option
None
TOC (Age)
Trace
Pollutant
Sediment
BOD
H2S
Corrosion
Trace Node:
Particle Settling Velocity:
Limiting Flow Velocity:
Temperature (°C):
20.000000
pH:
7.000000
Reaction Rate Coefficient (per day):
0.230000
Sulfide Flux Coefficient (meter/hour):
0.000320
Sulfide Loss Coefficient:
0.960000
Ionization Constant:
6.920000
Soluble Form of Total Sulfides (%):
85.000000
Relative Saturation (0.02 ~ 0.2) :
0.050000
Acid Reaction Efficiency (0.3 ~ 1.0):
0.500000
Default Pipe Material Type:
Cement-bonded
Default Alkalinity for Cement-bonded Pipes (0.1 ~ 1.0) :
0.500000

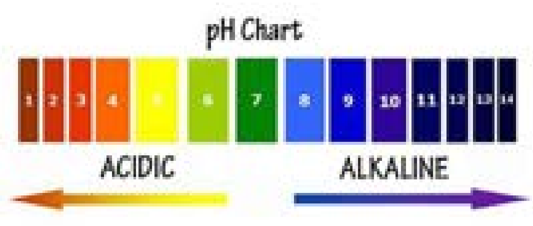
CORROSION PARAMETERS

These parameters are available when the Corrosion option is selected. Corrosion results are generated in addition to Hydrogen Sulfide and any other pollutants entering the system.

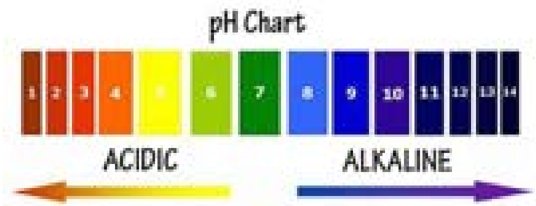


Material-Type - The conduit material (i.e., cement-bonded or ferrous) for gravity pipes.

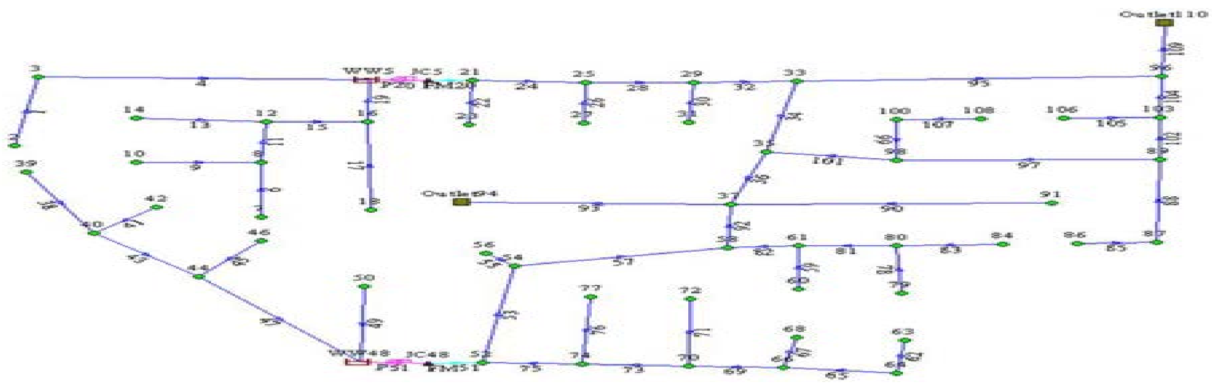
Cement-bonded Alkalinity - The alkalinity of the conduit material expressed as CaCO3 equivalents (for cement-bonded materials only). This value approximately ranges from 0.18 to 0.23 for granitic aggregate concrete, 0.9 for calcareous aggregate, 0.4 for mortar linings, and 0.5 for asbestos cement.



Acid Reaction Coefficient - The coefficient of efficiency for acid reaction considering the estimated fraction of acid remaining on the wall. This value could be as low as 0.3 and will approach 1.0 for a complete acid reaction.



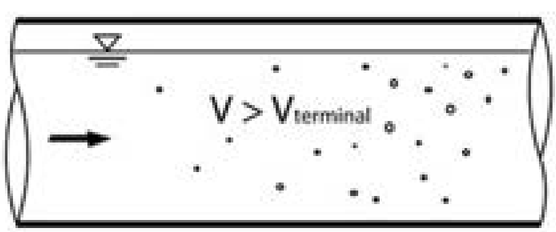
Relative Saturation - The relative saturation of Hydrogen Sulfide in the air compared to equilibrium concentration (typically 2 to 20 percent), expressed as decimal fraction.



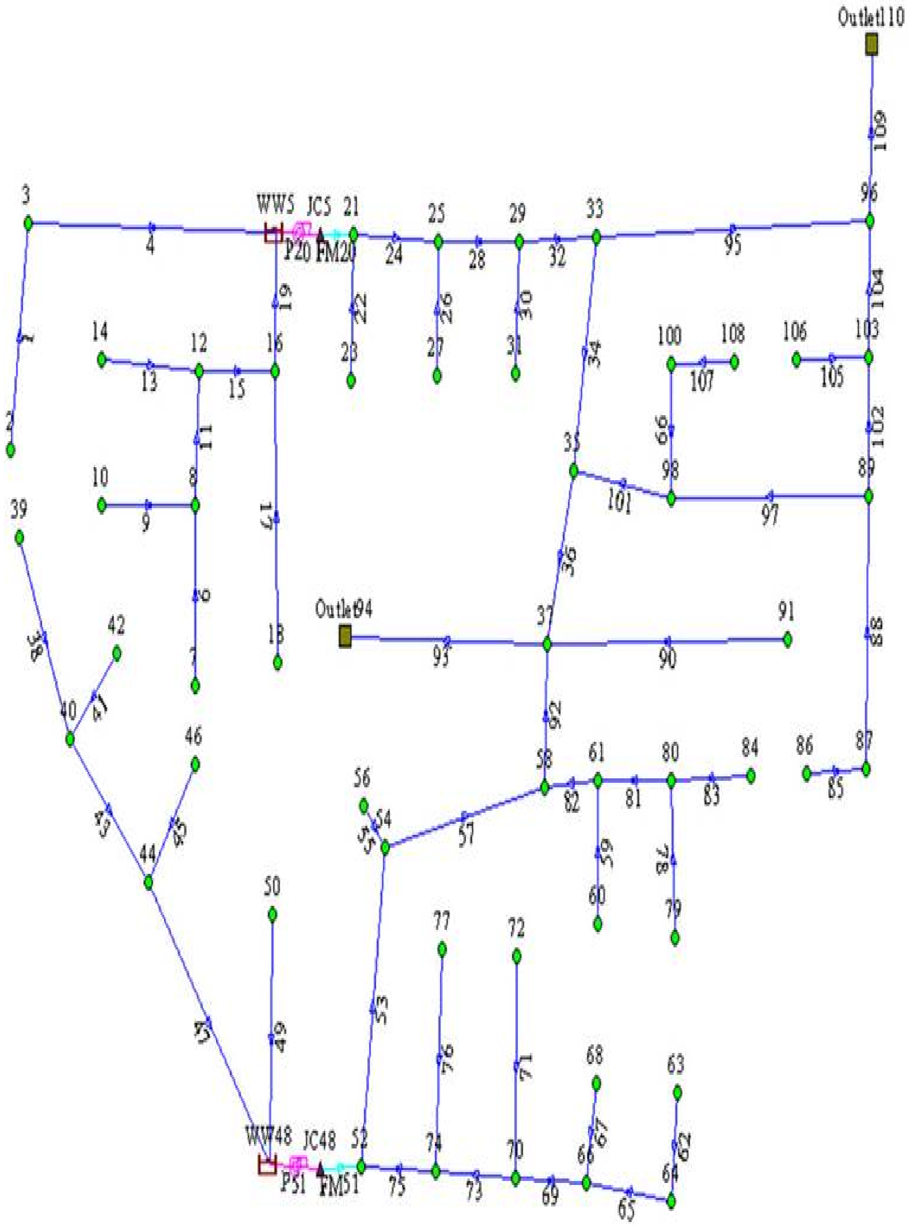
Sediment Transport

Sanitary sewer systems can carry substantial loads of suspended solids (waste solids). These sediments are complex mixtures of cohesive (organic) and non-cohesive (minerals) materials and exhibit a wide range of particle sizes and densities. They can collect causing blockages (shock loading under periods of low flow) and overflow events, as well as impairing the hydraulic capacity of the sewer pipes (by restricting their flow area and increasing the bed friction resistance). In addition, when waste solids are intermittently agitated and moved along the pipes, sulfide generation is increased which may cause various problems including odor, hazard to maintenance crews, and corrosion of unprotected sewer pipes produced from cementitious materials and metals. Limiting velocity criteria are generally adopted when designing sanitary sewer systems to control sediment deposition in the pipes.

Wastewater flow velocity and sediment transport in sewer systems are interdependent. As the flow increases from zero, flow-induced forces (lift and drag) acting on the sediment particle increase. When these forces exceed the submerged weight of the particle, the sediments start to move. With a further increase in velocity, the particles will be suspended by eddies of fluid turbulence and move downstream with the wastewater. With a subsequent decrease in flow velocity (beyond the limiting or terminal velocity), sediment particles will start to settle by gravity at a rate proportional to their settling velocity. A further increase in flow velocity provides the energy to scour and transport the deposited material along the sewer pipes. This dynamic process continues based on the flow conditions inherent in the sewer system.



InfoSewer can simulate the transport and gravitational settling of sediments (total suspended solids including grit) over time throughout the sewer collection system under varying hydraulic conditions. As long as flow velocity exceeds the critical/terminal velocity, InfoSewer assumes that the sewage flow has the capacity to transport all incoming sediments. Deposited sediment particles are also assumed to be scoured and transported downstream when velocity of the sewage flow exceeds the terminal velocity. Settling starts when flow velocity falls below the critical velocity. In the model, transport of the sediment particles is governed by advection (Eq. 24) implying that the particles are transported at local flow velocity. The model assumes that the sewer pipe diameter and roughness coefficient remain constant and are unaffected by sediment deposition.


Outlet110
WW5
JC5
P20
FM20
Outlet94
WW48
JC48
P51
FM51

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

---

InfoSewer™

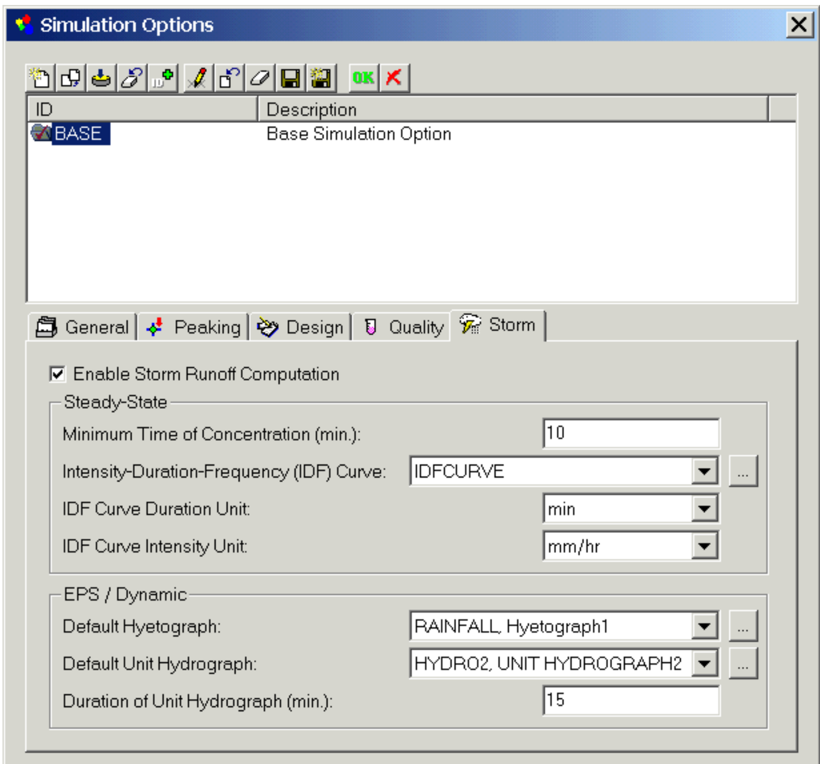
Modeling Made Easy

General Data

In addition to unit hydrograph data and rainfall data, dynamic simulation of storm runoff requires information on area of the subbasin draining to a manhole, default hyetograph and default unit hydrograph pattern, and duration of Unit hydrograph.

Data for area of the subbasins (subwatersheds) could be provided using the attribute browser or using the manhole hydraulic/hydrologic data base table. The remaining three information needs to be supplied using the storm dialog box which is accessible from the simulation options dialog box. From the simulation options dialog box, click on the stormwater simulation icon (

). This will active the storm simulation dialog box shown below.



On this dialog box, check on the storm runoff computation option (

) to activate stormwater modeling. Then provide the data required for dynamic (EPS) simulation (i.e., Default Hyetograph, Default Unit Hydrograph, and Duration of Unit Hydrograph). The model utilizes the default data for loading manholes that lack hyetograph and/or unit hydrograph.

As described in the unit hydrograph data section, the model allows usage of multiple unit hydrographs for modeling of a sewer collection system. Please note that duration of the unit hydrograph must be the same for all unit hydrographs.

Generation of the Unit Hydrograph

InfoSewer Pro generates a direct runoff hydrograph resulting from single or multiple storm events using a unit hydrograph synthesized according to any of the three techniques described above, or from a user-specified natural unit hydrograph. The model relies on a user input hyetograph (i.e., rainfall intensity versus time graph). The model; can derive effective rainfall from user supplied total rainfall hyetographs.

An entire watershed of a collection system may not be well represented by a single unit hydrograph owing to variability in topography, land use, and soil characteristics of the subwatersheds. In the case of separate sewer systems, the magnitude and type of sources of infiltration/inflow will vary by subbasin. Accordingly, InfoSewer Pro allows usage of multiple unit hydrographs, each representing part of the watershed being modeled. The unit hydrographs could be of any duration (e.g., 15-minute, 1-hr), but the duration must be the same for the entire unit hydrographs involved in modeling of the watershed.

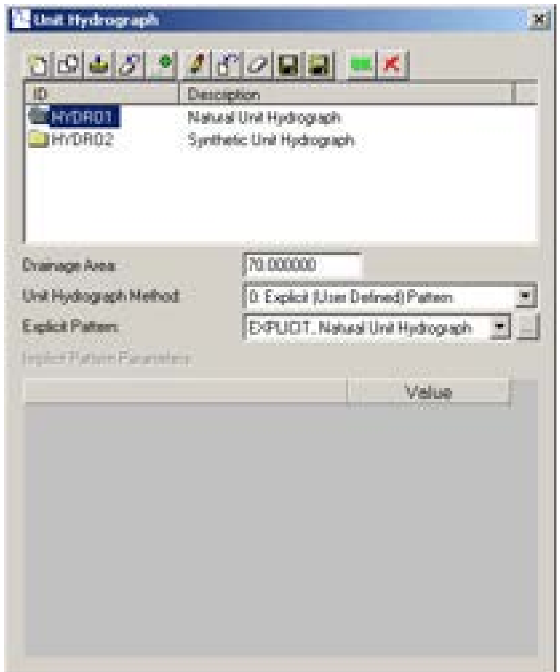
To derive the storm hydrographs, InfoSewer Pro applies the basic assumptions of unit hydrograph theory described above (i.e., linearity, time invariance, and the principle of superposition). Storm events are assumed to have constant intensity over the duration of the unit hydrograph. Excess rainfall resulting from single or multiple storm events is discretized at intervals of unit hydrograph duration. For example, if a sewer collection system is being modeled using a 15-minute unit hydrograph, and if duration of the excess rainfall under investigation is 1-hour, the rainfall duration will be divided into four 15-minute rainfall events of constant intensity. This discretization approach, along with the unit hydrograph assumptions of linearity, time invariance, and superposition, enables InfoSewer Pro to simulate storm runoff hydrograph at any location (e.g., loading manhole) throughout the collection system for any number of storm events. During generation of runoff hydrographs for locations other than the site where the unit hydrograph is originally derived, ordinates of the unit hydrographs are adjusted according to the ratio of drainage area of the two locations. Once the storm hydrographs for every loading manhole in the collection system are known, the storm load will be added to other loading types such as sanitary loads, if any, and will be routed through the collection system using the powerful Muskingum-Cunge’s dynamic flow routing algorithm.

Unit Hydrograph Data

Unit hydrograph information could be supplied to InfoSewer Pro using the Unit Hydrograph icon located at the operation data tab of the control center. Using this icon, the user may create a new unit hydrograph from scratch or by cloning an existing one, and may edit or delete an already developed unit hydrograph.

The procedure for creating a new unit hydrograph is described below.

- Right click on the unit hydrograph icon (Unit Hydrograph).
- Select new, provide unit hydrograph ID and description on the initiated dialog box, and then click Ok. At this stage, a Unit Hydrograph dialog box shown below would appear.

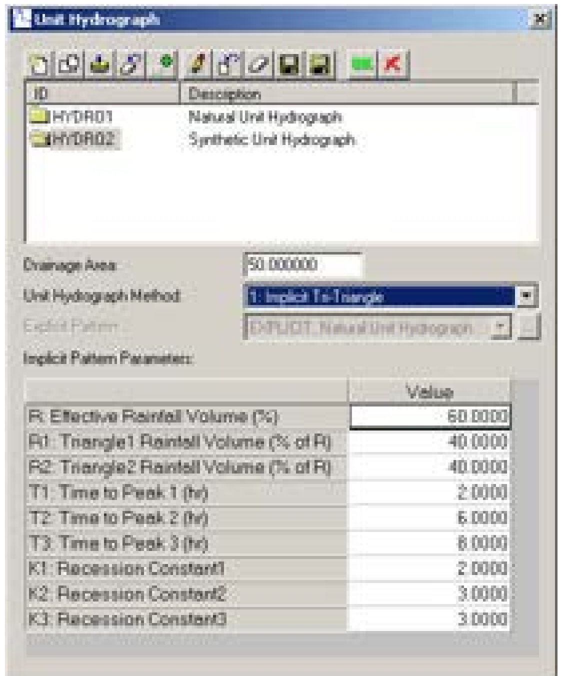


- Depending on the type of unit hydrograph (i.e., natural or synthetic) to be created, one needs to provide all the necessary information. Each of these data is described below.

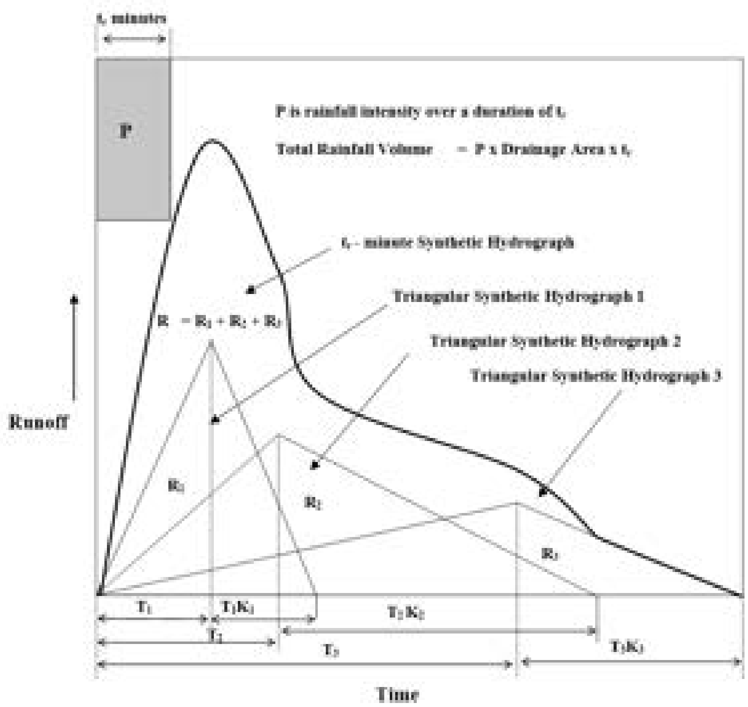
Drainage Area: This refers to total area of the subwatersheds draining up to the location where the unit hydrograph is being derived. If the unit hydrograph is being developed at the outlet of the watershed, the area should be that of the entire watershed. This figure is mandatory, and must be provided whether natural unit hydrograph or synthetic unit hydrograph is used. The model utilizes this area to generate unit hydrographs at other locations in the watershed.

Unit Hydrograph Method: This refers to the type (i.e., natural or synthetic) of unit hydrograph, and the method used to generate synthetic unit hydrographs.

- If the unit hydrograph is provided by the user (i.e., an existing unit hydrograph is used), an explicit option has to be chosen. The model accepts existing unit hydrographs in the form of pattern. Assign the unit hydrograph pattern using the "**Explicit Pattern**" dialog box.
- If a synthetic unit hydrograph generated by InfoSewer Pro is to be used, the modeler may choose one of the four implicit unit hydrograph (i.e., the tri-triangle method, the SCS dimensionless unit hydrograph, the SCS triangular unit hydrograph, or the Colorado Urban Hydrograph Procedure) options. Inputs for each of these four synthetic unit hydrographs are described below.
- **Tri-triangle method:** Once this option is selected, the following dialog box would appear enabling the user to specify parameters of the tri-triangle.



These parameters are described using the following figure.

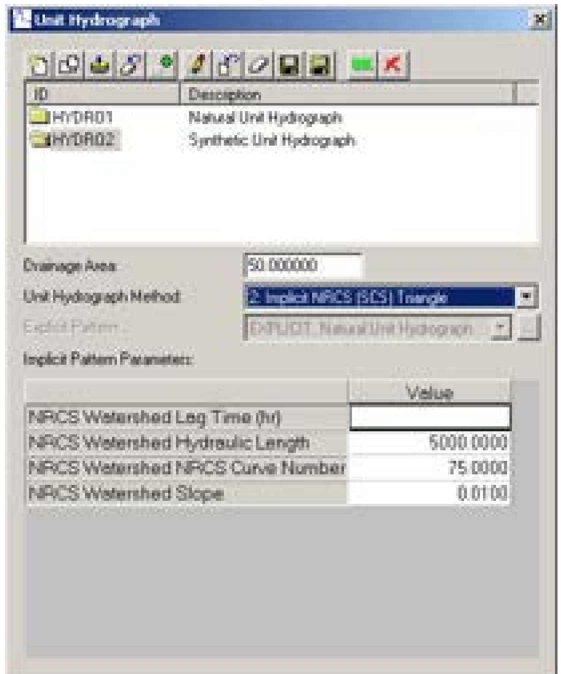


R is the percentage of the total rainfall volume that becomes runoff. Please, note that InfoSewer Pro has advanced rainfall loss modeling capability and can estimate effective rainfall (, i.e., part of rainfall that becomes runoff). **If this loss modeling feature is used, R should be set to 100%.**

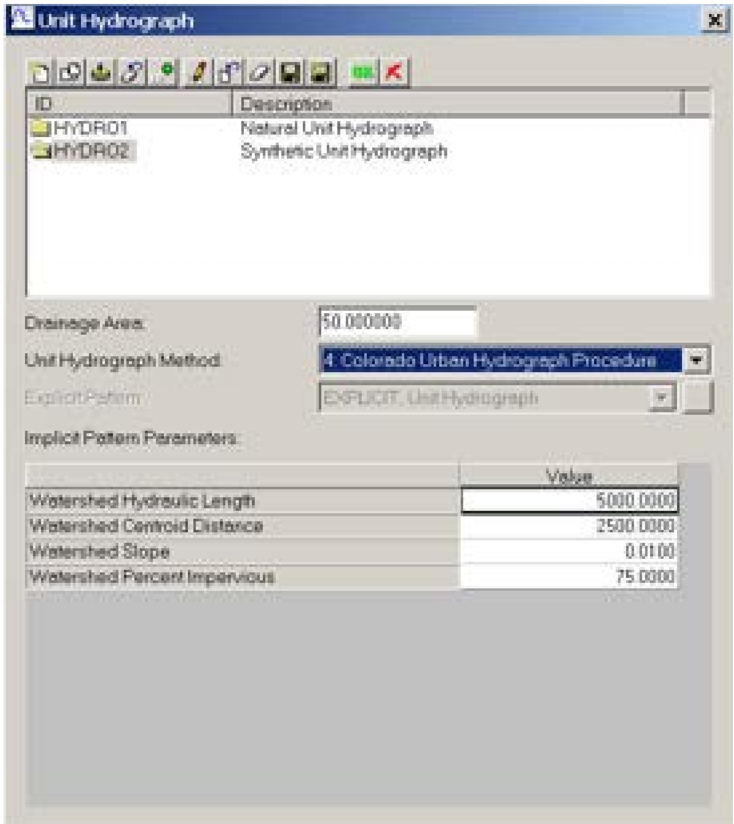
$\mathbf{R_1}$, $\mathbf{R_2}$, and $\mathbf{R_3}$ are percentages of **R** that are allocated to triangle 1, triangle 2, and triangle 3, respectively. For example, assume that ***R*** equals 40 % of the total rainfall volume and $\mathbf{R_1}$and $\mathbf{R_2}$ are 30% and 40%, respectively. The actual volume of runoff allocated to triangle 1 and triangle 2 would be 0.4*0.3* total rainfall volume and 0.4 *0.4* total rainfall volume, respectively. R in InfoSewer Pro is the same as R1 in SWMM 5 and InfoSWMM, R1 is the same as R2 in SWMM5 and InfoSWMM and R2 is the same as R3 in SWMM5 and InfoSWMM.

$\mathbf{T_1}$, $\mathbf{T_2}$, and $\mathbf{T_3}$ are time to peaks of triangle 1, triangle 2, and triangle 3, respectively. $\mathbf{K_1}$, $\mathbf{K_2}$, and $\mathbf{K_3}$ refers to recession constants of triangle 1, triangle 2 and triangle 3, respectively. The default values used by InfoSewer Pro for $\mathbf{T_1}$, $\mathbf{T_2}$, $\mathbf{T_3}$, $\mathbf{K_1}$, $\mathbf{K_2}$, and $\mathbf{K_3}$ are 1 hour, 4 hour, 12 hour, 2, 3, and 3, respectively.

- **NRCS (SCS) Triangular Unit Hydrograph:** Generation of unit hydrograph using this method requires specification of the inputs displayed in the following dialog box.



  - **Watershed Lag Time** - this represents the time from the center of mass of effective rainfall to the time to peak of a unit hydrograph. In other words, lag time is a delay in time, after a brief rain over a watershed, before the runoff reaches its peak. This parameter can either be specified by the user, or can be calculated by the model based on watershed characteristics. If lag time of the watershed is directly specified by the user, the modeler does not need to provide the remaining inputs (i.e., hydraulic length, the curve number, and the slope).
  - **Watershed Hydraulic Length** - This refers to travel distance of water from the most upstream location of the watershed to the point where the unit hydrograph is being derived. This input is required if watershed lag time is not specified. The value should be given in feet for US Customary system, and in meters for SI units.
  - **Watershed NRCS (SCS) Curve Number -** This parameter refers to runoff generating capacity of a watershed, and its value depends on the soil, antecedent moisture condition, cover and the hydrologic conditions of the watershed. Recommended values of curve number are available from standard hydrology books. The SCS suggests the curve number values to be within 50 and 95. This input is required only if watershed lag time is not specified.
- **Watershed Slope** - Refers to average slope of the watershed. This input is required if watershed lag time is not specified.
- **NRCS Dimensionless Unit Hydrograph:** The inputs required for this method are the same as those described with regard to the NRCS (SCS) triangular unit hydrograph.
- **Colorado Urban Hydrograph Procedure:** This technique requires provision of the following subwatershed information.



- **Watershed Hydraulic Length** - This refers to travel distance of water from the most upstream location of the watershed to the point where the unit hydrograph is being derived. The value should be given in feet for US Customary system, and in meters for SI units.
- **Watershed Centroid Distance** - Travel distance of water from the most upstream location of the watershed to the centroid of the catchment. The value should be given in feet for US Customary system, and in meters for SI units.
- **Watershed Slope** - Refers to average slope of the watershed. This input is required if watershed lag time is not specified.
- **Watershed Percent Impervious** - Portion of the watershed area that is impervious in nature (e.g., pavement, roof top, etc.)

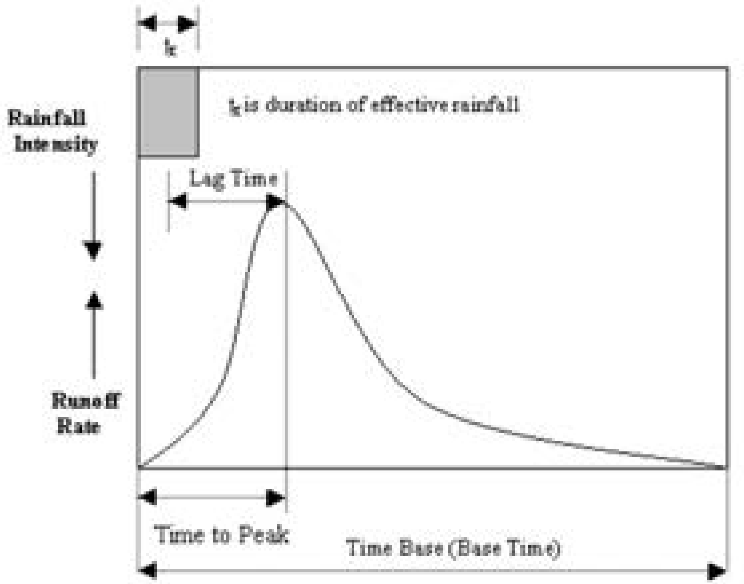
An entire watershed of a collection system may not be well represented by a single unit hydrograph due to variability in topography, land use, and soil characteristics of the subwatersheds. InfoSewer Pro allows usage of multiple unit hydrographs, each representing part of the watershed being modeled. This is an advanced functionality usually referred to as **spatially distributed modeling**. This implies that, if interested, the user may use unique unit hydrograph for every loading manhole in the collection system. All the unit hydrographs utilized for modeling of the watershed should be created following the procedure described above. The created unit hydrographs may be assigned to loading manholes in the collection system through the Attribute Browser or by using the hydraulic/hydrologic manhole data base table.

It should be noted that, if multiple unit hydrographs are used for modeling of a collection system, the type of all the unit hydrographs need not be the same. Some of the unit hydrographs could be NRCS (SCS) triangular, some others could be NRCS (SCS) dimensionless, and the remaining could be explicit or could be derived using the tri-triangle method.

Unit Hydrograph

The unit hydrograph method is the most commonly used method of stormwater modeling for applications that require generation of a complete hydrograph with reasonable accuracy. A unit hydrograph is defined as the direct runoff hydrograph resulting from a unit depth of excess (effective) rainfall produced by a storm of uniform intensity and specified duration. Unit hydrograph is a specific type of hydrograph that represents the effects of the physical characteristics of the watershed on the input rainfall excess. A typical unit hydrograph is shown below.

Basic terminology and assumptions of the unit hydrograph method are presented below, followed by a description of the technique used by InfoSewer Pro for regeneration of unit hydrographs and runoff hydrographs.



Excess Rainfall

During a storm event, not all of the rainfall is converted to runoff. Part of the rainfall is “lost” in the form of deep infiltration (percolation), evaporation, interception, and depression storage. The amount of the precipitation (rainfall) actually reaching the outlet of the watershed or the subwatershed in the form of runoff is known as excess rainfall. Excess rainfall is sometimes called effective rainfall.

Direct Runoff

Direct runoff is a storm flow resulting from excess rainfall. It is an aggregate of surface runoff and quick subsurface runoff. Surface runoff is an overland flow that occurs when rainfall intensity exceeds infiltration capacity of the soil. Surface runoff flows on the surface of the watershed and through tributary channels to the outlet or to the point of reference such as loading manhole. Quick subsurface runoff is part of an infiltrated rainfall that travels underground and contributes runoff to the location of interest during or soon after the storm event. Unit hydrographs and subsequent runoff hydrographs derived based on unit hydrographs represent direct runoff.

Duration of Unit Hydrograph

This is an important feature of a unit hydrograph, and represents duration of the rainfall excess (represented as tr in the above figure) that produced the unit hydrograph. For a given watershed or subwatershed, all storm events with a given duration of excess rainfall produce direct runoff hydrographs that have the same time to peak and same time base. If the effective duration of a storm differs from that of another on the same basin, the resulting direct runoff hydrograph will have different times to peak and different time bases. Unit hydrograph is named in terms of its duration (i.e., 15-minute unit hydrograph, 1-hour unit hydrograph). As an example, a unit hydrograph derived for a watershed from excess rainfall of a unit depth collected over 15-minute duration is known as a 15-minute unit hydrograph.

Basic assumptions in Unit Hydrograph

- Intensity of excess rainfall is constant within the duration of the excess rainfall for the entire watershed represented by the unit hydrograph.
- One of the fundamental assumptions of the unit hydrograph theory is linearity. It is assumed that an increase in depth of excess rainfall increases ordinates of the direct runoff hydrograph proportionally.
- The unit hydrograph method also assumes that for a given pattern (temporal distribution) of rainfall, the ratio of ordinates of direct runoff hydrograph to depth of effective rainfall is time invariant (e.g., seasonally and within rainfall event(s)).
- Direct runoff hydrograph resulting from a given pattern of rainfall excess can be built by superimposing the unit hydrographs resulting from the separate amounts of rainfall excess occurring in each unit period. This is known as the principle of superposition. Application of these basic assumptions is illustrated in the figure below.

Rainfall Intensity

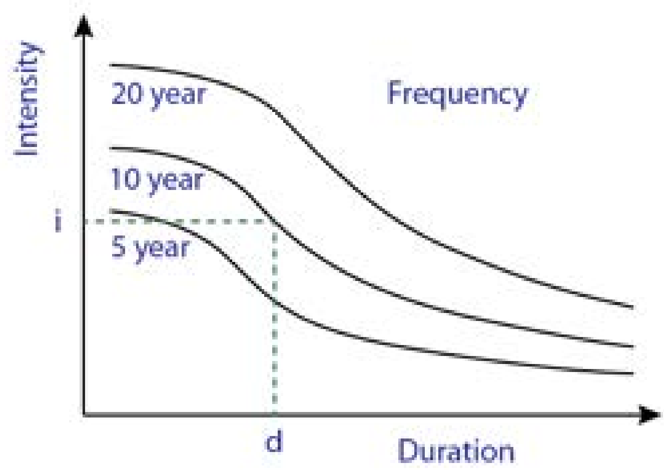
Rainfall intensity is defined as the depth of rain that falls in a given duration of time divided by that duration. It may rain harder (high rainfall intensity) for a short period of time, but the longer a rainfall event lasts, the lower its average rainfall intensity, implying that rainfall intensity is inversely proportional to the duration of a rainfall event.

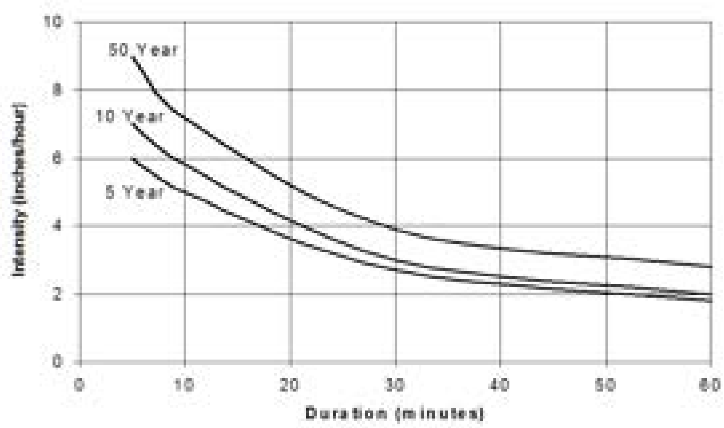
Intensity of a storm is also a function of return period or frequency of the rainfall occurrence. Intense rainfall events occur less frequently than less intense storms. Return period, also known as recurrence interval, is defined as the expected length of time (e.g., T years) between occurrences of two storm events that exceed a given intensity. Return period does not imply that multiple rainfall events exceeding a given intensity will not occur within T years, nor does it assure occurrence of a storm event exceeding the intensity within T years. All it means is that the rainfall event will occur at an average rate of once in T years.

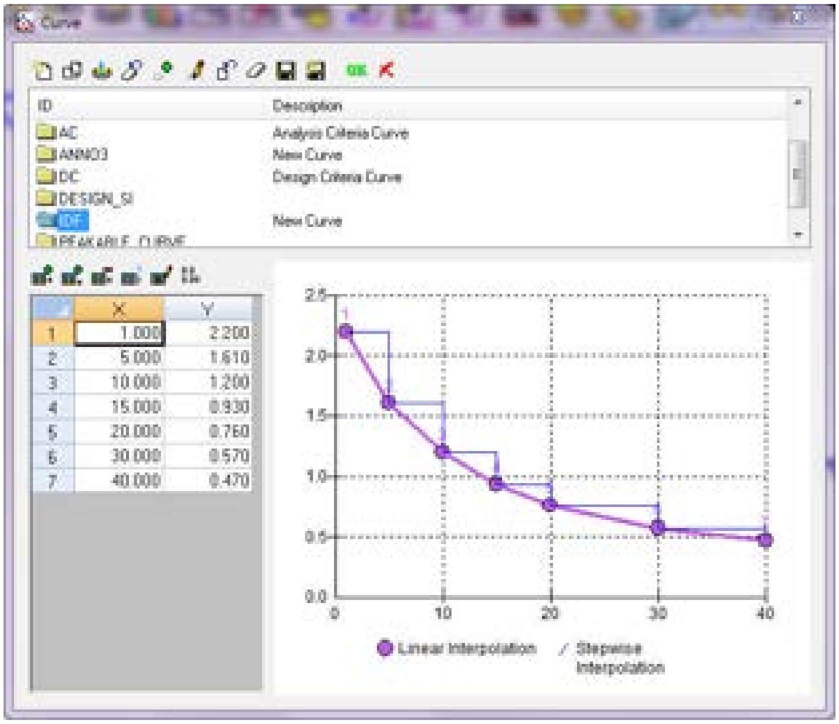
The higher the return period (i.e., the less frequent the storm event), the higher the intensity of the rainfall event would be. As an example, a storm with 10-year return period represents a rainfall event whose magnitude is expected to be exceeded once in ten years. This implies that the frequency of the storm event is 1/10, or that there is a 10% probability for the intensity of the storm event to be exceeded in any given year.

Intensity-Duration-Frequency Curve

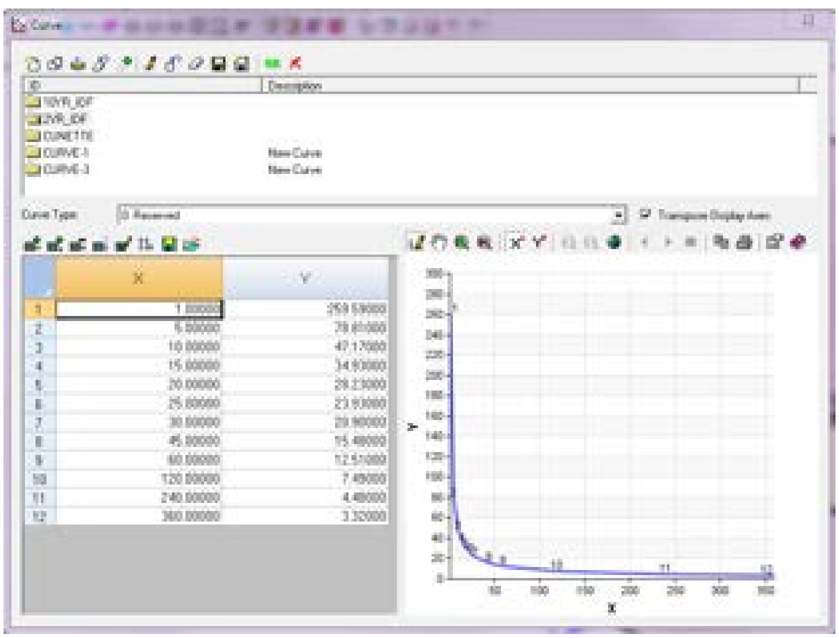
The relationship between rainfall intensity, rainfall duration, and frequency of occurrence of storm events is often presented for a certain region in the form of an Intensity-Duration-Frequency Curve, commonly referred to as an IDF curve. A sample IDF curve is given below for return periods of 5, 10, and 50 years. InfoSewer Pro calculates the rainfall intensity used by the Rational method based on a user-specified IDF curve. The user should provide an IDF curve for a desired return period. The model derives rainfall intensity from the IDF curve corresponding to the duration equal to the time of concentration calculated for the manhole following the techniques described above.







Sewer IDF Curver Table in the Atrribute Browser in USA Units



An Example IDF Curve from InfoSWMM in SI Units

Time of Concentration in Hydrology

The Rational method is based on the premise that peak runoff occurs at a given design point when the rainfall duration is sufficiently long so that all locations in the watershed draining to the point contribute flow, and that intensity of the rainfall is constant for that duration. Some locations of the contributing subbasins are hydraulically closer to the design point and may yield runoff to the point sooner than other locations. The Rational method uses the concept of time of concentration to ensure contribution of all locations in the upstream subwatersheds to the design point. Time of concentration, from the perspective of the Rational method and storm runoff (not to be confused with the water quality definition given in preceding sections), is defined as the time it takes for a drop of water falling on the most remote point of upstream subwatersheds to reach the design point. Remoteness refers to hydraulic travel time rather than distance.

There are a number of empirical methods for estimation of time of concentration. The techniques are formulated as a function of subwatershed characteristics such as travel distance and subwatershed slope. InfoSewer Pro calculates time of concentration as the sum of overland flow travel time and channel (pipe) flow travel time.

$$T_c = t_o + t_c$$

where T_c = time of concentration for the manhole (in minutes)

t_o = overland flow travel time (in minutes)

t_c = channel flow travel time (in minutes)

Pipe flow travel time is calculated using the flow velocity of the pipe and the pipe length. Overland flow travel time is calculated using the method derived by Kirpich, a technique commonly used in the US. Kirpich's formula (Mays 2001) is given as:

$$t_o = 0.0078L^{0.77}S^{-0.385}$$

where t_o = overland flow travel time for a subwatershed (in minutes)

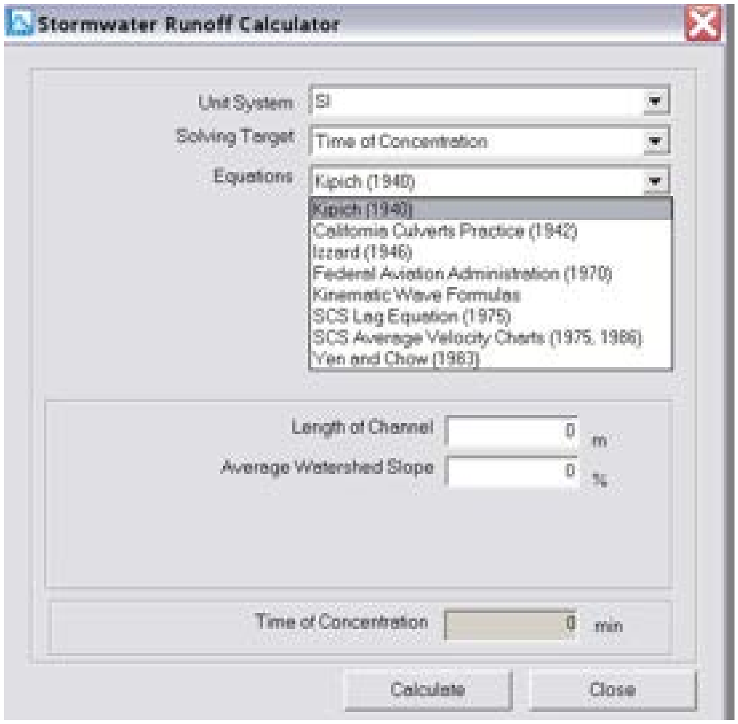
L = length of flow path from the remotest spot in the subwatershed (in feet)

S = average slope of the subwatershed (unitless)

There could be multiple flow paths from upstream subwatershed to a given manhole leading to different flow times. The time of concentration used by InfoSewer Pro is the longest flow time among different possible flow paths to the manhole. In order to avoid unreasonably low storm duration and unreasonably high rainfall intensities, InfoSewer Pro allows the user to specify a minimum time of concentration, and uses the maximum of the user-specified minimum time of concentration and an internally calculated time of concentration. The default minimum time of concentration used by InfoSewer Pro is 10 minutes.

Time of Concentration from Innovyze H_2OCalc for Reference

The dialog box for estimating time of concentration using the Kirpich method is shown below.


Stormwater Runoff Calculator
Unit System
SI
Solving Target
Time of Concentration
Equations
Kipich (1940)
Kipich (1940)
California Culverts Practice (1942)
Izzard (1946)
Federal Aviation Administration (1970)
Kinematic Wave Formulas
SCS Lag Equation (1975)
SCS Average Velocity Charts (1975, 1986)
Yen and Chow (1983)
Length of Channel
0
m
Average Watershed Slope
0
%
Time of Concentration
0
min
Calculate
Close

- **General Input for the time of concentration methods:**
- **Unit System** – English or SI unit.
- **Solving Target** – Choose time of concentration.
- **Equations** – Select one of the eight empirical equations listed in the dialog box shown above.• Unit System – English or SI unit.
- **Solving Target** – Choose time of concentration.
- **Equations –** Select the desired time of concentration estimation equation among the available eight methods. The table given below described these equations in further detail.

The inputs required for specific the time of concentration estimation methods (see the following table below for the specific formula)

- For Kirpich (1940) Equation
- **Length of Channel** – Length of the longest overland flow path for the watershed in feet.
- **Average Watershed Slope** – Average slope for the longest flow channel.

- For California Culverts Practice (1942) Equation
- **Length of Longest Channel** – Length of the longest overland flow path for the watershed in miles.
- **Elevation Difference Between Divide and Outlet** – The difference in elevation (in feet) between the upstream end of the flow path and the outlet of the watershed.

- For Izzard (1946) Method
- **Rainfall Intensity** – Intensity of the design rainfall (in/hr)
- **Length of Flow Path** – Length of the longest overland flow path for the watershed in feet. The product rainfall intensity and the length of flow path should be < 500.
- **Slope of Flow Path** –Average slope for the longest flow channel.
- **Retardance Coefficient** – Coefficient that accounts for friction effect of the channel material. Retardance factor ranges from 0.007 for smooth pavement to 0.012 for concrete and to 0.06 for dense turf. The product rainfall intensity and the length of flow path should be < 500.

- Federal Aviation Administration (1970) Method
- **Runoff Coefficient** – Refers to the runoff coefficient used in rational formula.
- **Length of Overland Flow**– Length of the longest overland flow path for the watershed.
- **Surface Slope** – Average slope of the watershed.

- Kinematic Wave Formula
- **Rainfall Intensity** – Intensity of the design rainfall (in/hr).
- **Length of Overland Flow** – Length of the longest overland flow path for the watershed.
- **Average Overland Slope** - Average slope for the longest flow channel.
- **Manning's Roughness Coefficient** – Resistance coefficient used in Manning equation.

- **SCS Lag Equation (1975)**
- **Length of Flow Path** – Intensity of the design rainfall (in/hr).
- **Average Watershed Slope** - Average slope for the watershed.
- **Curve Number (CN)** – NRCS curve number used as an index of the watershed's runoff generation potential.

- SCS Average Velocity Charts (1975, 1986)
- **Length vs Velocity Chart** – Specify average flow velocity for various channel lengths.

- Yen and Chow (1983) Method
- **Length of Flow** – Length of the longest overland flow path for the watershed in feet.
- **Average Watershed Slope** – Average slope for the watershed..
- **Coefficient Ky** – Coefficient. KY ranges from 1.5 for light rain (intensity <0.8) to 1.1 for moderate rain (0.8 < intensity < 1.2), and to 0.7 for heavy rain (intensity >1.2)
- **Overland Texture Factor** – Overland texture factor. See Table 3.13.

- **Output for the SCS peak discharge method**
- **Time of Concentration** – The time it takes for flow to travel from the hydraulically remotest point in the watershed to reach outlet of the watershed.

Table 3-11: Formulas for Computing Time of Concentration

Method	Formula	
Kirpich (1940)	$t_c = 0.0078L^{0.77}S - 0.385$ L = length of channel (ft) S = average watershed slope (ft/ft)	
California Culverts Practice (1942)	$t_c = 60\left(11.9L^3 / H\right)^{0.385}$ L = length of the longest channel (mi) H = elevation difference between divide and outlet (ft)	
Izzard (1946)	$t_c = \frac{41.025(0.0007i + c)L^{1/3}}{S^{1/3}i^{2/3}}$ i = rainfall intensity (in/h) c = retardance coefficient	Retardance factor, c, ranges from 0.007 for smooth pavement to 0.012 for concrete and to 0.06 for dense turf; product i times L should be < 500
Federal Aviation Administration (1970)	$t_c = \frac{0.39(1.1 - C)L^{1/2}}{S^{1/3}}$ C = rational method runoff coefficient (see Table 3.9)	
Kinematic wave	$t_c = \frac{0.938L^{0.6}n^{0.6}}{i^{0.4}S^{0.3}}$ n = Manning's roughness coefficient	
SCS lag equation	$t_c = \frac{100L^{0.8}\left[(1000/CN) - 9\right]^{0.7}}{19000S^{1/2}}$ CN = SCS runoff curve number (see Table 3.10)	
SCS average velocity charts	$t_c = \frac{1}{60}\sum_{j=1}^{N}\left(L_j / V_j\right)$ V = average velocity (ft/s)	
Yen and Chow (1983)	$t_c = K_Y\left(\frac{NL}{S^{1/2}}\right)^{0.6}$ K_Y = Coefficient N = Overland texture factor (see Table 3.13)	K_Y ranges from 1.5 for light rain (i<0.8) to 1.1 for moderate rain (0.8<i<1.2), and to 0.7 for heavy rain (i>1.2)

Source: Nicklow et al. (2004)

Rational Method

InfoSewer Pro adds comprehensive rainfall-runoff and infiltration/inflow modeling capability on top of the sanitary sewer simulation features provided by InfoSewer , equipping the industry with a single powerful tool that can be used for planning, design, and operational management of sanitary sewer, storm sewer, and combined sewer systems. InfoSewer Pro shares all the features described above in relation to InfoSewer including the methodologies used to model the physical and the non-physical components of a collection system, steady state and dynamic analysis of sanitary loads, steady state design, dynamic routing of flows through the collection system, calculation of HGLs, and water quality modeling. This section presents the theory pertinent to InfoSewer Pro, which is stormwater modeling.

Determination of stormwater runoff resulting from rainfall event(s) is the most important component of stormwater management, and is commonly known as rainfall-runoff modeling. Similar modeling techniques are applicable to the determination of rainfall-dependent infiltration/inflow in separate sanitary sewer systems. There are various rainfall-runoff modeling techniques. The choice of the method to use depends, among other considerations, on the type of analysis (e.g., steady state or dynamic simulation) and the information required (e.g., peak flow or complete hydrograph). Peak flow information is appropriate for steady state simulation of sewer systems whereas knowledge of the complete hydrograph is essential for dynamic simulations.

For estimation of peak flow during steady state simulations, InfoSewer applies the Rational method, which is a very popular tool of choice used by many practicing engineers across the globe. For dynamic simulation, complete hydrographs are derived based on the practical and highly effective unit hydrograph theory. The unit hydrograph can be observed or synthetic. This method allows the entire hydrograph (including peak and volume) resulting from an actual or design storm event to be adequately simulated. The following sections describe the Rational method and the unit hydrograph techniques incorporated into InfoSewer .

Rational Method

The Rational method is the most widely used technique for estimation of peak flows from urban and rural drainage basins (Maidment 1993; Mays 2001). The concept of the Rational method is attractive and easy to understand. If rainfall occurs over a watershed at a constant intensity for a period of time that is sufficiently long to produce steady state runoff at a desired design point, then the peak flow rate will be proportional to the product of rainfall intensity and watershed area.

Mathematically, the Rational formula is expressed as:

$$Q = CiA$$

where Q = peak runoff rate (flow unit)

C = runoff coefficient (unitless)

i = rainfall intensity (intensity unit)

A = watershed area (area unit).

The units need to be consistent. For example, if intensity is in ft/s and area is in ft2, then the resulting peak flow rate would be in cfs. InfoSewer allows several intensity unit alternatives (in/hr, mm/hr, in/min, mm/min). The area should be in acres if US Customary units are used, and in square meters for SI units. InfoSewer converts the units internally to preserve consistency.

TIME OF CONCENTRATION

The Rational method is based on the premise that peak runoff occurs at a given design point when the rainfall duration is sufficiently long so that all locations in the watershed draining to the point contribute flow, and that intensity of the rainfall is constant for that duration. Some locations of the contributing subbasins are hydraulically closer to the design point and may yield runoff to the point sooner than other locations. The Rational method uses the concept of time of concentration to ensure contribution of all locations in the upstream subwatersheds to the design point. Time of concentration, from the perspective of the Rational method and storm runoff (not to be confused with the water quality definition given in preceding sections), is defined as the time it takes for a drop of water falling on the most remote point of upstream subwatersheds to reach the design point. Remoteness refers to hydraulic travel time rather than distance.

There are a number of empirical methods for estimation of time of concentration. The techniques are formulated as a function of subwatershed characteristics such as travel distance and subwatershed slope. InfoSewer calculates time of concentration as the sum of overland flow travel time and channel (pipe) flow travel time.

$$T_c = t_o + t_c$$

where Tc = time of concentration for the manhole (in minutes)

to = overland flow travel time (in minutes)

tc = channel flow travel time (in minutes)

Pipe flow travel time is calculated using the flow velocity of the pipe and the pipe length. Overland flow travel time is calculated using the method derived by Kirpich, a technique commonly used in the US. Kirpich's formula (Mays 2001) is given as:

$$t_o = 0.0078\,L^{0.77}S^{-0.385}$$

where to = overland flow travel time for a subwatershed (in minutes)

L = length of flow path from the remotest spot in the subwatershed (in feet)

S = average slope of the subwatershed (unitless)

There could be multiple flow paths from upstream subwatershed to a given manhole leading to different flow times. The time of concentration used by InfoSewer is the longest flow time among different possible flow paths to the manhole. In order to avoid unreasonably low storm duration and unreasonably high rainfall intensities, InfoSewer allows the user to specify a minimum time of concentration, and uses the maximum of the user-specified minimum time of concentration and an internally calculated time of concentration. The default minimum time of concentration used by InfoSewer 10 minutes.

RAINFALL INTENSITY

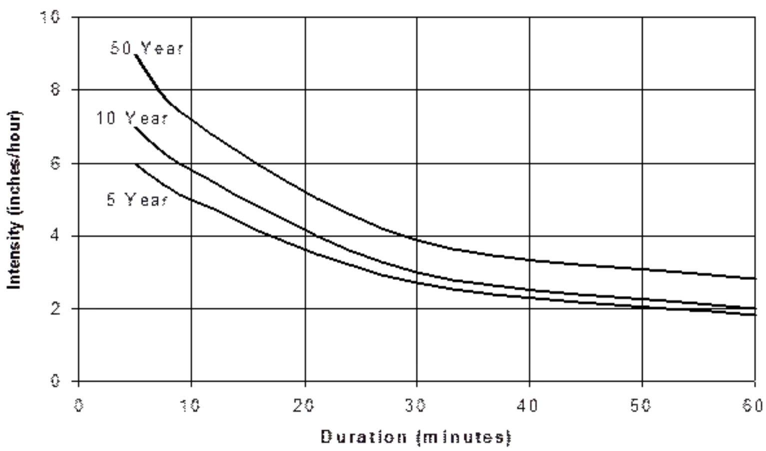
Rainfall intensity is defined as the depth of rain that falls in a given duration of time divided by that duration. It may rain harder (high rainfall intensity) for a short period of time, but the longer a rainfall event lasts, the lower its average rainfall intensity, implying that rainfall intensity is inversely proportional to the duration of a rainfall event.

Intensity of a storm is also a function of return period or frequency of the rainfall occurrence. Intense rainfall events occur less frequently than less intense storms. Return period, also known as recurrence interval, is defined as the expected length of time (e.g., T years) between occurrences of two storm events that exceed a given intensity. Return period does not imply that multiple rainfall events exceeding a given intensity will not occur within T years, nor does it assure occurrence of a storm event exceeding the intensity within T years. All it means is that the rainfall event will occur at an average rate of once in T years.

The higher the return period (i.e., the less frequent the storm event), the higher the intensity of the rainfall event would be. As an example, a storm with 10-year return period represents a rainfall event whose magnitude is expected to be exceeded once in ten years. This implies that the frequency of the storm event is 1/10, or that there is a 10% probability for the intensity of the storm event to be exceeded in any given year.

INTENSITY-DURATION-FREQUENCY CURVE

The relationship between rainfall intensity, rainfall duration, and frequency of occurrence of storm events is often presented for a certain region in the form of an Intensity-Duration-Frequency Curve, commonly referred to as an IDF curve. A sample IDF curve is given below for return periods of 5, 10, and 50 years. InfoSewer calculates the rainfall intensity used by the Rational method based on a user-specified IDF curve. The user should provide an IDF curve for a desired return period. The model derives rainfall intensity from the IDF curve corresponding to the duration equal to the time of concentration calculated for the manhole following the techniques described above.



RUNOFF COEFFICIENT

Runoff coefficient is loosely defined as the ratio of runoff to rainfall, and is a function of watershed characteristics including land use, soil type, and slope of the watershed. The value of runoff coefficient ranges between 0.0 and 1.0. A value of 0.0 means that all of the rainfall is lost in the form of abstractions such as infiltration, interception, and evaporation and none of the rainfall is converted to runoff. The value of 1.0 implies that all the rainfall is converted to runoff and is discharged from the watershed. As an example, most of the rain that falls on impervious areas such as pavement and roof would be immediately converted to runoff. A value of C for such land uses is close to 1.0. Runoff coefficient values recommended by the American Society of Civil Engineers and Water Environment Federation for return periods not exceeding 10 years are given below for various land uses, soil types, and slope conditions.

Description Of Area	Runoff Coefficient
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
Pavement	
Asphaltic and concrete	0.70 - 0.95
Brick	0.70 - 0.85
Roofs	0.75 - 0.95
Lawns, sandy soil	
Flat, 2%	0.05 - 0.10
Average, 2-7 %	0.10 - 0.15
Steep, 7%	0.15 - 0.20
Lawns, heavy soil	
Flat, 2%	0.13 - 0.17
Average, 2-7 %	0.18 - 0.22
Steep, 7%	0.25 - 0.35

For return periods that exceed 10 years, the runoff coefficient from the table should be multiplied by a frequency adjustment factor, C_f , given below.

Frequency (Years)	C_f
25	1.1
50	1.2
100	1.25

For a subwatershed composed of multiple land uses, a composite runoff coefficient Cw should be determined by weighting C values of each of the land uses by their corresponding area according to the equation:.

$$C_w = \sum_{i=1}^{N}(C_i A_i) \Big/ \sum_{i=1}^{N}(A_i)$$

where Ci = runoff coefficient for individual land use in the subwatershed.

Ai = area of the individual land use in the subwatershed.

N = total number of land uses in the subwatershed.

ASSUMPTIONS OF THE RATIONAL METHOD

For credible engineering application, the modeler needs to understand the following basic assumptions of the rational method. Firstly, the rainfall intensity is constant throughout the watershed over a period of time that equals the time of concentration of the analysis site. Secondly, the runoff coefficient is invariant, regardless of season of the year or intensity of rainfall. Finally, the area of the contributing watershed is small (not more than 300 acres).

THE SAN Diego Rational/Modifed Rational Method from Innovyze H_2OCalc for Reference

The San Diego County hydrology manual (San Diego County, 2003) recommends rational method to compute peak storm flow for drainage areas less than 1 square mile. If more than one drainage path is flowing to a junction, the County recommends the modified rational method described below.

The county uses the following relationship to estimate design rainfall intensity (in/hr).

I = 7.44P6D-0.645

Where: P6 = 6-hour storm rainfall amount (in).

D = duration (time of concentration) in minutes.

The time of concentration is composed of the initial time of concentration and the travel time.

InfoSewer requires the user to provide initial time of concentration for each subwatershed.

The travel time is computed based on flow velocity and pipe length.

The modified rational method differs from the rational method only when a junction receives flow from two or more tributaries. The peak flow (Q), the time of concentration (Tc) and the rainfall intensity (I) for each of the independent tributaries is calculated according to rational method. The modified rational method is then used to determine Q, Tc and I for the confluence.

The rational method is used for the next point of interest along the flow direction.

Q and Tc, for each independent tributaries are ranked in order of increasing Tc. Let Q1, T1, and I1 correspond to the tributary area with the shortest Tc. Likewise, Q2, T2, and I2 correspond to the tributary area with the next longer Tc; and so on. The tributaries are combined as follows.

For T1 < T2 < T3,

QT1 = Q1 +

$$Q_{T1} = Q_1 + \frac{T_1}{T_2} Q_2 + \frac{T_1}{T_3} Q_3$$

$$Q_{T2} = Q_2 + \frac{I_2}{I_1} Q_1 + \frac{T_2}{T_3} Q_3 \qquad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots.. (38)$$

$$Q_{T3} = Q_3 + \frac{I_3}{I_1} Q_1 + \frac{I_3}{I_2} Q_2$$

The largest of QT1, QT2 and so on and the associated Tc should be used for the junction. The equation can be expanded for more than three tributaries. InfoSewer can handle unlimited number of tributaries.

Runoff Coefficient

Runoff coefficient is loosely defined as the ratio of runoff to rainfall, and is a function of watershed characteristics including land use, soil type, and slope of the watershed. The value of runoff coefficient ranges between 0.0 and 1.0. A value of 0.0 means that all of the rainfall is lost in the form of abstractions such as infiltration, interception, and evaporation and none of the rainfall is converted to runoff. The value of 1.0 implies that all the rainfall is converted to runoff and is discharged from the watershed. As an example, most of the rain that falls on impervious areas such as pavement and roof would be immediately converted to runoff. A value of C for such land uses is close to 1.0. Runoff coefficient values recommended by the American Society of Civil Engineers and Water Environment Federation for return periods not exceeding 10 years are given below for various land uses, soil types, and slope conditions.

DESCRIPTION OF AREA	RUNOFF COEFFICIENT
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	
Pavement	
Asphaltic and concrete	0.70 - 0.95
Brick	0.70 - 0.85
Roofs	0.75 - 0.95
Lawns, sandy soil	
Flat, 2%	0.05 - 0.10
Average, 2-7 %	0.10 - 0.15
Steep, 7%	0.15 - 0.20
Lawns, heavy soil	
Flat, 2%	0.13 - 0.17
Average, 2-7 %	0.18 - 0.22
Steep, 7%	0.25 - 0.35

For return periods that exceed 10 years, the runoff coefficient from the table should be multiplied by a frequency adjustment factor, Cf , given below.

Frequency (Years)	C_{fi}
25	1.1
50	1.2
100	1.25

For a subwatershed composed of multiple land uses, a composite runoff coefficient Cw should be determined by weighting C values of each of the land uses by their corresponding area according to Equation 35.

$$C_w = \sum_{i=1}^{N}(C_i A_i) \Big/ \sum_{i=1}^{N}(A_i)$$

where C_i = runoff coefficient for individual land use in the subwatershed.

A_i = area of the individual land use in the subwatershed.

N = total number of land uses in the subwatershed.

Stormwater Runoff and the Rational Method from Innovyze H_2OCalc for Reference

For storm sewer loading, the focus shifts to hydrologic analysis of excess precipitation and associated runoff. Common techniques for analysis include the rational method and unit hydrograph methods, as well as the use of more advanced hydrologic models.

For small drainage areas, peak runoff is commonly estimated by the rational method. This method is based on the principle that the maximum rate of runoff from a drainage basin occurs when all parts of the watershed contribute to flow and that rainfall is distributed uniformly over the catchment area. Since it neglects temporal and spatial variability in rainfall, and ignores flow routing in the watershed, collection system, and any storage facilities, the rational method should be used with caution only for applications where the assumptions of rational method are valid.

Rational Method from Innovyze H_2OCalc for Reference

The rational formula is expressed as

$$Q_p = KCiA$$

where Q_p = peak runoff rate (m^3/s, ft^3/s)

C = dimensionless runoff coefficient (see Table 3-9)

I = average rainfall intensity (mm/hr, in/hr) for a duration of the time of concentration (t_c)

A = drainage area (km^2, acres)

K = conversion constant (0.28 in SI, 1 in English)

The time of concentration t_c used in the rational method is the time associated with the peak runoff from the watershed to the point of interest. Runoff from a watershed usually reaches a peak at the time when the entire watershed is contributing; in this case, the time of concentration is the time for a drop of water to flow from the remotest point in the watershed to the point of interest. Time of concentration, t_c (min), for the basin area can be computed using one of the formulas listed in Table 3-10.

Table: Runoff Coefficients for 2 to 10 Year Return Periods

Description of drainage area		Runoff coefficie
Business	Downtown	0.70-0.95
	Neighborhood	0.50-0.70
Residential	Single-family	0.30-0.50
	Multi-unit detached	0.40-0.60
	Multi-unit attached	0.60-0.75
Suburban		0.25-0.40
Apartment dwelling		0.50-0.70

Industrial	Light		0.50-0.80
	Heavy		0.60-0.90
Parks and cemeteries			0.10-0.25
Railroad yards			0.20-0.35
Unimproved areas			0.10-0.30
Pavement	Asphalt		0.70-0.95
	Concrete		0.80-0.95
	Brick		0.75-0.85
Roofs			0.75-0.95
Lawns	Sandy soils	Flat (2%)	0.05-0.10
		Average (2-7%)	0.10-0.15
		Steep (≥7%)	0.15-0.20
	Heavy soils	Flat (2%)	0.13-0.17
		Average (2-7%)	0.18-0.22
		Steep (≥7%)	0.25-0.35

Source: Nicklow et al. (2006)

Table 3-10: Formulas for Computing Time of Concentration

Method	**Formula**	
Kirpich (1940)	$t_c = 0.0078L^{0.77}S^{-0.385}$ L = length of channel (ft) S = average watershed slope (ft/ft)	
California Culverts Practice (1942)	$t_c = 60\left(11.9L^3/H\right)^{0.385}$ L = length of the longest channel (mi) H = elevation difference between divide and outlet (ft)	
Izzard (1946)	$t_c = \dfrac{41.025(0.0007i + c)L^{1/3}}{S^{1/3}i^{2/3}}$ i = rainfall intensity (in/h) c = Retardance coefficient	Retardance factor, c, ranges from 0.007 for smooth pavement to 0.012 for concrete and to 0.06 for dense turf; product i times L should be < 500
Federal Aviation Administration (1970)	$t_c = \dfrac{0.39(1.1 - C)L^{1/2}}{S^{1/3}}$	

	C = rational method runoff coefficient (see Table 3.9)	
Kinematic wave	$$t_c = \frac{0.938 L^{0.6} n^{0.6}}{i^{0.4} S^{0.3}}$$ n = Manning's roughness coefficient	
SCS lag equation	$$t_c = \frac{100 L^{0.8} \left[(1000/CN) - 9\right]^{0.7}}{19000 S^{1/2}}$$ CN = SCS runoff curve number (see Table 3.11)	
SCS average velocity charts	$$t_c = \frac{1}{60} \sum_{j=1}^{N} \left(L_j / V_j\right)$$ V = average velocity (ft/s)	
Yen and Chow (1983)	$$t_c = K_Y \left(\frac{NL}{S^{1/2}}\right)^{0.6}$$ K_Y = Coefficient N = Overland texture factor (see Table 3.13)	K_Y ranges from 1.5 for light rain (i<0.8) to 1.1 for moderate rain (0.8<i<1.2), and to 0.7 for heavy rain (i>1.2)

Source: Nicklow et al. (2004)

Table 3-11: Runoff Curve Numbers for Urban Land Uses

Land use description	**Soil Group**			
	A	B	C	D
Lawns, open spaces, parks, golf courses:				
Good condition: grass cover on 75% or more area	39	61	74	80
Fair condition: grass cover on 50% to 75% of area	49	69	79	84
Poor condition: grass cover on 50% or less of area	68	79	86	89
Paved parking lots, roofs, driveways, etc	98	98	98	98
Streets and roads:				
Paved with curbs and storm sewers	98	98	98	98
Gravel	76	85	89	91
Dirt	72	82	87	89
Paved with open ditches	83	89	92	93

Commercial and business areas (85% impervious)	89	92	94	95
Industrial districts (72% impervious)	81	88	91	93
Row houses, town houses and residential with lot sizes of 1/8 ac or less (65% impervious)	77	85	90	92
Residential average lot size:				
1/4 ac (38% impervious)	61	75	83	87
1/3 ac (30% impervious)	57	72	81	86
1/2 ac (25% impervious)	54	70	80	85
1 ac (20% impervious)	51	68	79	84
2 ac (12% impervious)	46	65	77	82
Developing urban area (newly graded; no vegetation)	77	86	91	94

Adapted from SCS (1985)

Table 3-12: Description of NRCS Soil Classifications

Group	Description	Min. infiltration (in/hr)
A	Deep sand; deep losses; aggregated silts	0.30-0.45
B	Shallow loess; sandy loam	0.15-0.30
C	Clay loams; shallows sandy loam; soils low in organic content; soils usually high in clay	0.05-0.15
D	Soils that swell significantly	0-0.05

Adapted from SCS (1985)

Table 3-13: Overland Texture Factor *N*

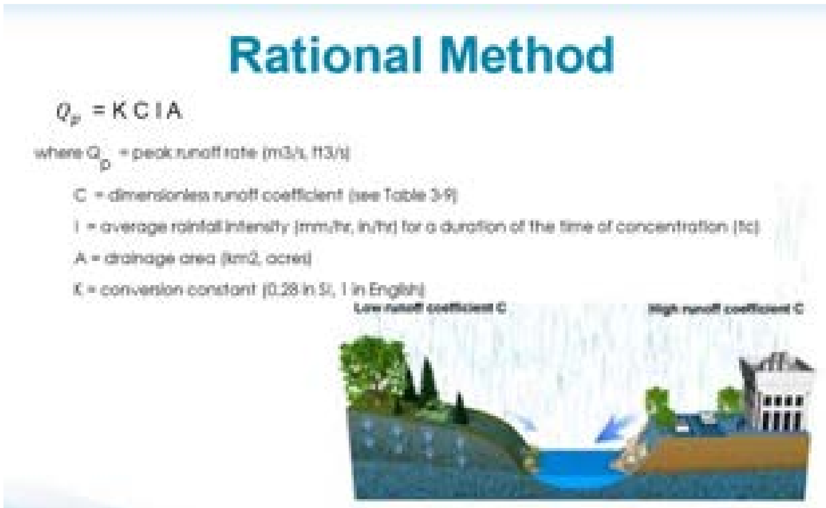
Overland flow surface	Low	Medium	High
Smooth asphalt pavement	0.010	0.012	0.015
Smooth impervious surface	0.011	0.013	0.015
Tar and sand pavement	0.012	0.014	0.016
Concrete pavement	0.014	0.017	0.020
Rough impervious surface	0.015	0.019	0.023
Smooth bare packed soil	0.017	0.021	0.025
Moderate bare packed soil	0.025	0.030	0.035

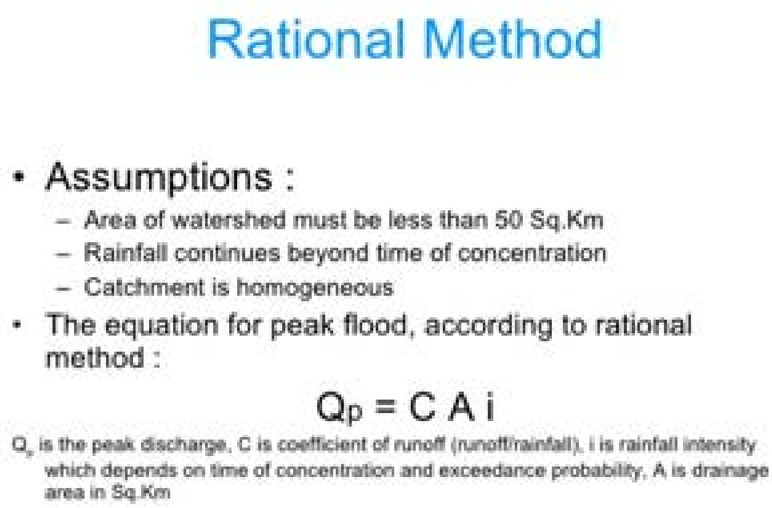
Rough bare packed soil	0.032	0.038	0.045
Gravel soil	0.025	0.032	0.045
Mowed poor grass	0.030	0.038	0.045
Average grass, closely clipped sod	0.040	0.055	0.070
Pasture	0.040	0.055	0.070
Timberland	0.060	0.090	0.120
Dense grass	0.060	0.090	0.120
Shrubs and bushes	0.080	0.120	0.180
Land use	Low	Medium	High
Business	0.014	0.022	0.35
Semi-business	0.022	0.035	0.050
Industrial	0.020	0.035	0.050
Dense residential	0.025	0.040	0.060
Suburban residential	0.030	0.055	0.080
Parks and lawns	0.040	0.075	0.120

Adapted from Yen and Chow (1983)

Assumptions of the Rational Method

For credible engineering application, the modeler needs to understand the following basic assumptions of the rational method. Firstly, the rainfall intensity is constant throughout the watershed over a period of time that equals the time of concentration of the analysis site. Secondly, the runoff coefficient is invariant, regardless of season of the year or intensity of rainfall. Finally, the Area of the contributing watershed is small (not more than 300 acres).





Ground Cover	C	
	Low	High
Lawns	0.05	0.35
Forest	0.05	0.25
Cultivated land	0.08	0.41
Meadow	0.10	0.50
Parks, cemeteries	0.10	0.25
Unimproved areas	0.10	0.30
Pasture	0.12	0.62
Residential areas	0.30	0.75
Business areas	0.50	0.95
Industrial areas	0.50	0.90
Streets		
bricks	0.70	0.85
asphalt	0.70	0.95
concrete	0.70	0.95
Roofs	0.75	0.95

Infiltration Losses

During storm events, some of the rainfall is lost in the form of infiltration and depression/retention storage depending on soil type, land use, and topographic conditions of the modeled catchment. InfoSewer Pro estimates part of rainfall that is lost in the form of infiltration and depression/retention storage, and uses the resulting effective (excess) rainfall to determine runoff hydrograph.

The model applies the well known and the widely used Horton's method to determine infiltration loss through pervious portions of the subwatersheds. Horton's infiltration equation is given as:

$$f_t = f_\infty + (f_i - f_\infty)e^{-\alpha t}$$

where, f_t = infiltration rate at time t, in inches/hour.

f_i= maximum or initial infiltration rate (at t = 0), in inches/hour. The default value is 3

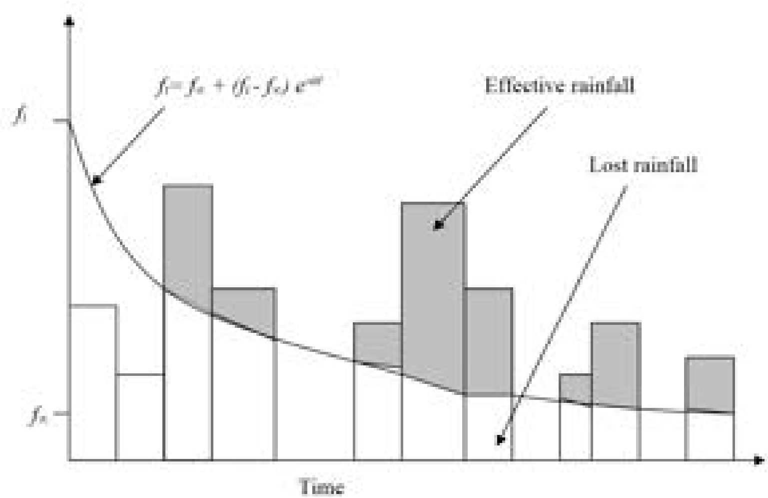
inches/hour.

f_∞ = final infiltration rate (at t = ∞), in inches/hour. The default value is 0.5 inches/hour.

α = decay coefficient, in per second. The default value is 0.001/second.

t = time from beginning of storm, in seconds.

The actual infiltration at time t during the rainfall event(s) is the lesser of rainfall intensity and rate of infiltration, ft. The following figure describes Horton's infiltration model graphically. Only the shaded portion of the rainfall intensities, the region above the infiltration curve, becomes runoff. The remaining rainfall is lost in the form of infiltration.



Infiltration Regeneration

Infiltration capacity can recover (regenerate) during dry periods. InfoSewer/Pro models infiltration regeneration whenever there is dry time steps (when there is no precipitation for the catchment) using the following drying curve. Modeling of infiltration regeneration enables the user to account for the effects of antecedent precipitation on rainfall abstractions, thus enhancing simulation accuracy.

$$f_t = f_i - (f_i - f_\infty)e^{-\alpha_d (t - t_w)}$$

where, f_t = infiltration rate at time t, in inches/hour.

f_i = maximum or initial infiltration rate (at t = 0), in inches/hour. The default value is 3 inches/hour.

f_∞ = final infiltration rate (at t = ∞), in inches/hour. The default value is 0.5 inches/hour.

α_d = decay coefficient for the recovery curve, in per second. It is generally considered to be less than α, implying a longer drying curve than wetting curve.

t = time from beginning of storm, in seconds.

t_w = hypothetical projected time, internally computed by the model.

Depression/Retention Loss

In addition to infiltration losses, InfoSewer Pro can also model losses due to depression/retention storage. The model allows the user to input separate depression storage loss values for the pervious and for the impervious areas of the catchment. The model internally derives representative depression storage loss value for the catchment by weighting the user input depression storage loss values by respective areas of the pervious and the impervious lands in the catchment.

Generation of the Unit Hydrograph

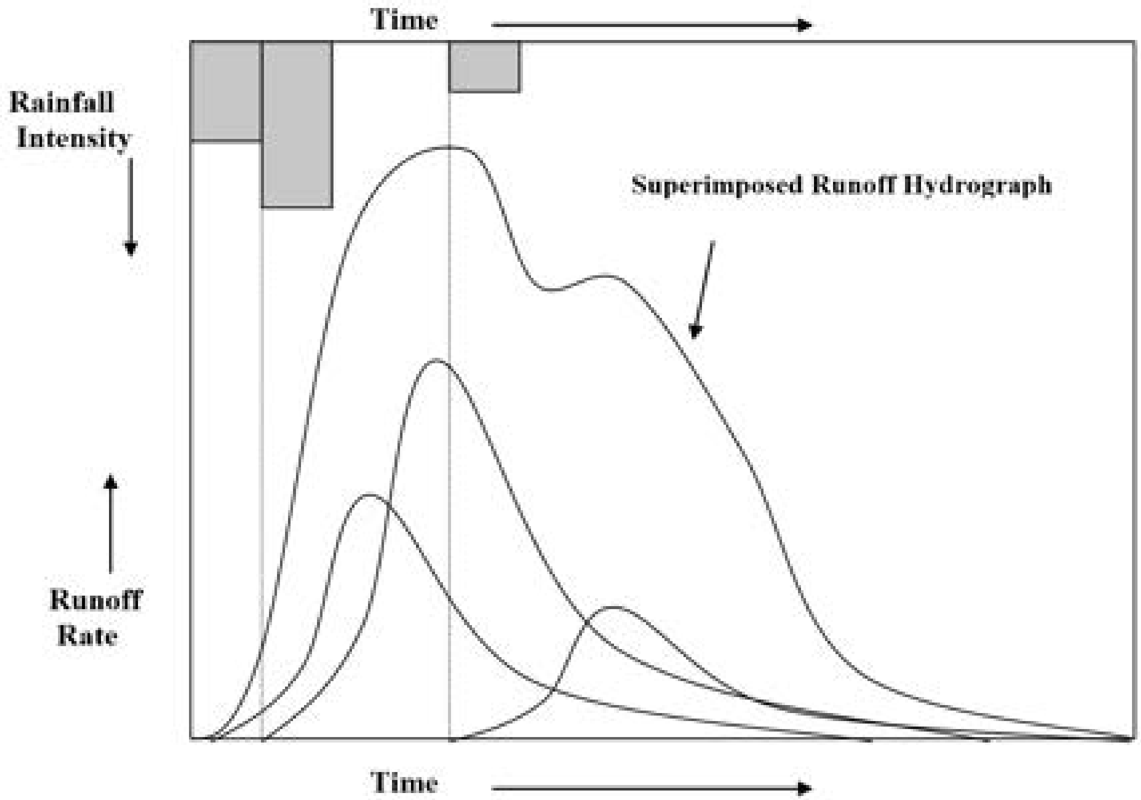
InfoSewer Pro generates a direct runoff hydrograph resulting from single or multiple storm events using a unit hydrograph synthesized according to any of the three techniques described above, or from a user-specified natural unit hydrograph. The model relies on a user input hyetograph (i.e., rainfall intensity versus time graph). The model; can derive effective rainfall from user supplied total rainfall hyetographs.

Unit hydrographs could be natural or synthetic. Natural unit hydrographs are derived from observed data, whereas synthetic hydrographs are generated following empirical techniques based on watershed parameters and storm characteristics to simulate the natural unit hydrograph.

If a natural hydrograph is available for a watershed, InfoSewer/Pro can utilize it to generate storm runoff at different locations in the collection system. In the absence of a natural unit hydrograph, the model can derive a synthetic unit hydrograph using four different techniques. Each of the unit hydrograph synthesizing approaches is described below.

An entire watershed of a collection system may not be well represented by a single unit hydrograph owing to variability in topography, land use, and soil characteristics of the subwatersheds. In the case of separate sewer systems, the magnitude and type of sources of infiltration/inflow will vary by subbasin. Accordingly, InfoSewer Pro allows usage of multiple unit hydrographs, each representing part of the watershed being modeled. The unit hydrographs could be of any duration (e.g., 15-minute, 1-hr), but the duration must be the same for the entire unit hydrographs involved in modeling of the watershed.

To derive the storm hydrographs, InfoSewer Pro applies the basic assumptions of unit hydrograph theory described above (i.e., linearity, time invariance, and the principle of superposition). Storm events are assumed to have constant intensity over the duration of the unit hydrograph. Excess rainfall resulting from single or multiple storm events is discretized at intervals of unit hydrograph duration. For example, if a sewer collection system is being modeled using a 15-minute unit hydrograph, and if duration of the excess rainfall under investigation is 1-hour, the rainfall duration will be divided into four 15-minute rainfall events of constant intensity. This discretization approach, along with the unit hydrograph assumptions of linearity, time invariance, and superposition, enables InfoSewer Pro to simulate storm runoff hydrograph at any location (e.g., loading manhole) throughout the collection system for any number of storm events. During generation of runoff hydrographs for locations other than the site where the unit hydrograph is originally derived, ordinates of the unit hydrographs are adjusted according to the ratio of drainage area of the two locations. Once the storm hydrographs for every loading manhole in the collection system are known, the storm load will be added to other loading types such as sanitary loads, if any, and will be routed through the collection system using the powerful Muskingum-Cunge's dynamic flow routing algorithm.



NRCS (SCS) DIMENSIONLESS UNIT HYDROGRAPH METHOD

The SCS dimensionless unit hydrograph, graphically descried below, is widely used in practice. To generate a tr-hour unit hydrograph for a watershed, time to peak (Tp) and the peak flow rate (Qp) are determined using watershed characteristics.

$$T_p = \frac{t_r}{2} + t_l$$

where t_r is duration of effective rainfall, and tl is lag time of the watershed. Lag time represents the time from the center of mass of effective rainfall to the time to peak of a unit hydrograph. In other words, lag time is a delay in time, after a brief rain over a watershed, before the runoff reaches its peak. The lag time can either be specified by the user, or can be calculated by the model using the following SCS equation (USDA 1986).

$$t_l = L^{0.8}\left(\frac{1000}{CN} - 9\right)^{0.7} \Big/ \left(19000 S^{0.5}\right)$$

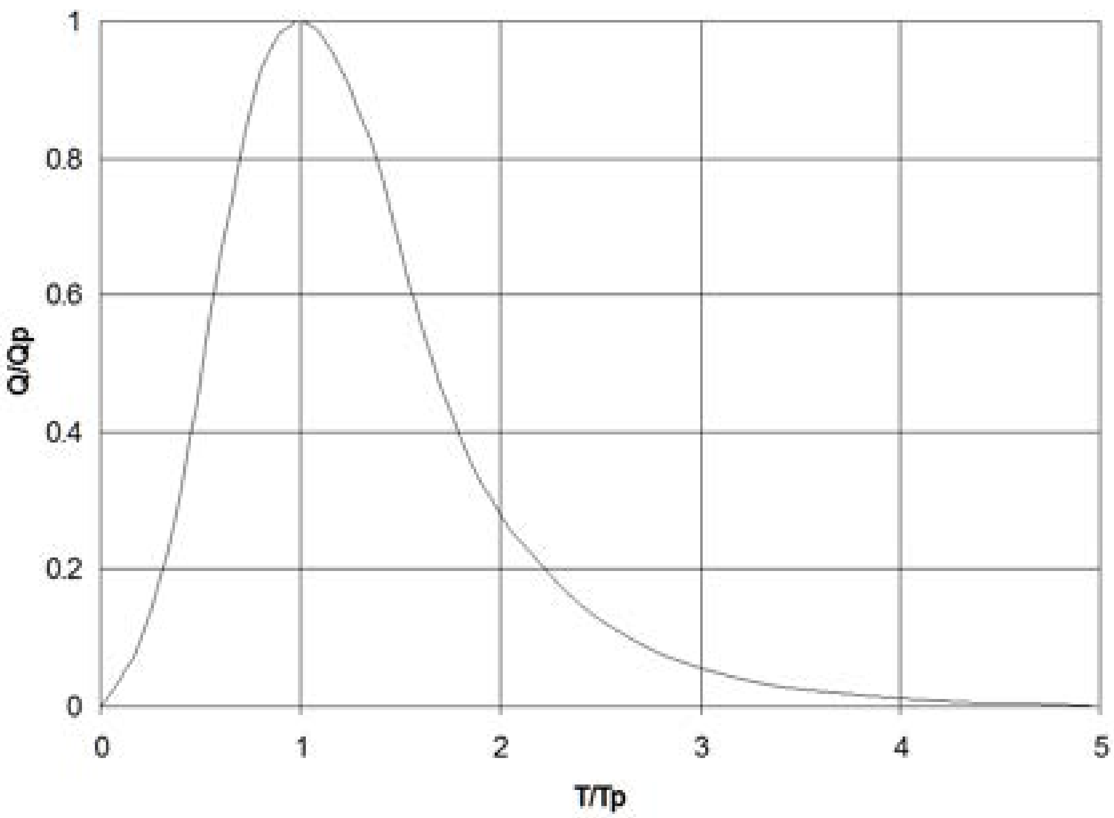
where

t_l = lag time of the watershed in hours.

L = hydraulic length of the watershed in ft. This refers to travel distance of water from the most upstream location of the watershed to the point where the unit hydrograph is derived.

CN = the SCS curve number. This is a measure of runoff generating capacity of a watershed, and it depends on the soil, the antecedent moisture condition, the cover, and the hydrologic conditions of the watershed. Recommended CN values are given in the following table for urban areas (USDA 1986). The SCS suggests the CN values for the above equation to be within 50 and 95.

S = average slope of the watershed.



Q/Qp
1
0.8
0.6
0.4
0.2
0
0
1
2
3
4
5
T/Tp

COVER DESCRIPTION		CURVE NUMBER FOR HYDROLOGIC SOIL GROUP			
Cover Type and Hydrologic Condition	**Average Percent Impervious Area**	**A**	**B**	**C**	**D**
Fully Developed Urban Areas (Vegetation Established)					
Open Space (lawns, parks, golf courses, cemeteries, etc.):					
Poor Condition (grass cover < 50%)		68	79	86	89
Fair Condition (grass cover 50% to 75%)		49	69	79	84
Good condition (grass cover > 75%)		39	61	74	80
Impervious areas:					
Paved parking lots, roofs, driveways, etc. (excluding right-of-way)		98	98	98	98
Streets and roads					
Paved; curbs, and storm sewers (excluding right-of-way)		98	98	98	98
Paved; open ditches (including right-of-way)		83	89	92	93
Gravel (including right-of-way)		76	85	89	91
Dirt (including right-of-way)		72	82	87	89
Western desert urban areas:					
Natural desert landscaping (pervious areas only)		63	77	85	88
Artificial desert landscaping (impervious weed barrier, desert shrub with I- to 2-inch sand or gravel mulch and basin borders)		96	96	96	96
Urban districts:					
Commercial and business	85	89	92	94	95
Industrial	72	81	88	91	93
Residential districts by average lot size:					
1/8 acre or less (town house)	65	77	85	90	92
1/4 acre	38	61	75	83	87
1/3 acre	30	57	72	81	86
1/2 acre	25	54	70	80	85
1 acre	20	51	68	79	84
2 acres	12	46	65	77	82
Developing urban areas:					
Newly graded areas (pervious areas only, no vegetation)		77	86	91	94

The peak flow rate is calculated as:

$$Q_p = \frac{484A}{T_p}$$

where

Q_p = peak flow rate in cfs.

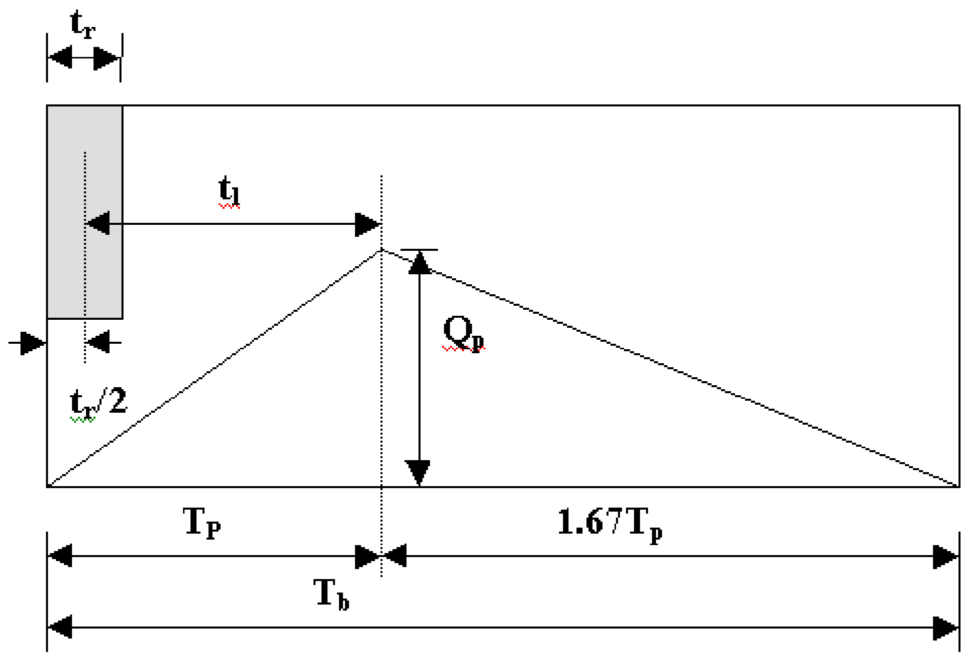
A = area of the watershed, in square miles, draining to the location of the unit hydrograph.

T_p = time to peak of the unit hydrograph in hours.

Once Tp and Qp are known, actual time and flow rate ordinates of the tr-hour unit hydrograph are determined by multiplying the dimensionless time (T/Tp) and the dimensionless flow rate ordinates (Q/Qp) by Tp and Qp, respectively.

NRCS (SCS) TRIANGULAR UNIT HYDROGRAPH METHOD

The SCS has also developed a triangular unit hydrograph (USDA 1986) (see figure below) that is an approximation to the dimensionless unit hydrograph described above. The triangular unit hydrograph is entirely defined in terms of three points, Qp, Tp, and Tb. The lag time, time to peak, and peak flow rate are calculated using the same equations as for the dimensionless unit hydrograph.

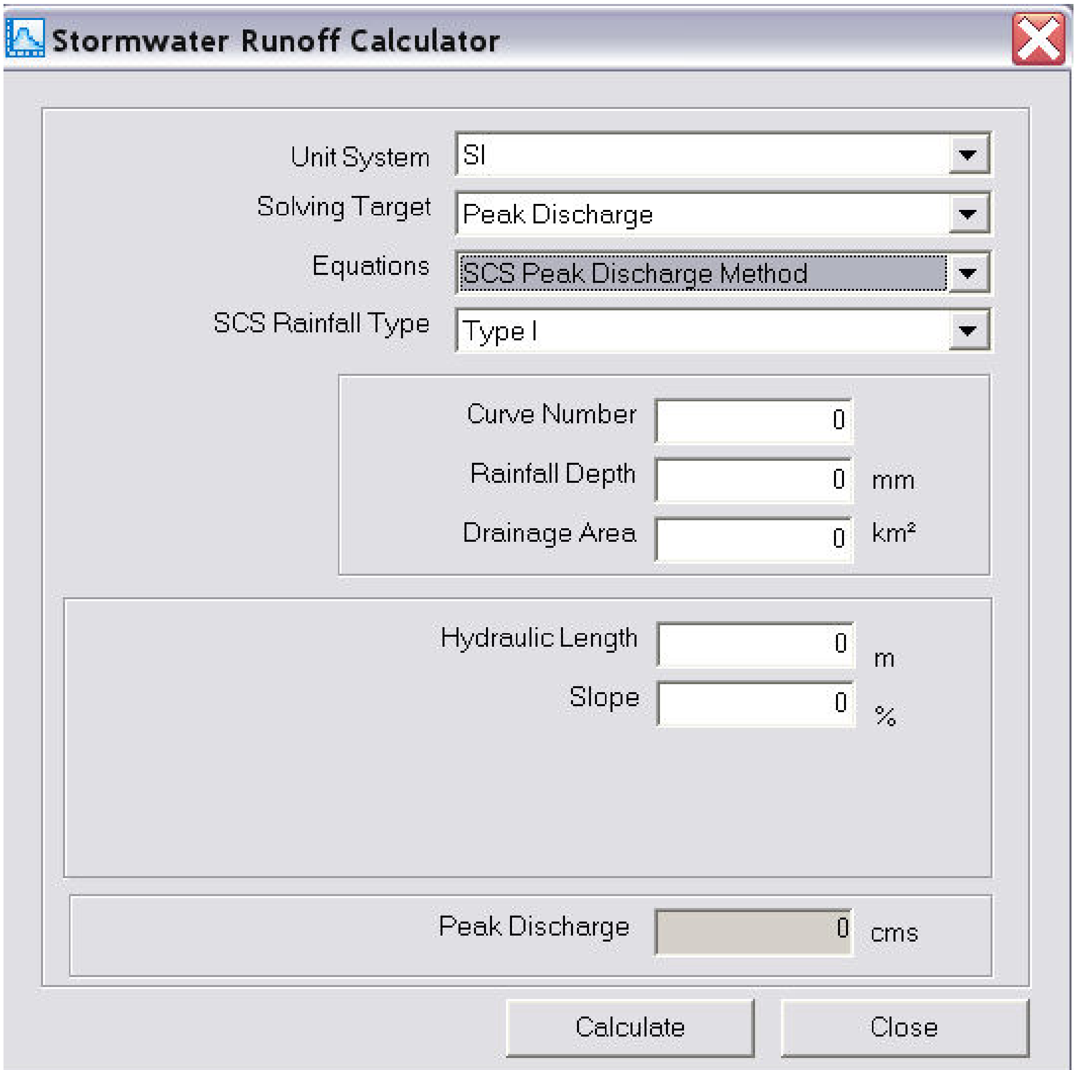


NRCS Dimensionless UH, NRCS Triangular UH from Innovyze H_2OCalc for Reference

- **Subcatchment Area** – Drainage area of the watershed.
- **Subcatchment Slope**– Average slope of the watershed.
- **Flow Length** – Length of the longest overland flow path for the watershed.
- **NRCS Curve Number** – Value of CN for the watershed. See Table 3.11. It is a function of soil type, land use type and quality, and antecedent moisture condition.
- **Time of Concentration** – The time it takes for flow to travel from the hydraulically remotest point in the watershed to get to outlet of the watershed. If time of concentration is defined, slope, length, and CN values will not be required.
- **Rainfall Depth** – Depth of the design rainfall.

Peak Discharge: SCS Peak Discharge Method from Innovyze H_2OCalc for Reference

The dialog box for the SCS peak discharge method is shown below. Click here for the methodology.



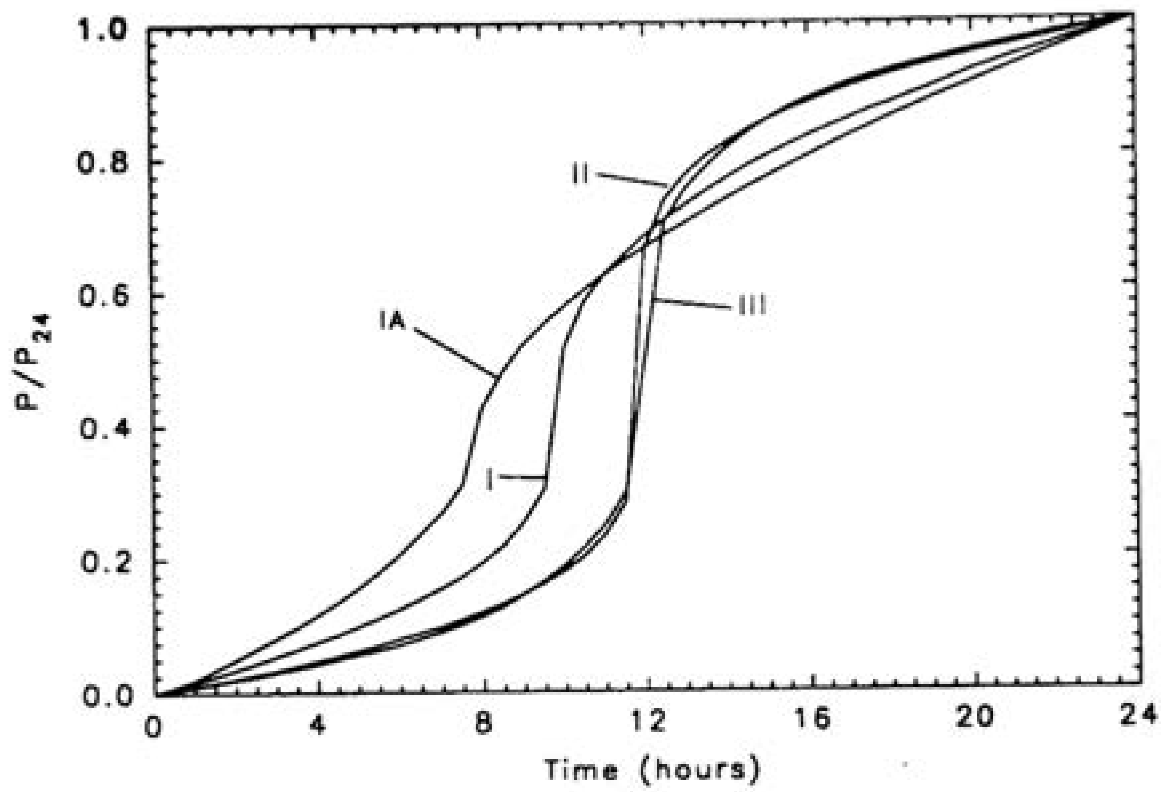
Input for the SCS peak discharge method:

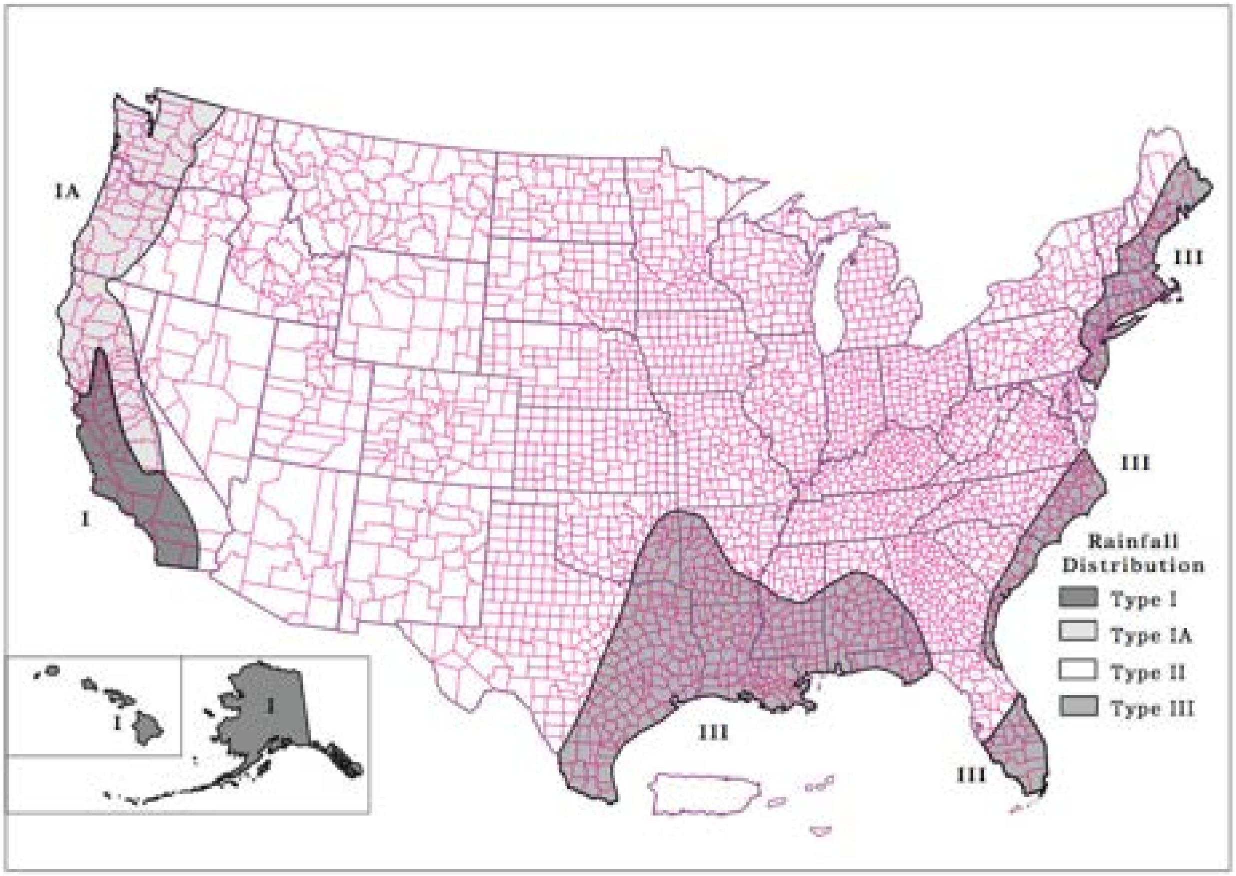
- **Unit System** – English or SI unit.
- **Solving Target** – Peak discharge or time of concentration.
- **Equations** – Rational method or the SCS peak discharge method.
- **SCS Rainfall Type** – Select one of the SCS rainfall types (i.e., Types I, IA, II, or III)

- **Curve Number** – SCS's dimensionless number that is used as a measure of runoff generation capability of a watershed (see Table 3.12). It is a function of soil, land cover and treatment.
- **Rainfall Depth** – 24-hr cumulative design rainfall depth. This rainfall depth will be distributed across the 24-hr duration according to the SCS rainfall type selected.
- **Drainage Area** – Area of the total watershed that drains to the location where the peaks flow is determined.

Output for the SCS peak discharge method:

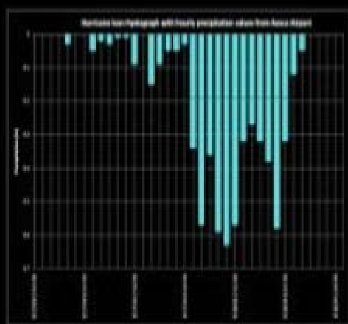
- **Peak Discharge** – Peak flow generated from the watershed.





Source: http://www.slideshare.net/damonweiss/workshop-on-storm-water-modeling-approaches

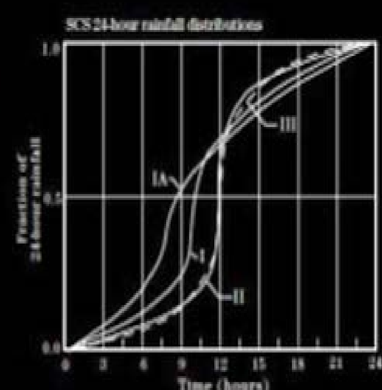
Hydrologic Modeling
Precipitation – Synthetic Rainfall Distributions

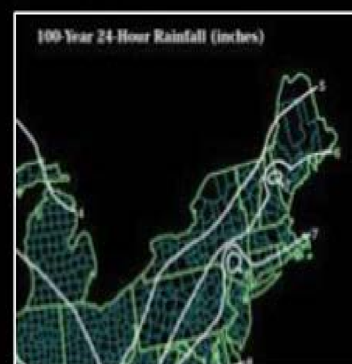


Actual Rainfall Data (Hurricane Ivan)

- SCS Synthetic rainfall distributions are used in lieu of actual storm events
- Type IA are the least intense storms
- Type II (Pittsburgh) are the most intense
- Represent fractional 24-hour rainfall, which translates well to any storm

Formulation of Synthetic Rainfall

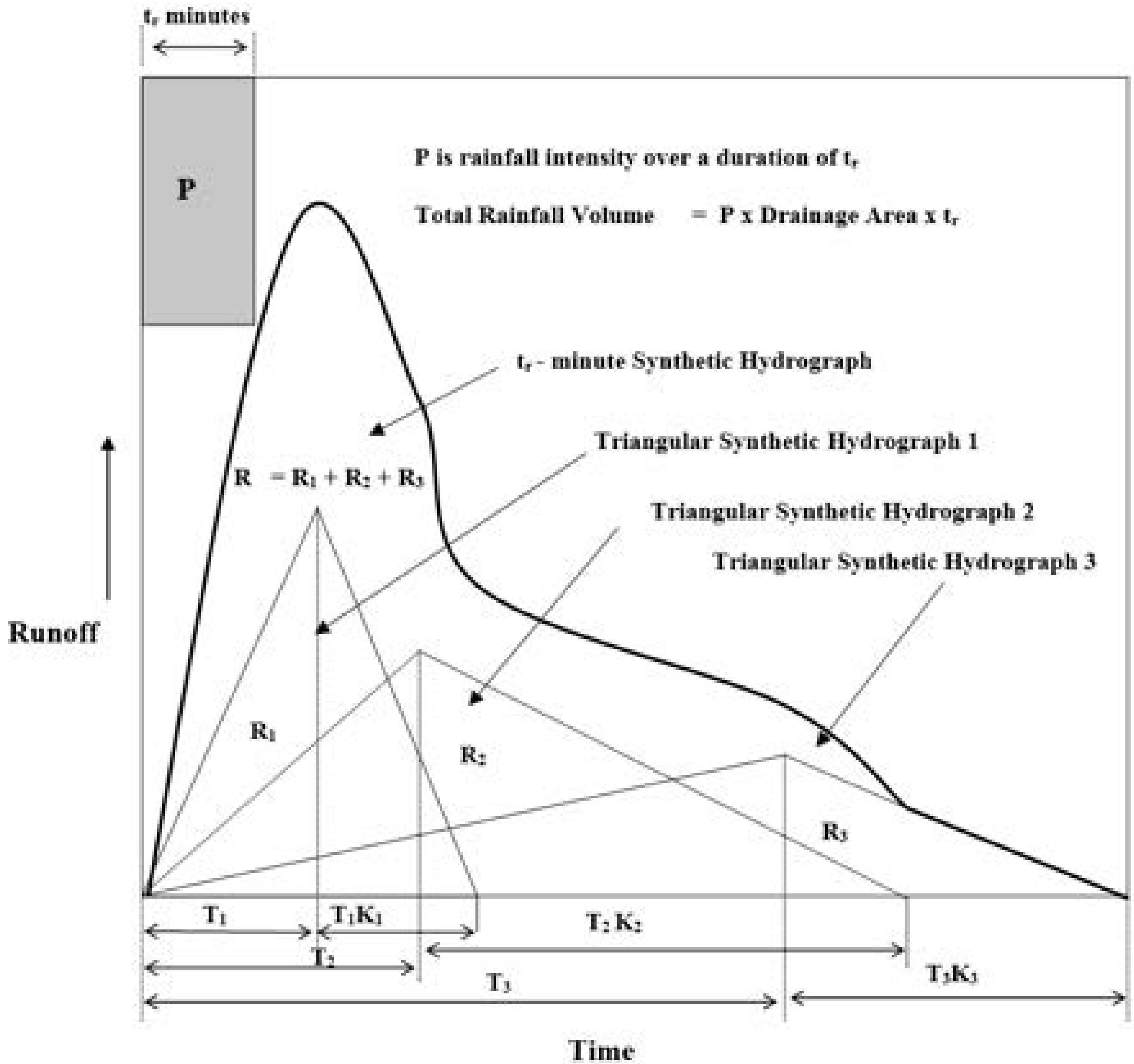




The Tri-Triangle or RTK Method

Tri-triangle is one of the techniques used by InfoSewer/Pro for synthesizing a hydrograph. The triangular unit hydrograph technique developed by the U.S. Soil Conservation Service (now known as Natural Resources Conservation Service NRCS) is one of the commonly used methods. As described above NRCS's triangular synthetic unit method uses a single triangle to represent a unit hydrograph. However, shape of unit hydrograph is too complex to be well captured by a single triangle. InfoSewer/Pro applies up to three triangular synthetic hydrographs, as the name implies, to simulate a hydrograph. The total synthetic hydrograph is the result of aggregating corresponding ordinates of the three triangular hydrographs. Each of these three triangular hydrographs has its own characteristic parameters, namely time to peak, recession constant, and fraction of an effective rainfall volume allocated to the triangle. The technique is graphically illustrated below.

R1, R2, and R3 are fractions of excess rainfall volume, R, allocated to triangular synthetic hydrographs 1, 2, and 3 respectively. Ti and Ki are time to peak and recession constants of the triangles, respectively. The three triangular synthetic hydrographs are conceptual representations of different components of direct runoff or rainfall-dependent infiltration/inflow. The first triangle represents rapidly responding (fast) components, such as contributions from pavements and rooftops, or direct inflow or rapid infiltration into separate sewer systems. The third triangle represents slow runoff components such as ground water contributions or slow infiltration into separate sewers. The second triangle represents runoff or infiltration with a medium time response.



Time to peak value of the first triangle typically varies between 1 and 2 hours, depending on the size of the tributary area in question. The second triangle takes T values ranging from 4 to 8 hours. The third triangle parameter varies greatly depending on the infiltration characteristics of the system being modeled, and has a T value generally between 10 and 24 hours. The default values used for T1, T2, and T3 are 1, 4, and 12 hours respectively. Value of K for the first triangle typically ranges between 2 and 3. The second and the third triangles take K values from 2 to 4. The default values of K are 2, 3, and 3 for the first, the second, and the third triangles, respectively.

RDII or Tri-Triangular Example

The RDII or Rainfall Dependent Infiltration Inflow method in InfoSewer is similar to the RDII or RTK method in InfoSWMM and InfoWorks ICM but with some differences. The RTK data for triangles 1, 2 and 3 are defined in the Unit Hydrograph but instead of individual R values, the overall R is set and the Percent R1, R2 and R3 are defined based on the total R. R3 is calculated internally as 100 – R1 – R2. Each loading manhole with RDII flow has a total area, a hyetograph and a Unit Hydrograph. The hyetograph has to be set at multiples of the unit hydrograph, so you can define the time or X columns with integers and then use the Block Edit command to change X to minutes by multiplying by the Unit Hydrograph time (Figure 1). You can use only one component if you set R1 or R2 to 100 percent or R3 to 100 percent by setting R1 and R2 to 0 percent (Figure 2). The overall area of the Unit Hydrograph is divided amongst the loading manhole using the Subbasin Area (Figure 3). The storm flows generated can be viewed using a Group Graph (Figure 4) and the area of the loading manholes has a relationship to to the total area of the defined unit hydrograph (Figure 5).

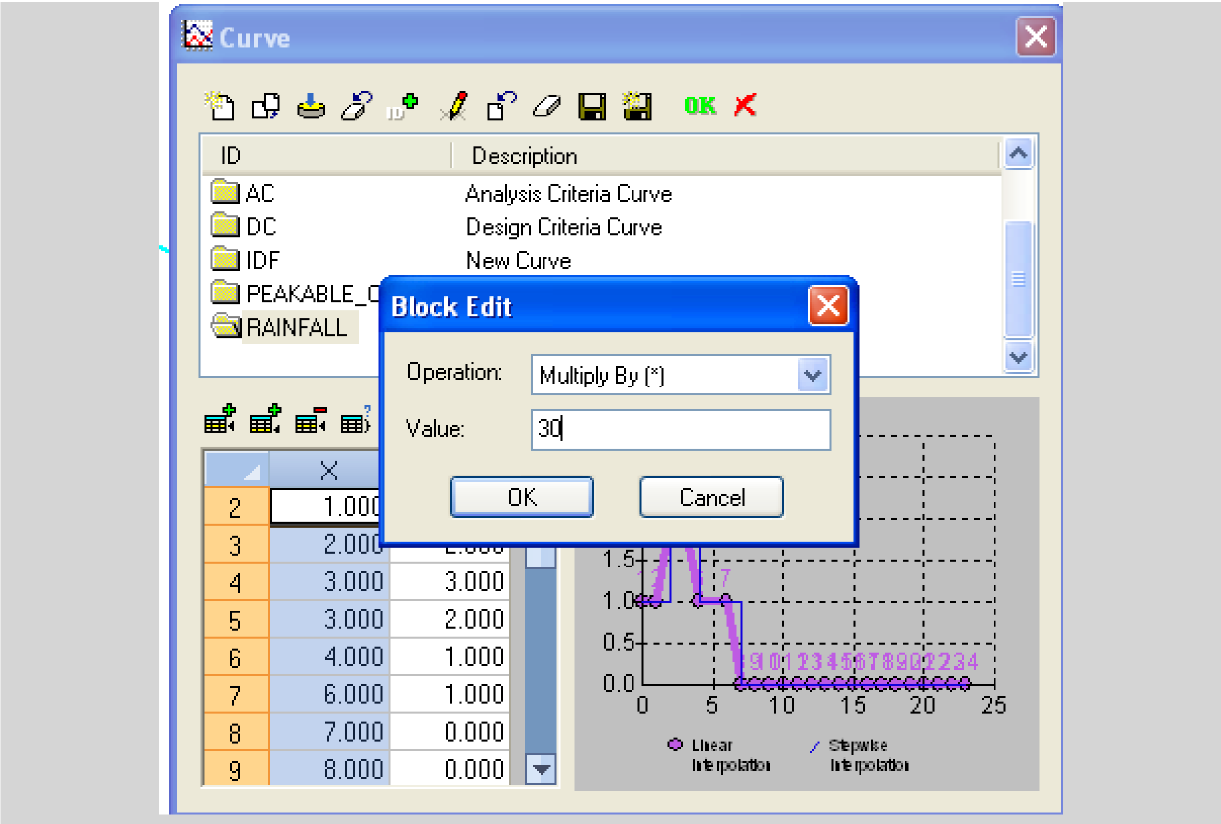


Figure 1. Hyetograph Curve for the RDII Unit Hydrograph

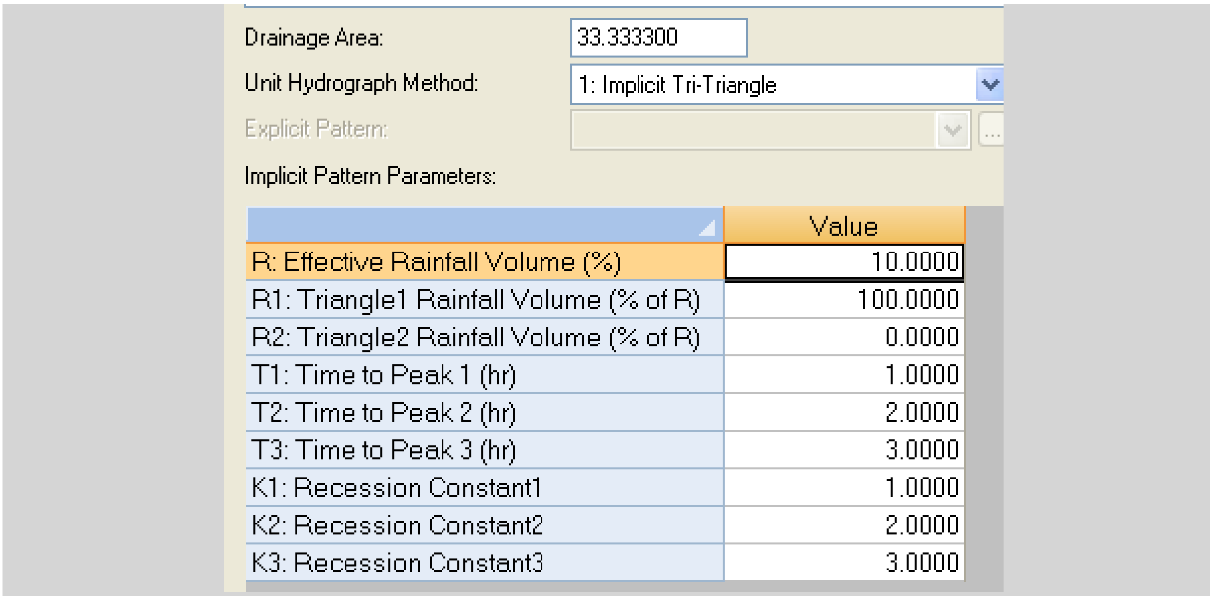
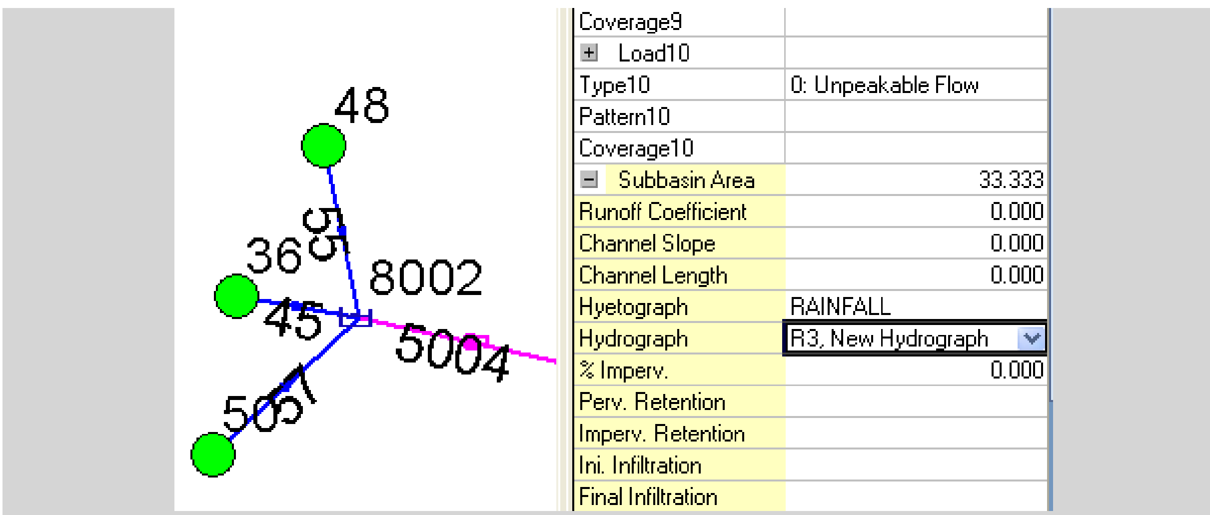


Figure 2. The Unit Hydrograph is defined for various values of R, R1, R2, T1, T2, T3, K1, K2 and K3



Figfure 3. The-Unit-Hydrograph-and-Hyetograph-are-tied-to-a-particular-loading-manhole-using-a-Subbasin-Area

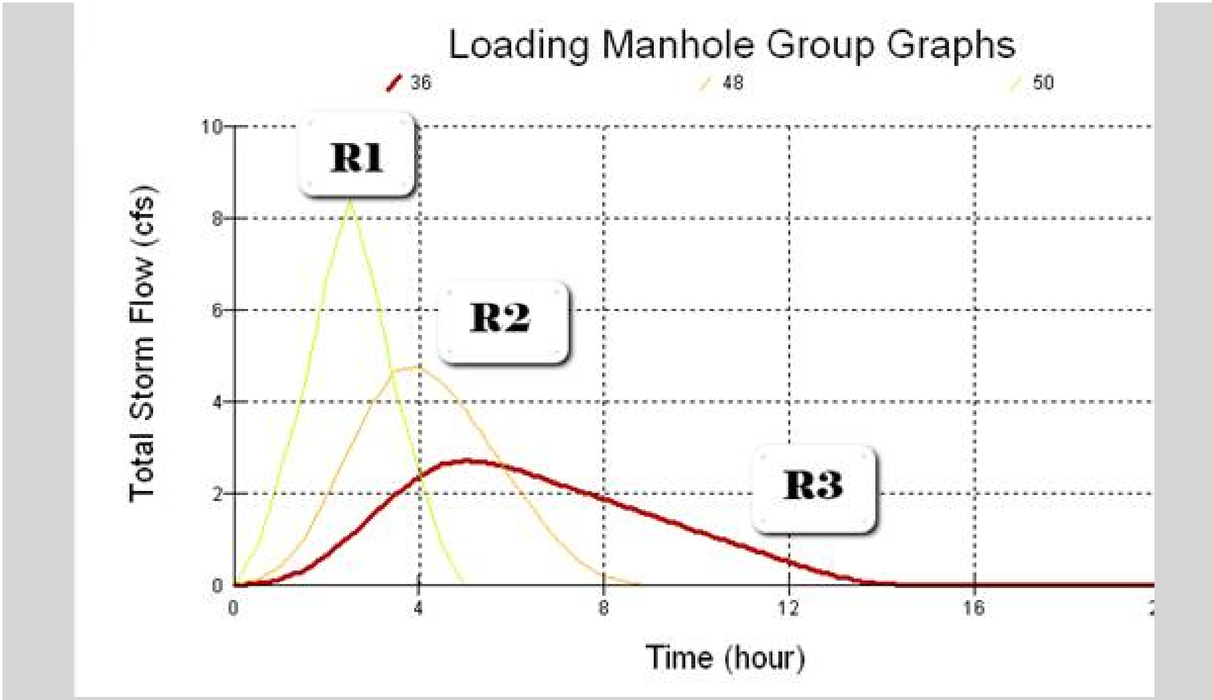
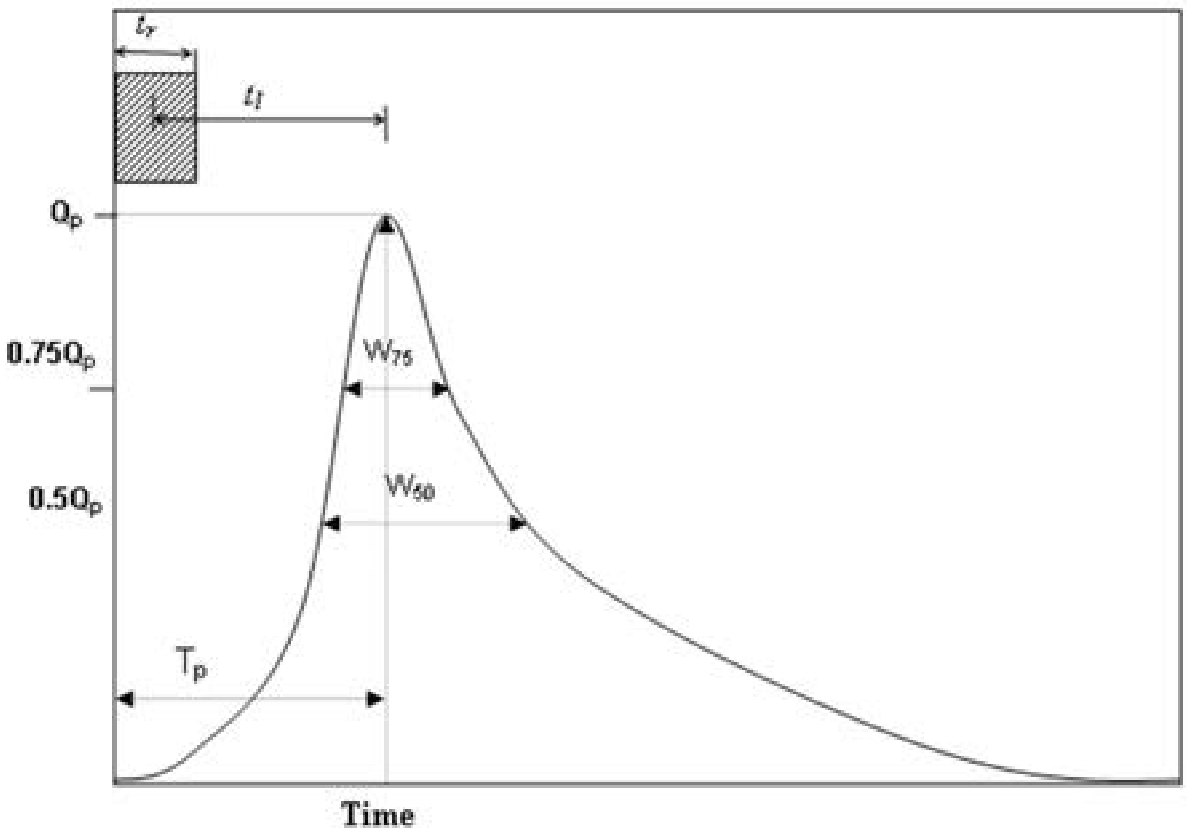


Figure 4. The Unit Hydrographs that are generated can be viewed using a Group loading Manhole Graph. The R1, R2 and R3 have only one triangle

The Colorado Urban Hydrograph Procedure

The Colorado Urban Hydrograph Procedure (CUHP) uses the equations and procedures presented in the Urban Drainage Criteria Manual (USDCM) of the Urban Drainage and Flood Control District (UDFCD). Shape of the CUHP synthetic unit hydrograph is determined using the following equations that relate unit hydrograph parameters to catchment properties.



Lag time (tl) of the watershed (catchment), defined as the time from the center of unit storm duration to the peak of the unit hydrograph, is determined as:

$$t_l = C_t \left(\frac{(L \cdot L_{Ca}}{\sqrt{S}} \right)^{0.48}$$

where t_l = lag time in hours.

L = length along the drainageway path from study point to the most upstream limits of the catchment in miles.

L_{ca} = length along stream from study point to apoint along stream adjacent to the centroid of the catchment in miles.

S = length weighted average slope of catchment along draiangeway path to upstream limits of the catchment.

C_t = time to peak coeficient.

Once the lag time is known, time to peak (Tp) of the unit hydrograph could be determined by adding 0.5tr to the lag time in consistent units.

Peak flow rate, Q_p, of the unit hydrograph is calculated as:

$$Q_p = \frac{640 C_p A}{T_p}$$

where Q_p = peak flow rate of the unit hydrograph, in cfs.

A = area of the catchment, in square miles.

C_p = unit hydrograph peaking coefficient, and is determined as:

$$C_p = P \cdot C_t \cdot A^{0.15}$$

where P = peaking parameter.

C_t and P are defined in terms of percent impervious (Ia) of the catchment as:

$$C_t = aI_a^2 + bI_a + c$$

$$P = dI_a^2 + eI_a + f$$

The coefficients a, b, c, d, e, and f are defined in terms of Ia in the following table.

IA	*A*	*B*	*C*	*D*	*E*	*F*
$Ia \leq 10$	0.0	-0.00371	0.163	0.00245	-0.012	2.16
$10 < Ia \leq 40$	0.000023	-0.00224	0.146	0.00245	-0.012	2.16
$Ia > 40$	0.0000033	-0.000801	0.120	-0.00091	0.228	-2.06

The widths of the unit hydrograph at 50% and 75% of the peak are estimated as:

$$W_{50} = \frac{500}{\left(\frac{Q_p}{A}\right)}$$

$$W_{75} = \frac{260}{\left(\frac{Q_p}{A}\right)}$$

where W_{50} = width of the unit hydrograph at 50% of the peak, in hours.

W_{75} = width of the unit hydrograph at 75% of the peak, in hours.

Q_p = peak flow rate, in cfs.

A = catchment area, in square miles.

It is recommended that a unit hydrograph duration of 5-minute be used for studies that apply the CUHP. The maximum recommended drainage area (catchment size) for any single CUHP unit hydrograph is 5 square miles. Whenever a larger watershed is studied, it needs to be subdivided into subcatchments of 5-square miles or less. For this synthetic unit hydrograph method, the minimum drainage area should be 90 acres. For catchments smaller than 90 acres, other unit hydrograph generation mechanisms should be used.

The Colorado Urban Hydrograph Procedure from Innovyze H_2OCalc for Reference

The Colorado Urban Hydrograph Procedure (CUHP) uses the equations and procedures presented in the Urban Drainage Criteria Manual (USDCM) of the Urban Drainage and Flood Control District (UDFCD 2001). Shape of the CUHP synthetic unit hydrograph is determined using the following equations that relate unit hydrograph parameters to catchment properties.

Lag time (t_l) of the watershed (catchment), defined as the time from the center of unit storm duration to the peak of the unit hydrograph, is determined as:

$$t_l = C_t \left(\frac{(L \cdot L_{Ca}}{\sqrt{S}} \right)^{0.48}$$

where t_l = lag time in hours.

L = length along the drainageway path from study point to the most upstream limits of

the catchment in miles.

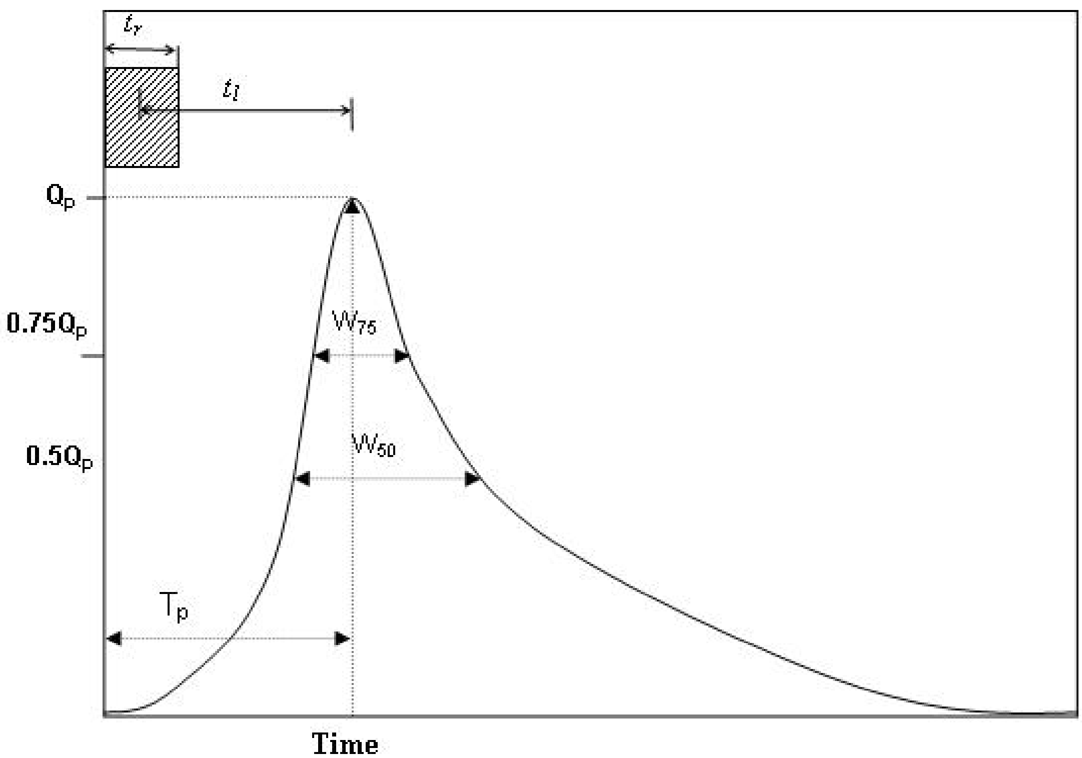
L_{ca} = length along stream from study point to a point along stream adjacent to the centroid

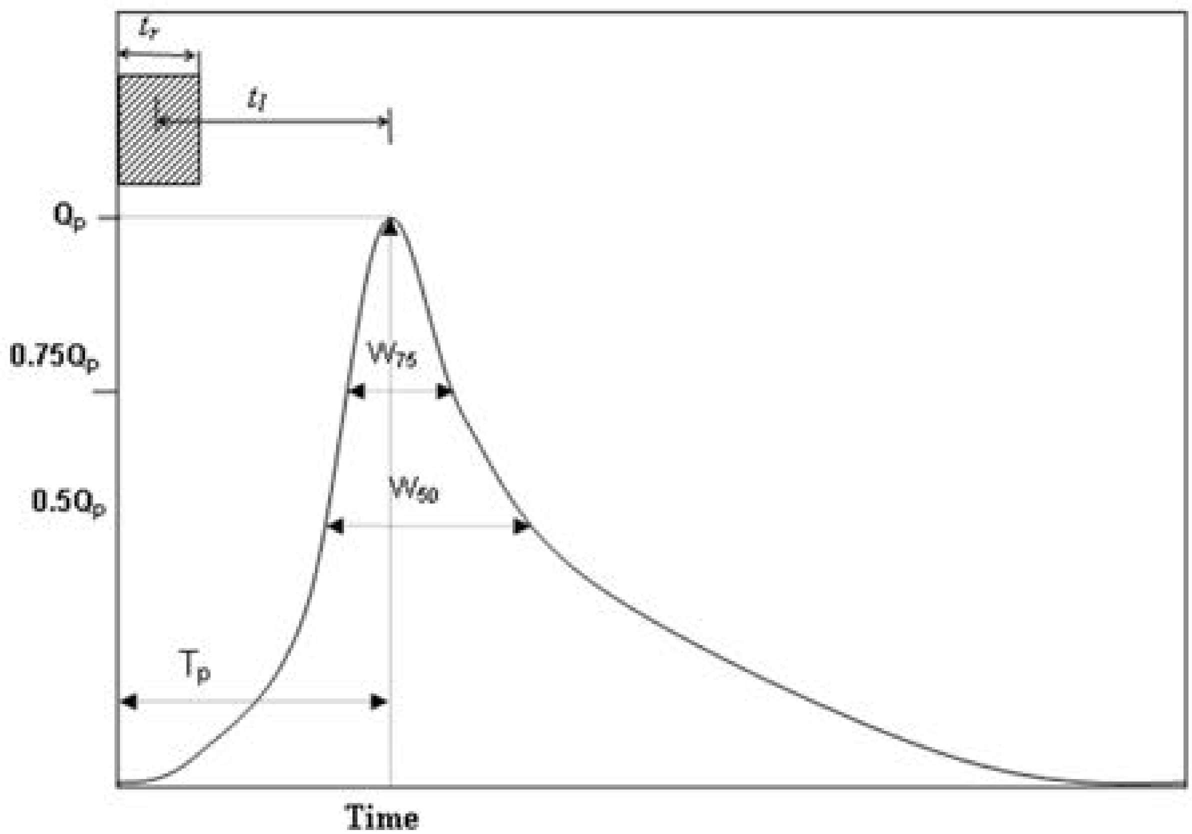
of the catchment in miles.

S = length weighted average slope of catchment along drainageway path to upstream

limits of the catchment.

C_t = time to peak coefficient.





Once the lag time is known, time to peak (T_p) of the unit hydrograph could be determined by adding $0.5t_r$ to the lag time in consistent units.

Peak flow rate, Q_p, of the unit hydrograph is calculated as:

$$Q_p = \frac{640C_pA}{T_p}$$

where Q_p = peak flow rate of the unit hydrograph, in cfs.

A = area of the catchment, in square miles.

C_p = unit hydrograph peaking coefficient, and is determined as:

$$C_p = P \cdot C_t \cdot A^{0.15}$$

where P = peaking parameter.

C_t and P are defined in terms of percent impervious (I_a) of the catchment as:

$$C_t = aI_a^2 + bI_a + c$$

$$P = dI_a^2 + eI_a + f$$

The coefficients *a*, *b*, *c*, *d*, *e*, and *f* are defined in terms of I_a in the following table.

IA	*A*	*B*	*C*	*D*	*E*	*F*
***Ia* ≤ 10**	0.0	-0.00371	0.163	0.00245	-0.012	2.16
***10 < Ia* ≤ 40**	0.000023	-0.00224	0.146	0.00245	-0.012	2.16
***Ia* > 40**	0.0000033	-0.000801	0.120	-0.00091	0.228	-2.06

The widths of the unit hydrograph at 50% and 75% of the peak are estimated as:

$$W_{50} = \frac{500}{\left(\frac{Q_p}{A}\right)}$$

$$W_{75} = \frac{260}{\left(\frac{Q_p}{A}\right)}$$

where W_{50} = width of the unit hydrograph at 50% of the peak, in hours.

W_{75} = width of the unit hydrograph at 75% of the peak, in hours.

Q_p = peak flow rate, in cfs.

A = catchment area, in square miles.

It is recommended that a unit hydrograph duration of 5-minute be used for studies that apply the CUHP. The maximum recommended drainage area (catchment size) for any single CUHP unit hydrograph is 5 square miles. Whenever a larger watershed is studied, it needs to be subdivided into Subcatchments of 5-square miles or less. For this synthetic unit hydrograph method, the minimum drainage area should be 90 acres. For catchments smaller than 90 acres, other unit hydrograph generation mechanisms should be used.

The San Diego Modifed Rational Method (SDMRH)

The San Diego County hydrology manual (San Diego County, 2003) recommends rational method to compute peak storm flow for drainage areas less than 1 square mile. If more than one drainage path is flowing to a junction, the County recommends the modified rational method described below.

The county uses the following relationship to estimate design rainfall intensity (in/hr).

$$I = 7.44 P_6 D^{-0.645}$$

Where: P_6 = 6-hour storm rainfall amount (in).

D = duration (time of concentration) in minutes.

The time of concentration is composed of the initial time of concentration and the travel time.

InfoSewer Pro requires the user to provide initial time of concentration for each subwatershed.

The travel time is computed based on flow velocity and pipe length

The modified rational method differs from the rational method only when a junction receives flow from two or more tributaries. The peak flow (Q), the time of concentration (Tc) and the rainfall intensity (I) for each of the independent tributaries is calculated according to rational method. The modified rational method is then used to determine Q, Tc and I for the confluence.

The rational method is used for the next point of interest along the flow direction.

Q and Tc, for each independent tributaries are ranked in order of increasing Tc. Let Q1, T1, and I1 correspond to the tributary area with the shortest Tc. Likewise, Q2, T2, and I2 correspond to the tributary area with the next longer Tc; and so on. The tributaries are combined as follows.

For $T_1 < T_2 < T_3$,

QT1 = Q1 +

$$Q_{T1} = Q_1 + \frac{T_1}{T_2} Q_2 + \frac{T_1}{T_3} Q_3$$

$$Q_{T2} = Q_2 + \frac{I_2}{I_1} Q_1 + \frac{T_2}{T_3} Q_3$$

$$Q_{T3} = Q_3 + \frac{I_3}{I_1} Q_1 + \frac{I_3}{I_2} Q_2$$

The largest of QT1, QT2 and so on and the associated Tc should be used for the junction. The equation can be expanded for more than three tributaries. InfoSewer Pror can handle unlimited number of tributaries.

San Diego Modified Rational Hydrograph from Innovyze H_2OCalc for Reference

- **Subcatchment Area** – Drainage area of the watershed.
- **Runoff Coefficient** – Rational formula runoff coefficient (see Table 3.9)
- **6-hr Precipitation** – Six-hour design rainfall depth.
- **Time of Concentration** – The time it takes for flow to travel from the hydraulically remotest point in the watershed to get to outlet of the watershed

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Default Values and Units

The following tables summarize the default values used by InfoSewer Pro in lieu of user-supplied values and units. Table 2.1 lists default parameter values while Table 2.2 lists input parameter units.

“Flow units” in the second table can correspond to either cubic feet per second (the default), gallons per minute (gpm), million gallons per day (mgd), imperial million gallons per day (imgd), acre-feet per day (afd), cubic meters per hour (cmh), cubic meters per day (cmd), million liters per day (mld), liters per second (lps), or liters per minute (lpm).

Table 2.1 - Summary of Default Parameter Values

Parameter	Default Value
Maximum Design Velocity	10 ft/s
Minimum Design Velocity	2 ft/s
Particle Settling Velocity	0.1 ft/s
Limiting Flow Velocity	2 ft/s
Reaction Rate Coefficient	0.23/day
Temperature	$20^{o}C$
pH	7.0

Sulfide Flux Coefficient	0.00032m/hr
Sulfide Loss Coefficient	0.96
Ionization Constant	6.92
Soluble Form of Total Sulfide	85 %
Relative Saturation of H2S	0.05
Acid Reaction Efficiency	0.5
Alkalinity	0.5
Time Pattern Multipliers	1 for all patterns
Pattern Time Step	1 hour
Pattern Start	0 hour
Pattern Type	Continuous
Simulation Duration	0 hour (steady-state)
Headloss Formula - Force Main Headloss Formula - Gravity Main	Hazen-Williams Manning

Flow Units	CFS
Global Loading Multiplier	1.0
Peaking Factor , K (Federov Eq.)	2.4
Peaking Factor, p (Federov Eq.)	0.89
Peaking Parameter , a (Harman and Babbitt Eqs.)	5.0
Peaking Parameter, b (Harman and Babbitt Eqs.)	0.0
Peaking Parameter, c (Harman and Babbitt Eqs.)	0.2
Peaking Parameter, d (Harman and Babbitt Eqs.)	0.0
Design Manning Coefficient	0.001
Maximum Design Cover	20 ft

	Minimum Design Cover	3 ft
	Angle of Excavation	0 degrees
	Cutoff Diameter	24 in
	Small Pipes Clearance	8 in
	Large Pipes Clearance	12 in
	Cutoff Depth	10 ft
	Minorloss Crown Drop	0.1 ft
	Maximum Allowable Diameter	0 ft
	Minimum Travel Distance	100 units
	Minimum Travel Time	1 second
	Conserve Pumped Flows	No (Conserve Load Components)
	Flow Attenuation	No
	Manhole Sealing Method	Unlocked

	Max. Number of Segments	100
	Accuracy	.0001
	Start Clocktime	0 hours
	Report Time Step	1 hour
	Pump Hydraulic Time Step	6 minutes
	Quality	None
	Page Size (Lines per Report Page)	0 (no limit)
	Manholes Reported On	Yes
	Links Reported On	Yes
	Report Status	Yes
	Minimum Time of Concentration	10 minutes
	IDF Curve Duration Unit	Minutes

IDF Curve Duration Unit	Minutes
Drainage Area	0 acre
Unit Hydrograph Pattern Type	Explicit

Enable Storm Runoff Computation	No
Maximum Crown Drop at Manhole	3 ft
Manhole Invert Offset	0 ft

Table 2.2 – Units of Measurement

Parameter	**US Customary Units**	**SI (Metric) Units**
Flow	CFS (cubic feet /sec) GPM (gallons / min) MGD (million gallon / day) IMGD (Imperial MGD) AFD (acre-feet / day)	LPS (liters / sec) LPM (liters / min) MLD (megaliters / day) CMH (cubic meters / hr) CMD (cubic meters / day)
Rim Elevation	Feet	Meters
Manhole Load	Flow units	Flow units
Manhole Diameter	Feet	Meters
Particle Settling Velocity	Feet/sec	Meters/sec
Limiting Flow Velocity	Feet/sec	Meters/sec
Reaction Rate Coefficient	Day^{-1}	Day^{-1}
Temperature	^oC	^oC

Wet Well Bottom Elevation	Feet	Meters
Wet Well Levels	Feet above bottom	Meters above bottom
Wet Well Diameter	Feet	Meters
Wet Well Volume	Cubic feet	Cubic meters
Pipe Length	Feet	Meters
Quality/Manholes: Concentration BOD Time of Concentration Source Trace Hydrogen Sulfide	 Milligrams/liter Milligrams/liter Hours Percent Milligrams/liter	 Milligrams/liter Milligrams/liter Hours Percent Milligrams/liter
Quality/Pipes: Concentration BOD Time of Concentration Source Trace Sediment/Deposition Hydrogen Sulfide Corrosion Rate	 Milligrams/liter Milligrams/liter Hours Percent Kilograms Milligrams/liter Inch/year	 Milligrams/liter Milligrams/liter Hours Percent Kilograms Milligrams/liter mm/year
Stormwater Modeling: **Drainage Area** **Subwatershed Area** **Channel Length** 	 **Acre** **Acre** **Feet** **Inch/hour**	 **Square meters** **Square meters** **Meters** **mm/hour**

Rainfall Intensity for Hyetograph **Rainfall Duration for Hyetograph** **Unit Hydrograph Ordinates** **Watershed Hydraulic Length** **Watershed Centroid Distance** **Impervious Retention Storage** **Pervious Retention Storage** **Infiltration Rate** **Horton's Decay Constant** **Infiltration Regeneration Constant**	**Minutes** **Flow Units** **Feet** **Feet** **Feet** **Feet** **Inch/hour** **Sec^{-1}** **None** **Feet** **Inch/hour** **Sec-1** **None**	**Minutes** **Flow Units** **Meters** **Meters** **Millimeters** **Millimeters** **mm/hour** **Sec^{-1}** **None** **Millimeters** **mm/hour** **Sec-1** **None**
Pipe Diameter	Inches	Millimeters
Pipe/Channel Depth	Inches	Millimeters
Pipe/Channel Width	Inches	Millimeters
Pipe/Channel Side Slopes	None	None
Pipe Cover	Inches	Millimeters
Minorloss Crown Drop	Inches	Millimeters
Pipe Roughness Hazen-Williams	 None	 None

Manning	None	None
Pump Head	Feet	Meters
Pump Flow	Flow units	Flow units
Pump Speed Ratio	None	None
Pump Fixed Flow Capacity	Flow units	Flow Units
Angle of Excavation	Degrees	Degrees
Clearance	Inches	Millimeters

Default H&H or Hydrology and Hydraulic Values from SWMM5

A.1 Units of Measurement

PARAMETER	US CUSTOMARY	SI METRIC
Area (Subcatchment)	acres	hectares
Area (Storage Unit)	square feet	square meters
Area (Ponding)	square feet	square meters
Capillary Suction	inches	millimeters
Concentration	mg/L (milligrams/liter) ug/L (micrograms/liter) Count/L (counts/liter)	mg/L ug/L Count/L
Decay Constant (Infiltration)	1/hours	1/hours
Decay Constant (Pollutants)	1/days	1/days
Depression Storage	inches	millimeters
Depth	feet	meters
Diameter	feet	meters
Discharge Coefficient: Orifice	dimensionless	dimensionless
Weir	CFS/foot^n	CMS/meter^n
Elevation	feet	meters
Evaporation	inches/day	millimeters/day
Flow	CFS (cubic feet / second) GPM (gallons / minute) MGD (million gallons/day)	CMS (cubic meters/second) LPS (liters/second) MLD (million liters/day)
Head	feet	meters
Hydraulic Conductivity	inches/hour	millimeters/hour

Infiltration Rate	inches/hour	millimeters/hour
Length	feet	meters
Manning's n	seconds/meter$^{1/3}$	seconds/meter$^{1/3}$
Pollutant Buildup	mass/length mass/acre	mass/length mass/hectare
Rainfall Intensity	inches/hour	millimeters/hour
Rainfall Volume	inches	millimeters
Slope (Subcatchments)	percent	percent
Slope (Cross Section)	rise/run	rise/run
Street Cleaning Interval	days	days
Volume	cubic feet	cubic meters
Width	feet	meters

A.2 Soil Characteristics

Soil Texture Class	K	Ψ	φ	FC	WP
Sand	4.74	1.93	0.437	0.062	0.024
Loamy Sand	1.18	2.40	0.437	0.105	0.047
Sandy Loam	0.43	4.33	0.453	0.190	0.085
Loam	0.13	3.50	0.463	0.232	0.116
Silt Loam	0.26	6.69	0.501	0.284	0.135
Sandy Clay Loam	0.06	8.66	0.398	0.244	0.136
Clay Loam	0.04	8.27	0.464	0.310	0.187
Silty Clay	0.04	10.63	0.471	0.342	0.210

Loam					
Sandy Clay	0.02	9.45	0.430	0.321	0.221
Silty Clay	0.02	11.42	0.479	0.371	0.251
Clay	0.01	12.60	0.475	0.378	0.265

K = saturated hydraulic conductivity, in/hr ψ = suction head, in.

φ= porosity, fraction **FC** = field capacity, fraction **WP** = wilting point, fraction

Source: Rawls, W.J. et al., (1983). *J. Hyd. Engr.*, 109:1316.

Note: The following relation between Ψ and K can be derived from this table:

$$\Psi = 3.23\ \mathbf{K}^{-0.328} \quad (R^2 = 0.9)$$

A.3 NRCS Hydrologic Soil Group Definitions

Group	Meaning	Saturated Hydraulic Conductivity (in/hr)
A	Low runoff potential. Water is transmitted freely through the soil. Group A soils typically have less than 10 percent clay and more than 90 percent sand or gravel and have gravel or sand textures.	> 1.42
B	Moderately low runoff potential.	0.57 – 1.42

	Water transmission through the soil is unimpeded. Group B soils typically have between 10 percent and 20 percent clay and 50 percent to 90 percent sand and have loamy sand or sandy loam textures.	
	Moderately high runoff potential.	
C	Water transmission through the soil is somewhat restricted. Group C soils typically have between 20 percent and 40 percent clay and less than 50 percent sand and have loam, silt loam, sandy clay loam, clay loam, and silty clay loam textures.	0.06 - 0.57
	High runoff potential.	
D	Water movement through the soil is restricted or very restricted. Group D soils typically have greater than 40 percent clay, less than 50 percent sand, and have clayey textures.	< 0.06

Source: Hydrology National Engineering Handbook, Chapter 7, Natural Resources Conservation Service, U.S. Department of Agriculture, January 2009.

A.4 SCS Curve Numbers[1]

Land Use Description	**Hydrologic Soil Group**			
	A	**B**	**C**	**D**
Cultivated land				
Without conservation treatment	72	81	88	91
With conservation treatment	62	71	78	81
Pasture or range land				
Poor condition	68	79	86	89
Good condition	39	61	74	80
Meadow				

Good condition	30	58	71	78
Wood or forest land				
Thin stand, poor cover, no mulch	45	66	77	83
Good cover[2]	25	55	70	77
Open spaces, lawns, parks, golf courses, cemeteries, etc.				
Good condition: grass cover on 75% or more of the area	39	61	74	80
Fair condition: grass cover on 50-75% of the area	49	69	79	84
Commercial and business areas (85% impervious)	89	92	94	95
Industrial districts (72% impervious)	81	88	91	93
Residential[3]				
Average lot size (% Impervious[4])				
1/8 ac or less (65)	77	85	90	92
1/4 ac (38)	61	75	83	87
1/3 ac (30)	57	72	81	86
1/2 ac (25)	54	70	80	85
1 ac (20)	51	68	79	84
Paved parking lots, roofs, driveways, etc.[5]	98	98	98	98
Streets and roads				
Paved with curbs and storm sewers[5]	98	98	98	98
Gravel	76	85	89	91
Dirt	72	82	87	89

Source: *SCS Urban Hydrology for Small Watersheds*, 2nd Ed., (TR-55), June 1986.

Footnotes:

1. Antecedent moisture condition II.

2. Good cover is protected from grazing and litter and brush cover soil.
3. Curve numbers are computed assuming that the runoff from the house and driveway is directed toward the street with a minimum of roof water directed to lawns where additional infiltration could occur.
4. The remaining pervious areas (lawn) are considered to be in good pasture condition for these curve numbers.
5. In some warmer climates of the country a curve number of 95 may be used.

A.5 Depression Storage

Impervious surfaces	0.05 - 0.10 inches
Lawns	0.10 - 0.20 inches
Pasture	0.20 inches
Forest litter	0.30 inches

Source: ASCE, (1992). *Design & Construction of Urban Stormwater Management Systems*, New York, NY.

A.6 Manning's n – Overland Flow

Surface	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 pipes	0.024
Cement rubble surface	0.024
Fallow soils (no residue)	0.05

Cultivated soils Residue cover < 20%	0.06
Residue cover > 20%	0.17
Range (natural)	0.13
Grass Short, prairie Dense	0.15 0.24
Bermuda grass	0.41
Woods Light underbrush Dense underbrush	0.40 0.80

Source: McCuen, R. et al. (1996), *Hydrology*, FHWA-SA96-067, Federal Highway Administration, Washington, DC

A.7 Manning's n – Closed Conduits

Conduit Material	Manning n
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) - Smooth forms	0.012 - 0.014
- Rough forms	0.015 - 0.017
Concrete pipe	0.011 - 0.015
Corrugated-metal pipe (1/2-in. x 2-2/3-in. 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 - Liner plates	 0.011 - 0.015 0.013 - 0.017

Source: ASCE (1982). *Gravity Sanitary Sewer Design and Construction*, ASCE Manual of Practice No. 60, New York, NY.

A.8 Manning's n – Open Channels

Channel Type	Manning n
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 - 0.40
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 - 0.140
Natural channels (minor streams, top width at flood stage < 100 ft)	
- Fairly regular section	0.030 - 0.070
- Irregular section with pools	0.040 - 0.100

Source: ASCE (1982). *Gravity Sanitary Sewer Design and Construction*, ASCE Manual of Practice No. 60, New York, NY.

A.9 Water Quality Characteristics of Urban Runoff

Constituent	Event Mean Concentrations
TSS (mg/L)	180 - 548
BOD (mg/L)	12 - 19
COD (mg/L)	82 - 178
Total P (mg/L)	0.42 - 0.88
Soluble P (mg/L)	0.15 - 0.28
TKN (mg/L)	1.90 - 4.18
NO2/NO3-N (mg/L)	0.86 - 2.2
Total Cu (ug/L)	43 - 118
Total Pb (ug/L)	182 - 443
Total Zn (ug/L)	202 - 633

Source: U.S. Environmental Protection Agency. (1983). *Results of the Nationwide Urban Runoff Program (NURP), Vol. 1*, NTIS PB 84-185552), Water Planning Division, Washington, DC.

Source: http://nepis.epa.gov/Exe/ZyPDF

References for InfoSewer

- American Society of Civil Engineers Manuals and Reports on Engineering Practice No. 60 (1982). *Gravity Sanitary Sewer Design and Construction*. Reston, VA.
- Babbitt, H.E. and Baumann, E.R. (1958). *Sewerage and Sewage Treatment*. John Wiley & Sons, Inc., New York, NY.
- Bedient, P. B. and Huber, W.C. 1992. Hydrology and Flood Plain Analysis. Addison-Wesley, New York.
- Chatila, J.G. (2003). Muskingum Method, EXTRAN and ONE-D for Routing Unsteady Flows in Open Channels. *Journal Canadian Water Resources*, Vol. 28, No. 3, pp. 481-498.
- Cunge JA and Wegner M 'Integration numerique des equations d'ecoulement de Barre de Saint-Venant parun schema implicite de differences finies: Application au cas d'une galerie tantot en charge, tantot a surfacelibre', La Houille Blance, No 1, 1964
- Dickinson, R.E., Huber, W.C., Roesener, L.A. and Aldrich, J.A. (June 1990) Advances in SWMM EXTRAN flow routing. Proceedings of 5th International Conference on Urban Storm Drainage, Tokyo Japan, pp. 287-292
- Goldberg, D.E. (1989). *Genetic Algorithms in Search, Optimization and Machine Learning*. Addison Wesley, Reading, MA.
- Gomez et al. Hydraulic Engineering Software IV, Fluid Flow Modelling, 1992
- Huber, C., & Dickinson, R.E. (1988). Storm Water Management Model: User’s Manual. U.S. EPA/600/3-88/001a. Athens: Georgia (USA).
- Huber, W.C., Heaney, J.P., Aggidis, D.A., Dickinson, R.E. and Wallace, R.W., "Urban Rainfall-Runoff-Quality Data Base," EPA-600/2-81-238 (NTIS PB82221094), Environmental Protection Agency, Cincinnati, OH, October 1981a.

- Huber, W.C., Heaney, J.P., Medina, M.A., Peltz, W.A., Sheikh, H. and Smith, G.F., "Storm Water Management Model User's Manual - Version II," EPA-670/2-75017 (NTIS PB-257809), Environmental Protection Agency, Cincinnati, OH, March 1975.
- Huber, W.C., Heaney, J.P., Aggidis, D.A., Dickinson, R.E. and Wallace, R.W., "Urban Rainfall-Runoff-Quality Data Base," EPA-600/2-81-238 (NTIS PB82221094), Environmental Protection Agency, Cincinnati, OH, October 1981a.
- Huber, W.C., Heaney, J.P. and Cunningham, B.A., "Storm Water Management Model (SWMM) Bibliography," EPA Report EPA/600/3-85-077 (NTIS PB86-136041/AS), Environmental Protection Agency, Athens, GA, September 1985.
- Huber, W.C., Heaney, J.P., Medina, M.A., Peltz, W.A., Sheikh, H. and Smith, G.F., "Storm Water Management Model User's Manual - Version II," EPA-670/2-75017 (NTIS PB-257809), Environmental Protection Agency, Cincinnati, OH, March 1975.
- Huber, W.C., Heaney, J.P., Nix, S.J., Dickinson, R.E. and Polmann, D.J., "Storm Water Management Model User's Manual, Version III," EPA-600/2-84-109a (NTIS PB84-198423), Environmental Protection Agency, Cincinnati, OH, November 1981b
- James, W., Rossman, L.A. , Huber, W.C. , Dickinson, R.E. , James, R.C. , Roesner, L.A., Aldrich, J.A. , 2008. User’s guide to SWMM5, 12th Edition, Computational Hydraulics International, Guelph, Ontario.
- Liggett, J A and Cunge, J A, "Numerical methods of solution for the unsteady flow equation, in unsteady flow in open channels", 1, Edited by Mahmood, K. and Yevjevich, V, Water Resources Publ., Fort Collins, Colorado, 1975.
- Maidment, D.R. (1993). *Handbook of Hydrology.* McGraw-Hill, New York, NY.
- Mays, L.W. (2001). *Stormwater Collection Systems Design Handbook*. McGraw-Hill, New York, NY.

- Metcalf & Eddy, Inc. (2003). *Wastewater Engineering Treatment and Reuse*. G. Tchobanoglous ed., Fourth Edition. McGraw Hill, New York, NY.
- Metcalf & Eddy, Inc. (1981). *Wastewater Engineering: Collection and Pumping of Wastewater*.
- G. Tchobanoglous ed.. McGraw Hill, New York, NY.
- Preissmann A, Propagation des intumescences dans les canaux et rivieres. Presented at the 1st Congress of the French association for Computation, Grenoble, France, 1961
- Richens, P J, "Development of an accurate simulation model for urban storm sewers", Report No IT297, DoE Contract PEDC 7/7/053, HR Wallingford Ltd, 1985.
- Rossman, Lewis A. 2008. Storm Water Management Model User's Manual Version 5.0. Pub. Water Supply and Water Resources Division, National Risk Management Research Laboratory, Cincinnati, OH.
- U.S. Department of Agriculture, Soil Conservation Service (1986). *Urban Hydrology for Small Watersheds*. Washington, D.C.
- Yen, B C 'Open channel flow equations revisited' J Eng Mech Div ASCE, Vol 99, No EM5, 1973
- Bennett, D. et al. (1999). Using flow prediction technologies to control sanitary sewer overflows. Water Environment Research Foundation (WERF), Project 97-CTS-8, Alexandria, VA.
- Boulos, P.F. (2005). InfoSWMM Suite – Users guide. MWH Soft, Inc., Broomfield, CO.
- Boulos, P.F. (2006). H2OMAP SWMM Suite – Users guide. MWH Soft, Inc., Broomfield, CO.

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Frequently Asked Questions

This section addresses some of the common questions that you might have pertaining to the InfoSewer model creation, data elements, result analysis and project maintenance.

- How do I start an InfoSewer session ?
- Click on the InfoSewer icon on your Desk top to start InfoSewer or alternatively launch InfoSewer from your Start -> Programs -> InfoSewer -> InfoSewer. Choose from the three options specified. Choose A new empty map to open a new project, A template to launch your template project or the An existing map: command to open an existing map. If you choose this option then choose the project that you want to open. Finally initialize your InfoSewer project by clicking on the InfoSewer initialize icon  on your InfoSewer Edit Network toolbar.
- How do I specify a coordinate system for an InfoSewer project?
- Refer to the section Create New InfoSewer Project to learn how to specify a coordinate system for an InfoSewer project.
- What does an InfoSewer project consist of ?
- An InfoSewer project consists of a drawing file and a database folder. For instance a project named Model will contain a Model.mxd (ArcMap drawing file) and a Model.IEDB (InfoSewer Database). The mxd file contains your network map while the IEDB folder contains all the data associated with your model.
- After a successful model simulation another folder called Model.Out is created. This folder contains all the output data related to the InfoSewer project Model. It is important to note that the Database folder (Model.IEDB), the drawing file (Model.mxd) and the Out folder (Model.Out) need to reside within the same parent folder such as say C:\Data.
- What are the different Data Elements in InfoSewer and how do I create and modify them ?

- Refer to the section on Data Elements
- What are Manholes & how do I create them ?
- Refer to the section on Manholes to learn more.
- What are the different types of Pumps in InfoSewer ? How do I create and edit them ?
- Refer to the section Pumps to learn more.
- How are Wet Wells nodes modeled in InfoSewer ?
- Refer to the section on Wet Wells for details.
- How are Pipes modeled in InfoSewer ? Tell me more about them.
- Refer to the section on Pipes for details.
- How do I digitize my model quickly?
- Use the Network creation process.
- How do I select a sub set of my model to display results, modify data etc ?
- Use the Domain feature of InfoSewer to select a sub set of your model for display and data edit purposes. The Domain Methodology section explains all the different stages in the Domain Creation process.
- How do I activate a sub set of my model for analysis? What are Facility Sets ? Can you explain the Facility Set creation process ?
- Facility sets may be used to activate a portion of your mode for your network analysis. Refer to the section on the Facility Methodology to learn about creating Facility Sets.
- How are Facility Sets different from Domains ?
- Facility vs Domain section of the help file highlights the difference between facility sets and domains.
- How do I edit data related to a selected group of elements ?
- Use the Group Edit option to modify data related to a group of elements in InfoSewer. Refer to the section on Group Editing

Methodology to learn more about the process and how it can be used in InfoSewer.

- Teach me more about Curves.
- Refer to the section on Curves
- Teach me more about Patterns.
- Refer to the section on Patterns
- I want to create Database Queries. Please explain the procedure.
- The DB Query Methodology section explains the process and how you can use it to query your InfoSewer Database.
- Can you explain the process of creating Query Sets and how they are different from DB Queries ?
- Refer to the section on the Query Sets Methodology to learn more.
- Can I save a domain in InfoSewer? How do I create Selection Sets and how can I use them in InfoSewer ?
- Use Selection Sets to save domains for later retrieval. A step by step process for creating selection sets is included in the Selection Set Methodology section. Also advantages and applications of Selection Sets in InfoSewer is explained.
- How do I create Output Relates ? What are the uses of Output Relates ?
- Refer to the section on the Output Relate Methodology to learn the Output Relate creation process.
- How can I use an Output Relate with my DB Query ? Can I query my Output results ?
- Refer to the section Query Output relate Methodology for step by step instructions on creating Output Relates and using them.
- How do I create a Query Report in InfoSewer ?
- Refer to the section on the Query Report Methodology to learn more.

- Can you explain the process to create Query Summation Reports in InfoSewer ?
- The Query Summation Report Methodology contains instructions to create Query Summation Reports in InfoSewer.
- How do I look at my InfoSewer Reports ?
- Refer to the Output Report Methodology section to learn more.
- How do I Compare Data in InfoSewer ?
- The How to Compare Output Data section contains the different methods and tools available in InfoSewer to compare data.
- What is a Graph Settings Template File ? How do I use it to create graph templates ?
- Refer to the section Graph Settings Methodology. This section will explain the process of creating templates and loading them in InfoSewer.
- I want to set Initial Controls on my Data Elements. How do I go about it ?
- Use the Initial Status Methodology section to learn more about setting initial status on your different InfoSewer data elements.
- Can you help me create and assign Simple Controls to my project ?
- Refer to the section on the Simple Controls Methodology to learn more about creating simple controls and using them in InfoSewer.
- What is an Extended Period Simulation and How do I run an EPS model in InfoSewer ?
- The EPS Modeling section will help you create and run an EPS model.
- I want to run all my scenarios together. How do I do it ?
- The Batch Run Manager can be used to run all your scenarios at the same time. Refer to the section on the Batch Run Methodology to learn more.
- How do I run simulations in InfoSewer ?

- The Run Manager Methodology section will give you step by step instructions on "running" a model in InfoSewer.
- I want to use the Real Time Connection command but don't know how to do it ?
- The Real Time Data Connection Methodology explains the procedure and uses of the Real Time Data Connection command in InfoSewer.
- Please explain the steps necessary to run a Steady State Model in InfoSewer ?
- The Steady State Modeling Methodology explains the different steps required to run a steady state model in InfoSewer.
- Can you help me run a Water Quality Simulation in InfoSewer ?
- The Water Quality Simulation Methodology section will walk you through the process of running a Water Quality Model.
- How do I Compare Scenarios in InfoSewer ?
- Refer to the section on Compare Scenarios Methodology to learn more.
- How do I create Data Sets in InfoSewer ?
- The Data Sets Methodology section explains the data set creation process in detail.
- What are Facility Sets and how do I use them in InfoSewer ?
- Refer to the section on the Facility Sets Methodology to learn more.
- Can I customize the Simulation Options for my scenarios and if so how ?
- The General Options Methodology section contains step by step instructions to help you customize your Simulation options.
- How do I create Scenarios in InfoSewer ?
- Refer to the section on the Scenario Creation Methodology to learn more.
- Can I create a model from my Polylines ?

- Use the Append Nodes Methodology section to learn the procedure for creating pipes and nodes from your AutoCAD polylines.
- I would like to use the InfoSewer ODBC command but don't know how ?
- The ODBC Methodology section will help you understand and use the Open Database Connection tool of InfoSewer to import or export to a third party data source.
- How can I create animations in InfoSewer ?
- Refer to the Animation Viewer Methodology section to learn more. This section contains detailed instructions to help you create animations and view them in InfoSewer.
- How do I color code my Data Elements in InfoSewer ?
- Use the Map Display Command to color code your Data Elements. For step by step instructions refer to the Map Display Methodology section of the InfoSewer help file.
- Can I create annotations in InfoSewer ? Can you help me create them ?
- Refer to the section on Annotation Methodology to learn the annotation creation process.
- Can I create contours in InfoSewer ? Can you help me create them ?
- The Contour Methodology section contains step by step instructions that will walk you through the contour creation process.
- I would like to work with Geodatabases. Can you explain the procedure ?

Refer to the section on Working with ESRI Geodatabases to learn more.

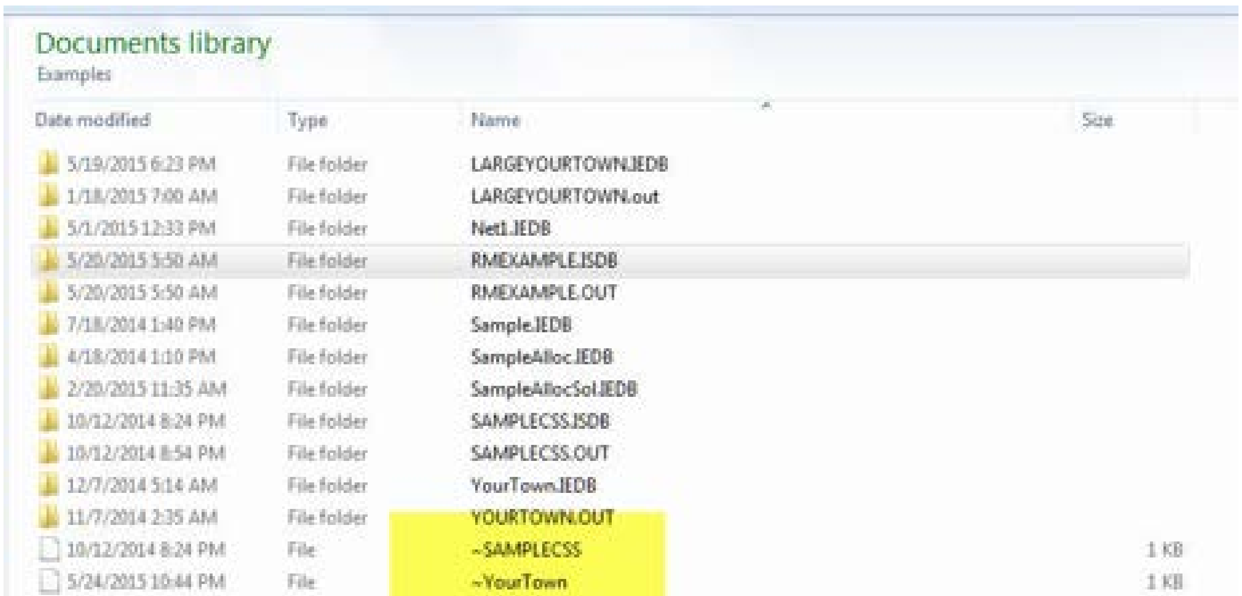
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

File Locking

File Locking for Multi-user Environments

The File Locking feature is designed to support a "Multi-User" and "Multi-Editing" environment like Enterprise Network Storage, Terminal Server, Remote Access, and CITRIX.



When a project gets "initialized", a project lock file is created which ensures the exclusive editing privilege of the current user. A project lock file is a text file that resides in parallel with the project file and the project database folder. The file name is composed of the project file name prefixed by a "~" (tilde) symbol and no file extension.

The presence of this file tells a multi-user operating system that the file is in use and it is not to be used or edited by any other users.

When a project is terminated normally, the lock file is purged by the application and it is automatically deleted. If an unexpected termination is encountered, the lock file may be left undeleted. An attempt to edit the project by the same user will be allowed. The familiar "Editing Session Found" dialog will be presented to the user which will allow the user to continue the previous editing session. The same workflow process is put in practice when the same user launches another editing session and tries to edit the same project.

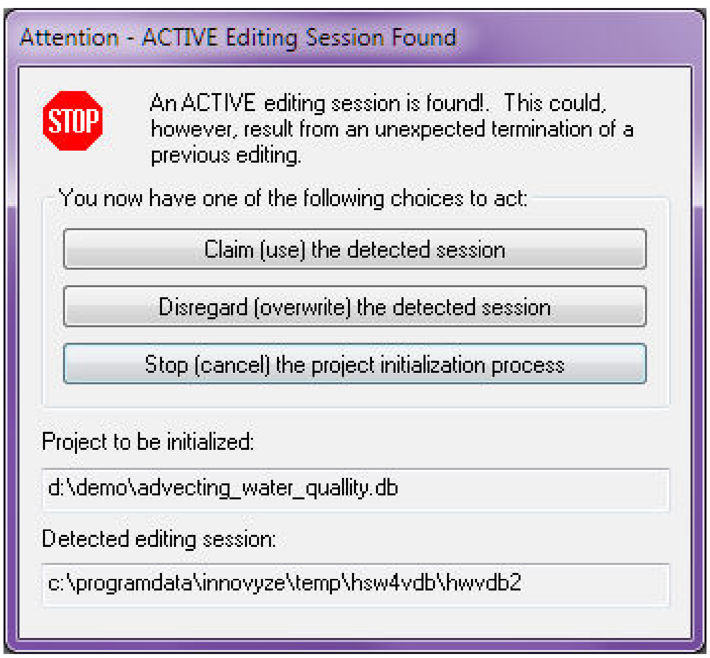
When a different user tries to initialize a project with an existing project lock file, the access to the project files will be declined and an access error message generated.

The lock file is not locked by the operating system. This is to allow a possible manual removal if the user has a reason to do so. In the event that the lock file is copied and delivered together with the project files to another user, the lock file can be deleted and the project loaded and edited as usual. If required, the manual removal of lock file will enable the project to be edited by another user.

Active Editing Session Found

The Active Editing Session Found message appears when trying to open a crashed project or a project being edited by another session. Several choices are presented and one of the choices must be selected to continue.

If a previous session terminated abnormally, the project may still be claimed by the inactive session. It is up to the user to decide what is appropriate response for this session.



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

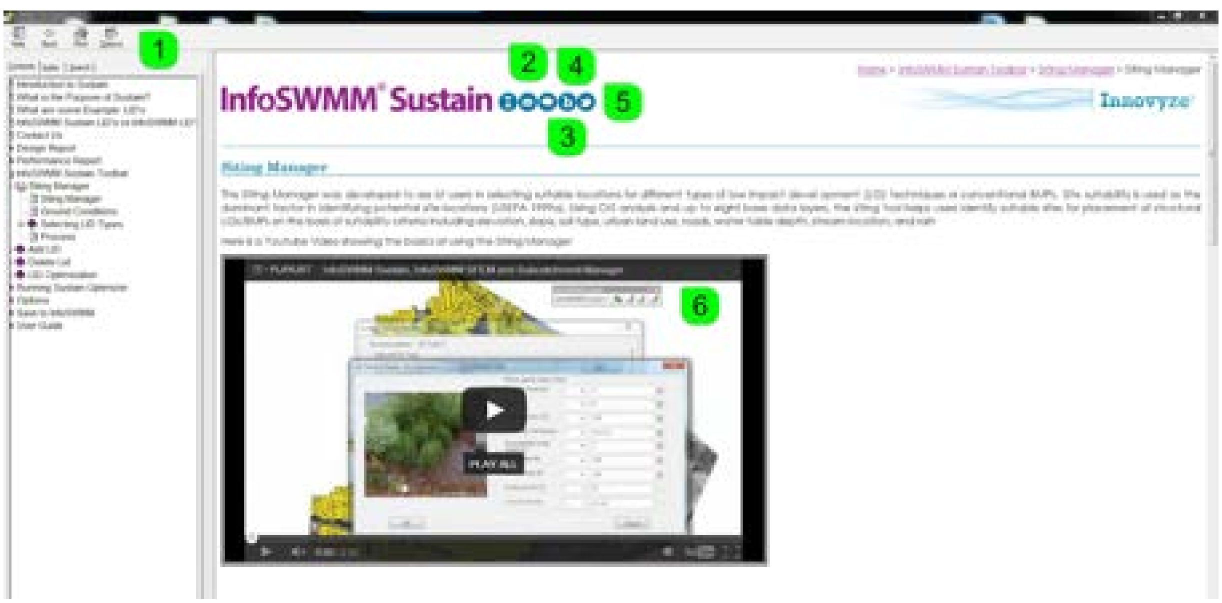
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Hybrid Learning Help File

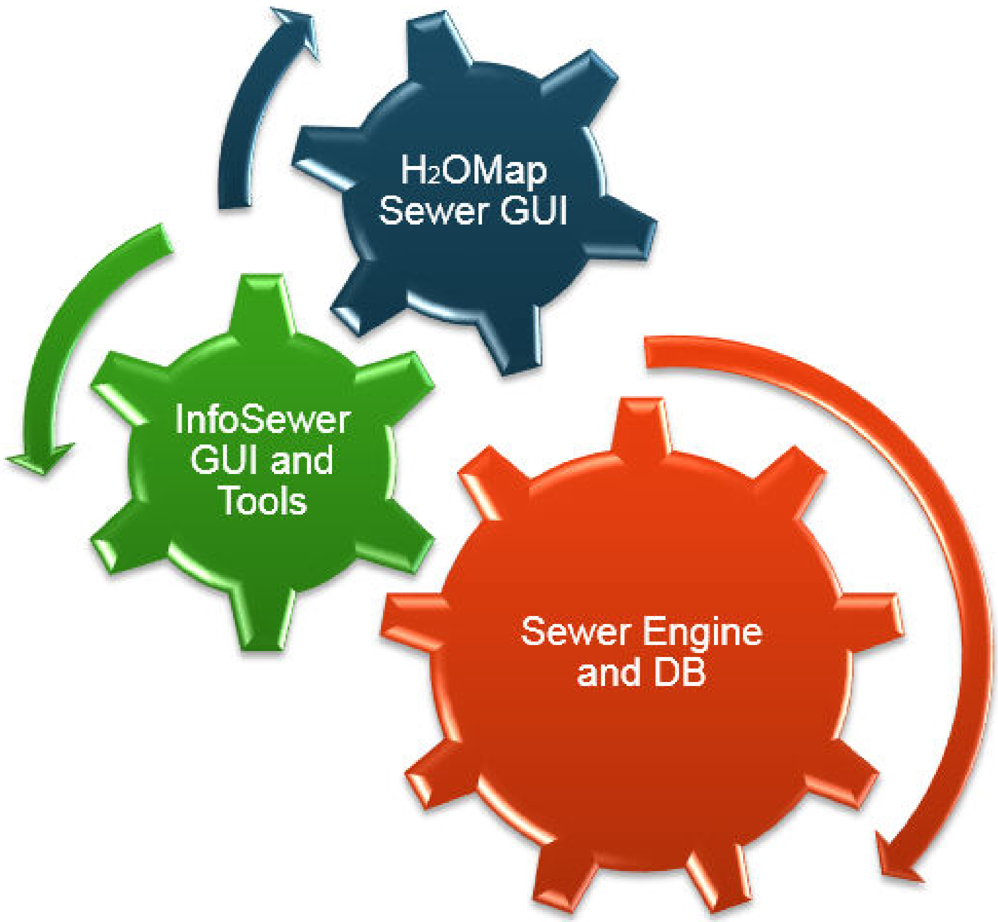
We are trying the concept of Hybrid Learning at Innovyze to help accommodate all the possible learning styles of our users. As part of this help file you will see a social media and multiple media forms of help which include:

- Help Files with links to Blogs, Forums and Videos
- Support Email/Phone Calls with our Support linking you to online Blogs and Videos. Contact information can be found on Contact or at the bottom of each page of the help file.
- Blogs, Innovyze and Other Blogs
- Forums for all of our products including SWMM5 and EPANET
- Twitter/LinkedIn
- YouTube and Vimeo in Help Files and as Youtube and Vimeo Online Videos . Including embedded videos in this help file.
- Multiple Tutorials and Example Files for Quickly Learning the power and Complexity of InfoSWMM Sustain



A Combined Sewer Help File

This is a combined Help file for InfoSewer and H_2OMap Sewer as both program share the same engine, the same DB or Database File format, the same Attribute Browser, the same output reports and graphs and many of the same menu commands and tools. The obvious difference is that InfoSewer has many more spatial tools as it is an extension inside Arc GIS.



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

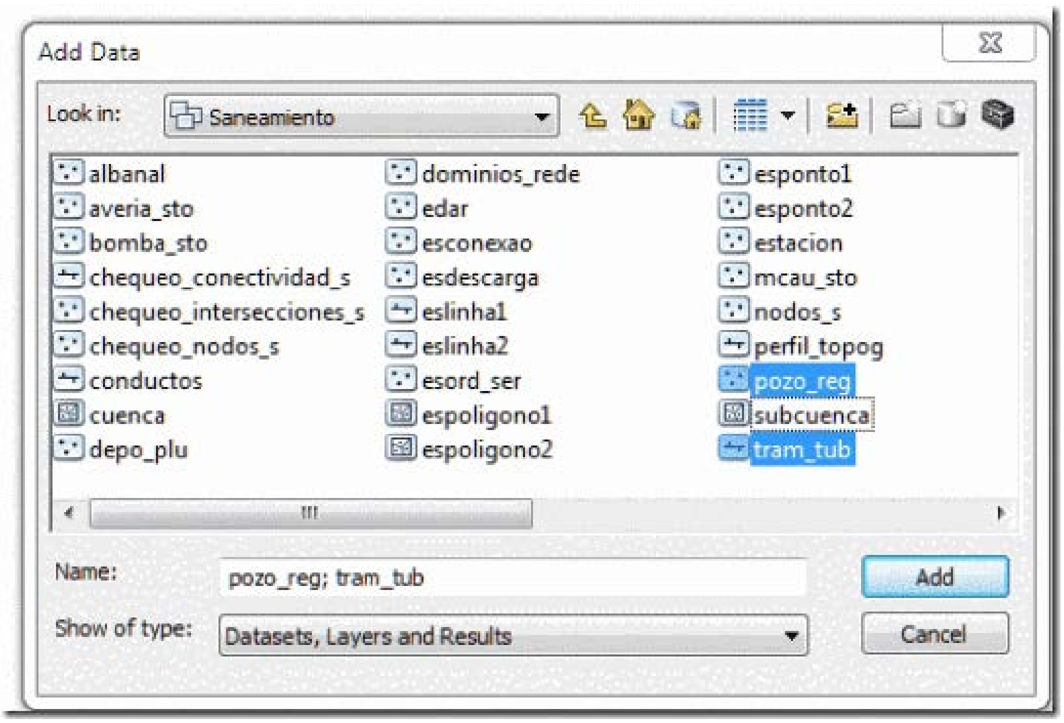
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

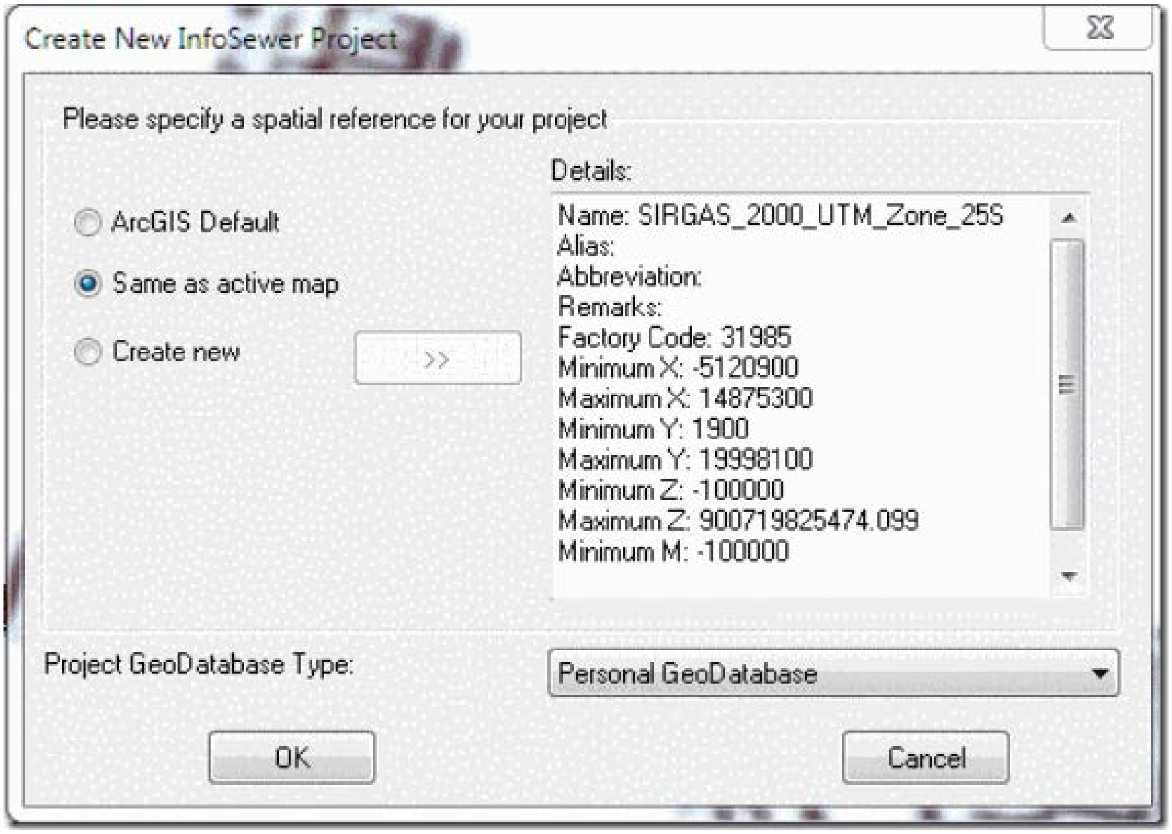
Steps to Import data into InfoSewer Using GIS Gateway

Here are the steps from the beginning for importing data into InfoSewer:

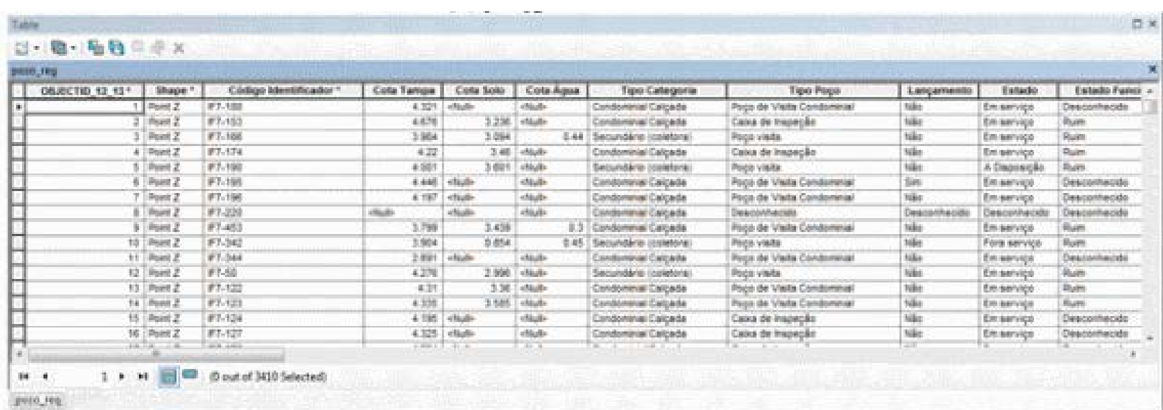
Make a new InfoSewer Model and add in the layers you want to import



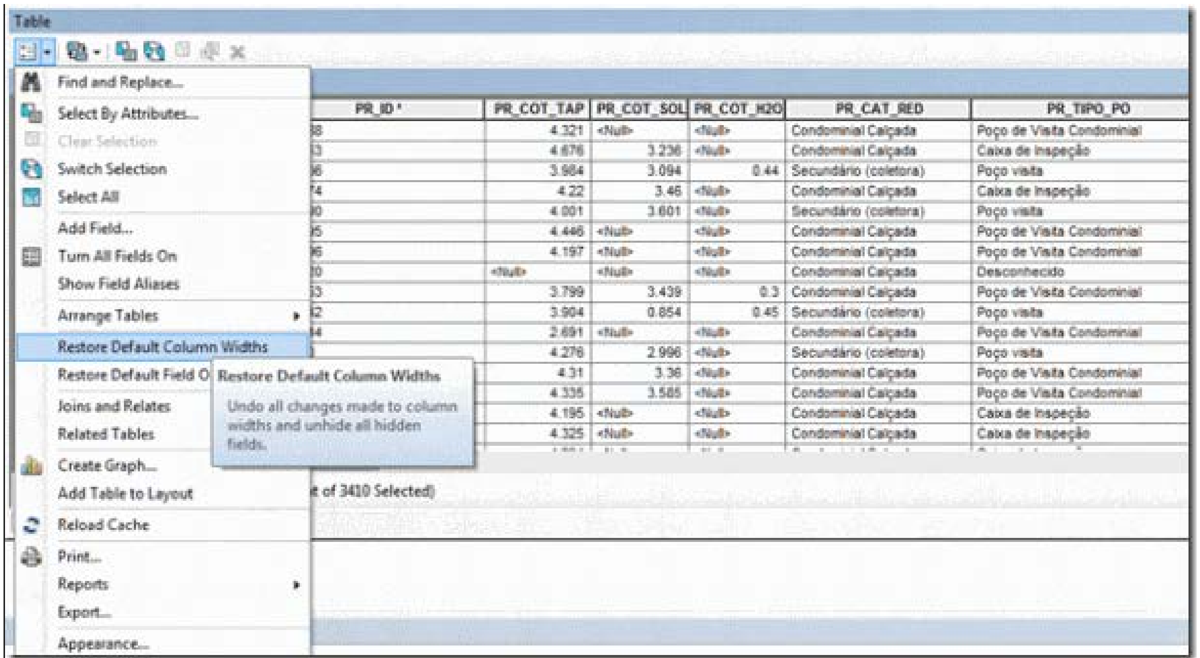
Initialize the InfoSewer Arc Map to the Coordinates of the Layers you have added



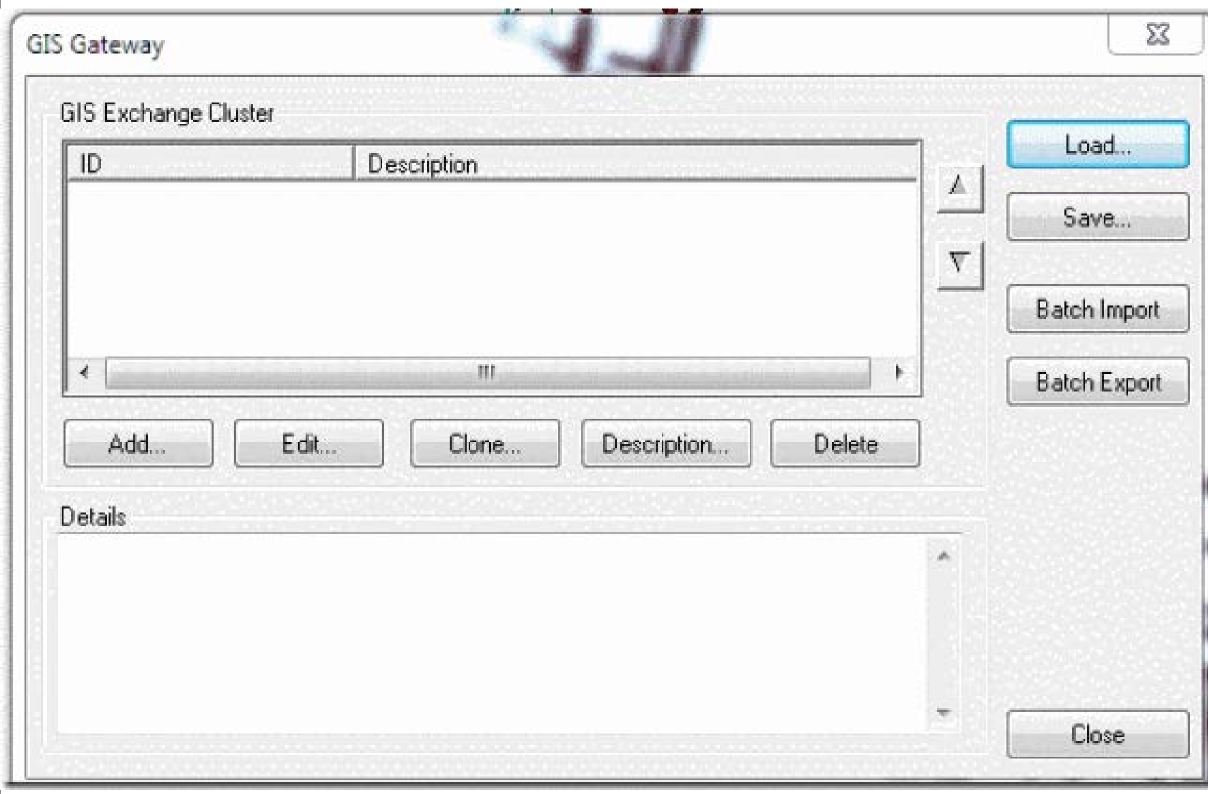
Look at the Attributes of the of the Layers in Arc Map by using the command Open Attribute Table in Arc GIS – right mouse click on the Arc Map Table of Contents (TOC)



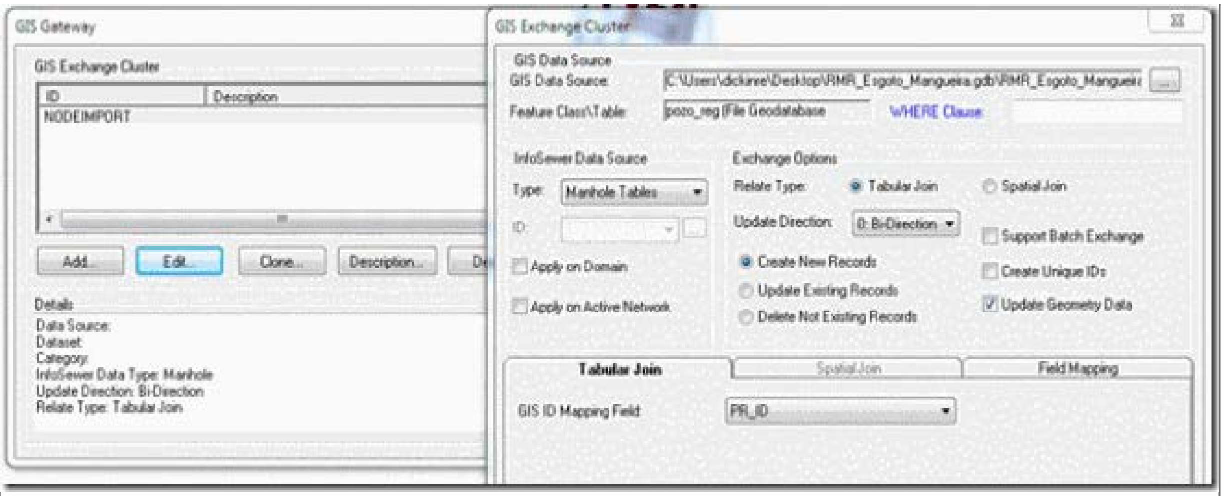
Turn off the Alias names as InfoSewer GIS Gateway only read the “real” column names



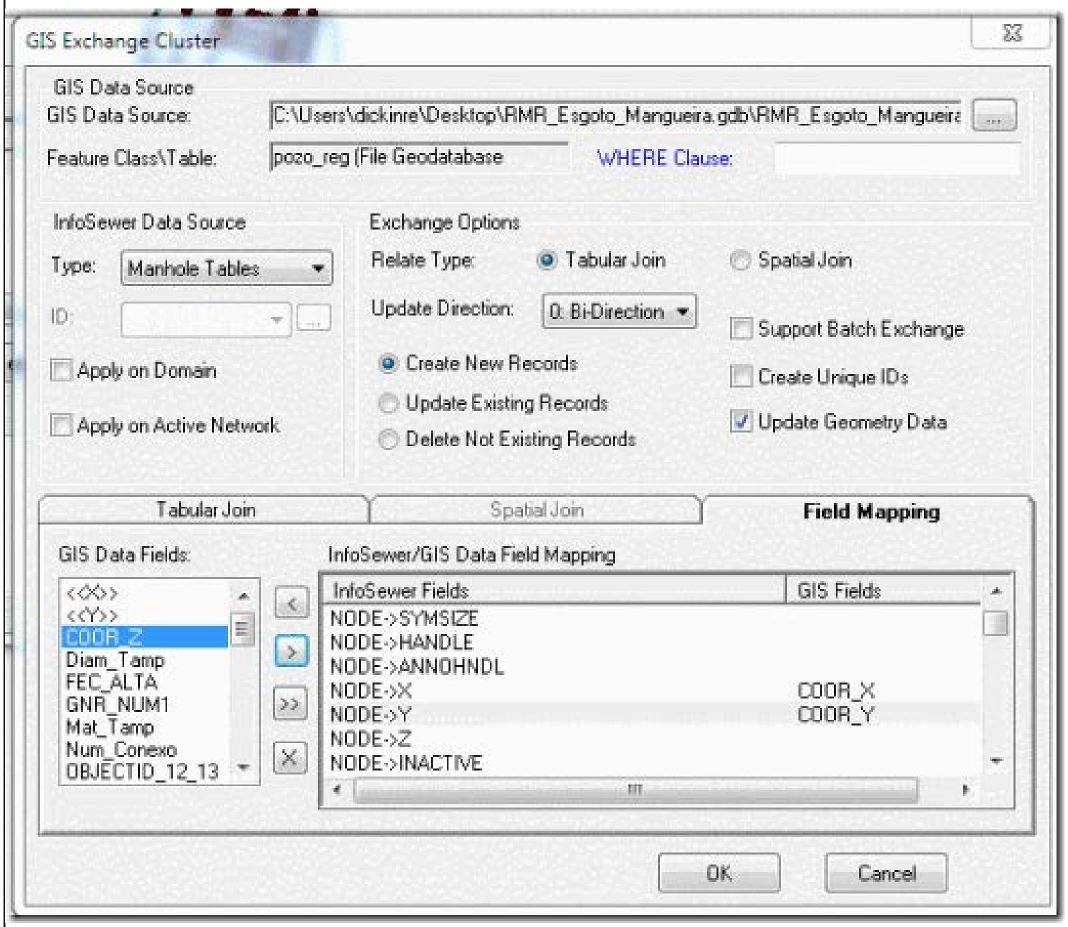
Use the InfoSewer GIS Gateway Tool



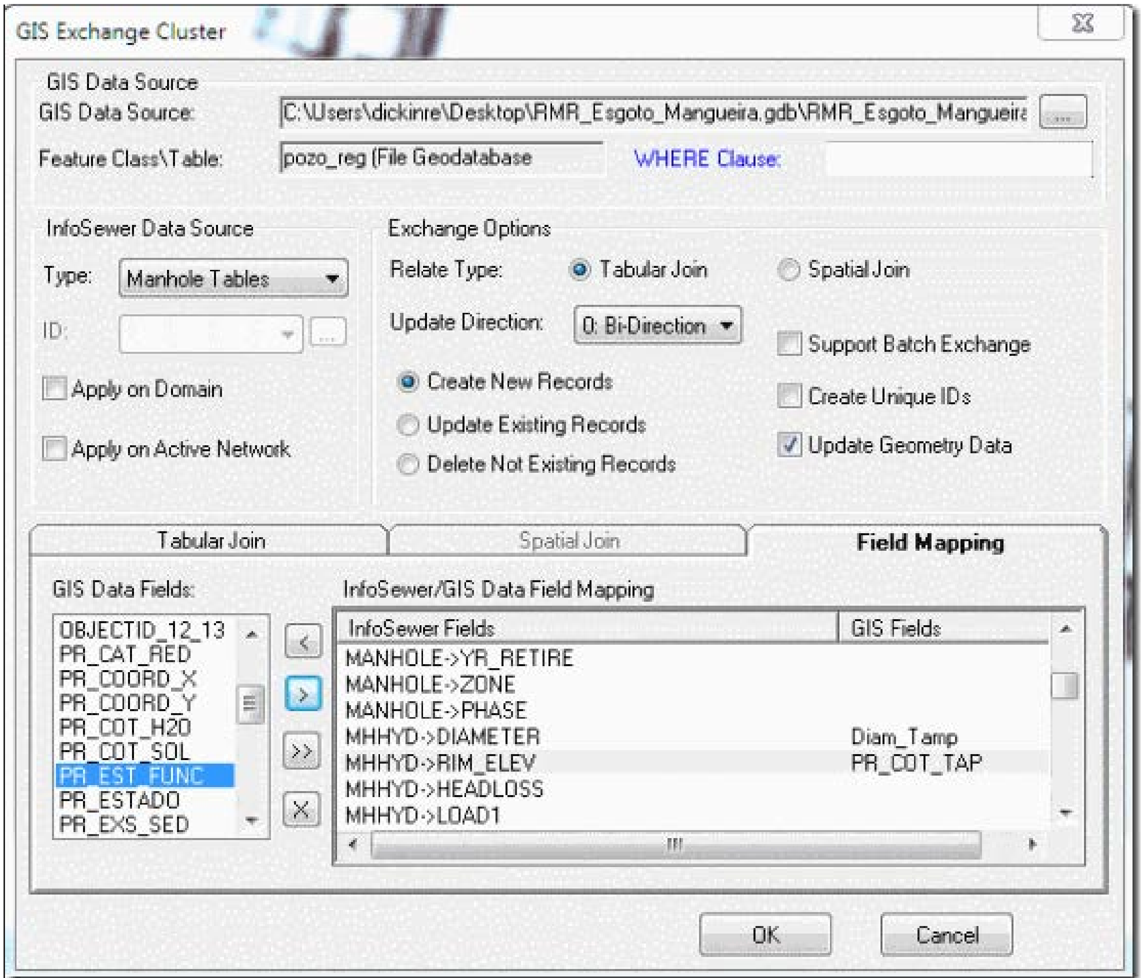
Load the Nodes 1st using the PR_ID as the Mapping Field



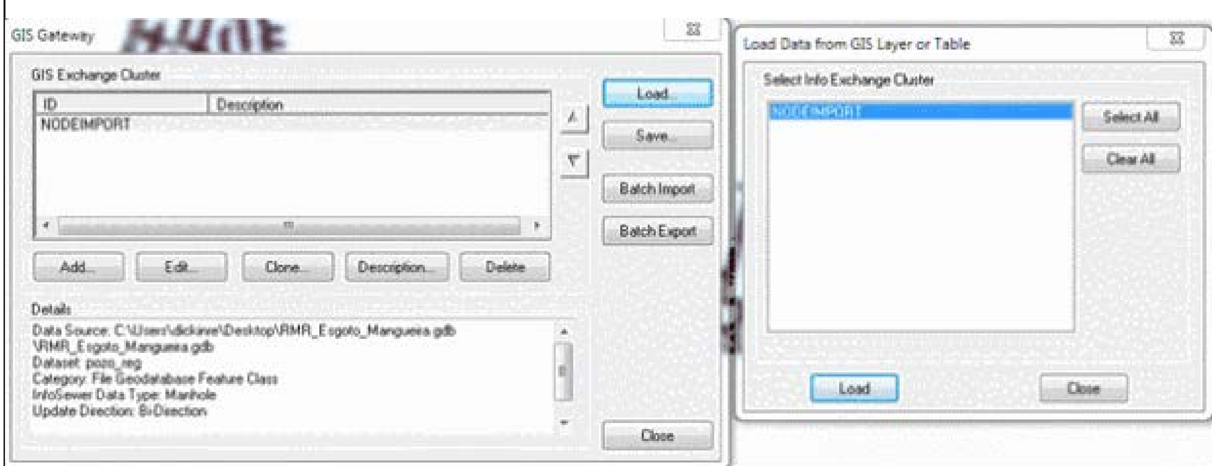
The map fields for the X, Y Coordinates



Import the Diameter (Diam_Tamp) and Rim Elevation (PR_COT_TAP) for the Node

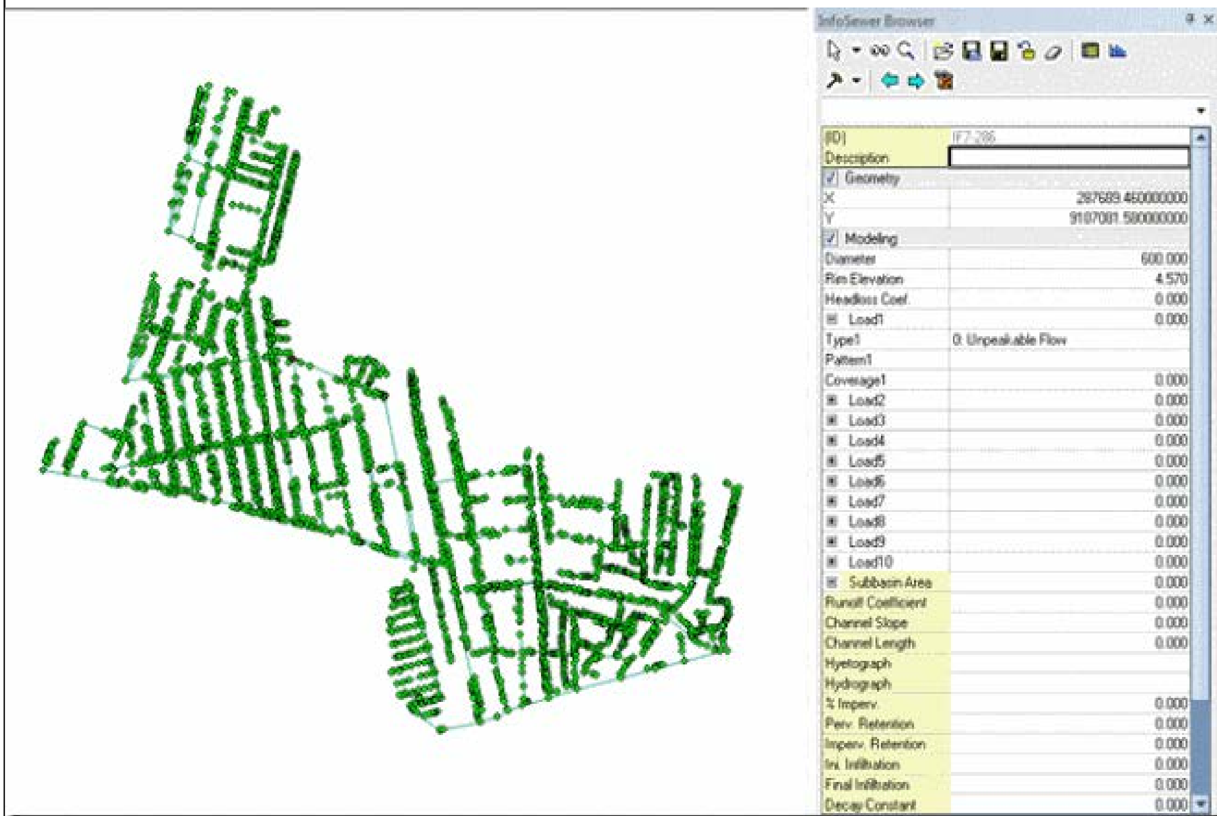


Load the defined GIS Gateway Layer

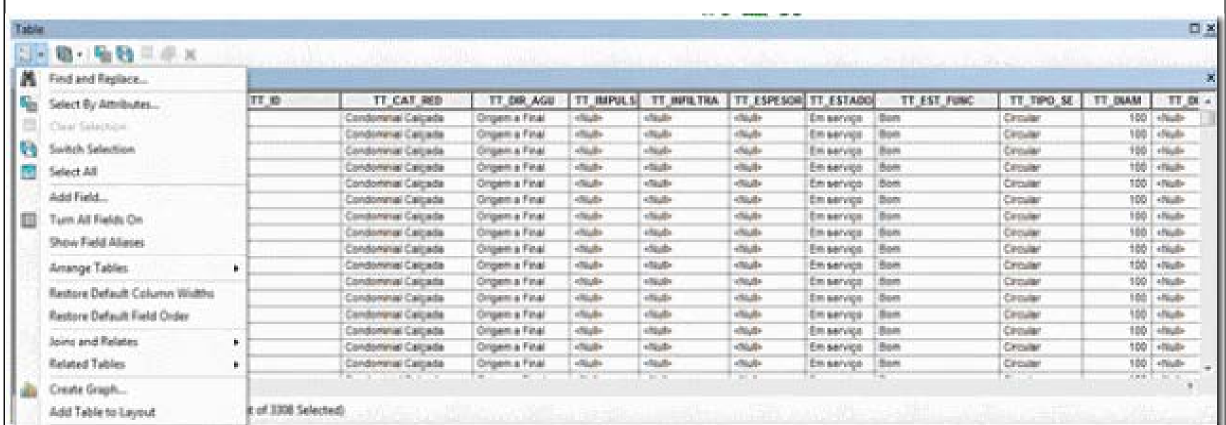


If the nodes were imported there should be a log message file and the nodes will be seen on the Map and in the InfoSewer Attribute Table

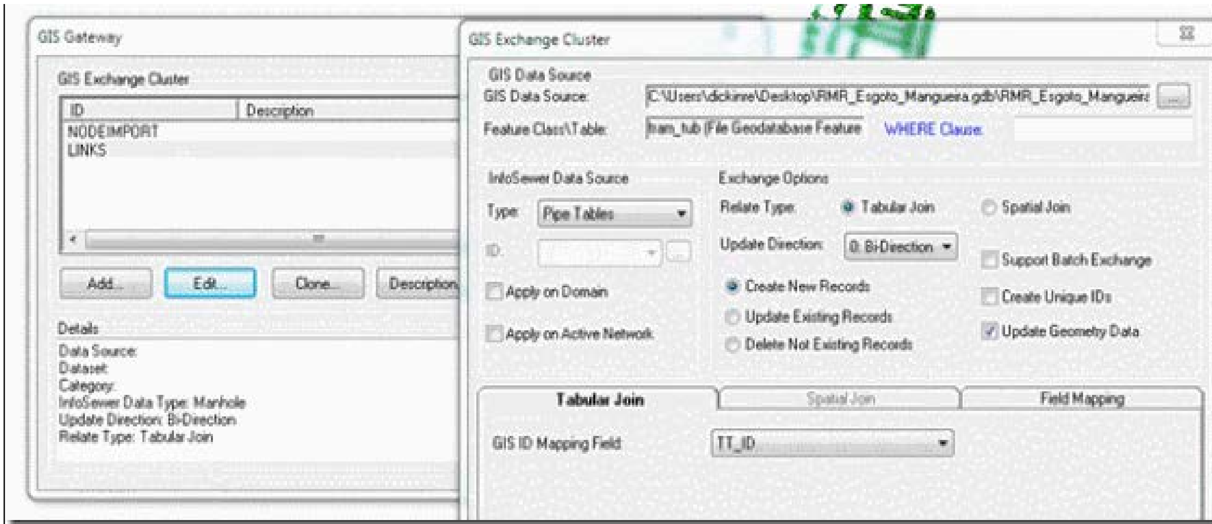
Exchanging data on 'NODEIMPORT'...



Turn off the Field Aliases for the Links so we can see the actual column names



Use TT_ID as the GIS ID Mapping Field



a. Gis Data Source: C:\Users\ … \tram_tub

b. InfoSewer Data Source Type: Pipe Tables

c. Relate Type: Tabular Join

d. Update Direction: 0:Bi-Direction

e. Exchange Options: Create New Records

f. Tabular Join – GIS ID Mapping Field: TT_ID

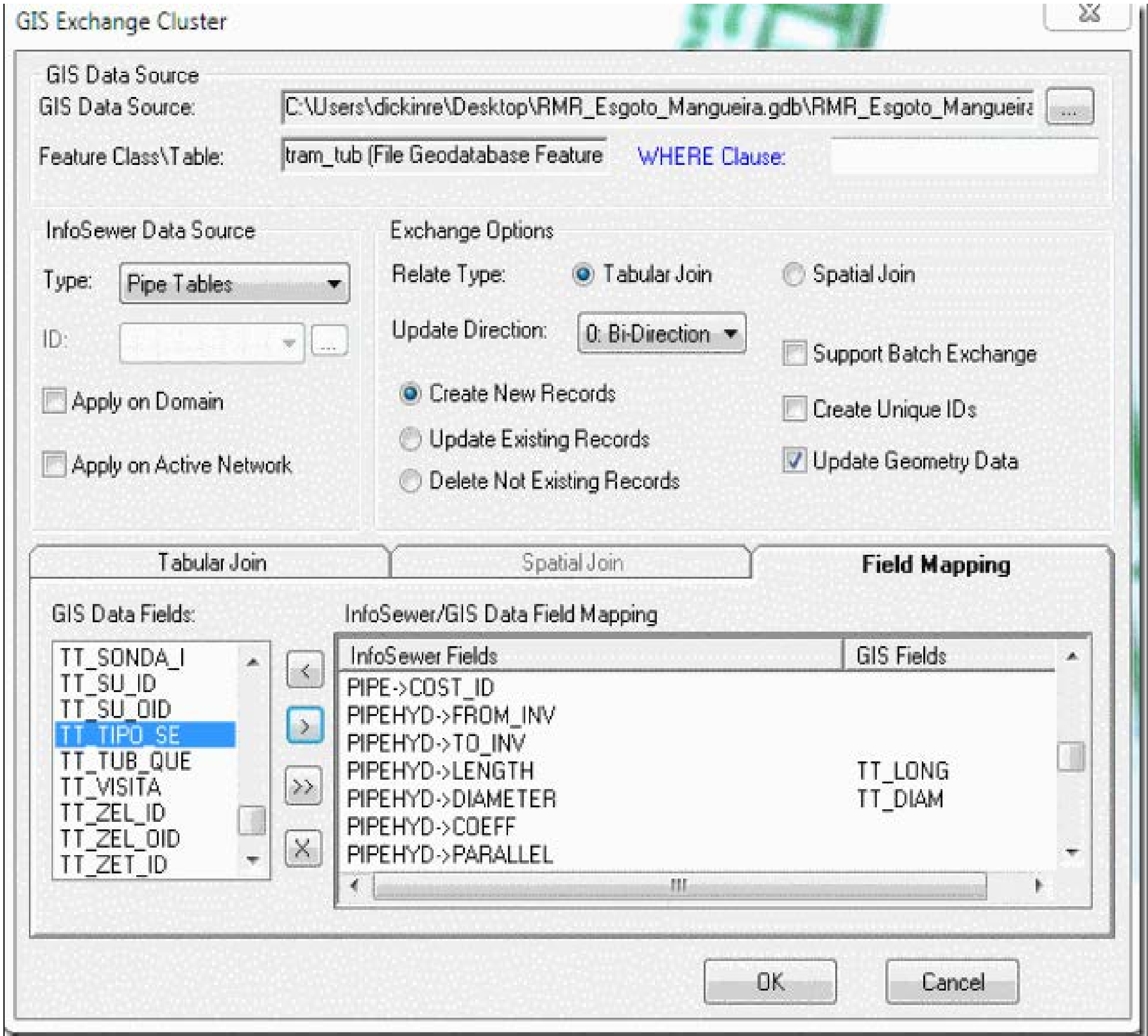
g. Field Mapping:

i. PIPE->TYPE : TT_TIPO_SE

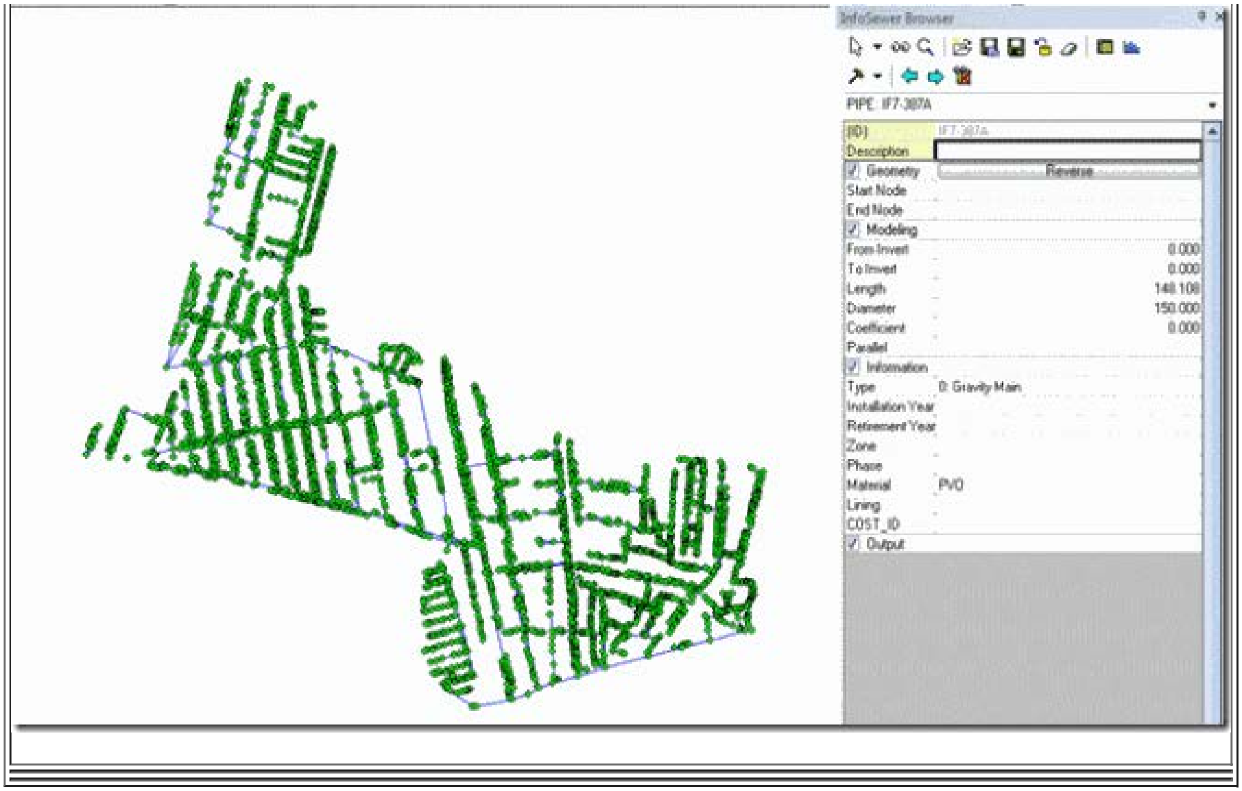
ii. PIPE->MATERIAL : TT_MATERIA

iii. PIPEHYD->LENGTH : TT_LONG

iv. **PIPEHYD->DIAMETER:** TT_DIAM



Load the Links from the GIS Gateway and you should have your links on the Map and in the Attribute Browser, I did not import the From and To Nodes and the From and To Invert as I was not certain of the Attribute Tables.


InfoSewer Browser
PIPE: IF7-387A
(ID)
IF7-387A
Description
Geometry
Reverse
Start Node
End Node
Modeling
From Invert
0.000
To Invert
0.000
Length
148.108
Diameter
150.000
Coefficient
0.000
Parallel
Information
Type
0: Gravity Main
Installation Year
Retirement Year
Zone
Phase
Material
PVC
Lining
COST_ID
Output

InfoSewer™

Modeling Made Easy

InfoSewer Flow Attenuation Sensitivity

The Effect of the Flow Attenuation Option in infoSewer and H2OMAP Sewer

The three Run manager parameters, Maximum Number of Segments, Minimum Travel Distance and the Minimum Travel Distance in InfoSewer and H2OMAP Sewer affect the shape and flow attenuation of the flow in a link. The effect of using the flow attenuation is to reduce the peak flow and spread out the flow compared to the No Flag option (Figure 1).

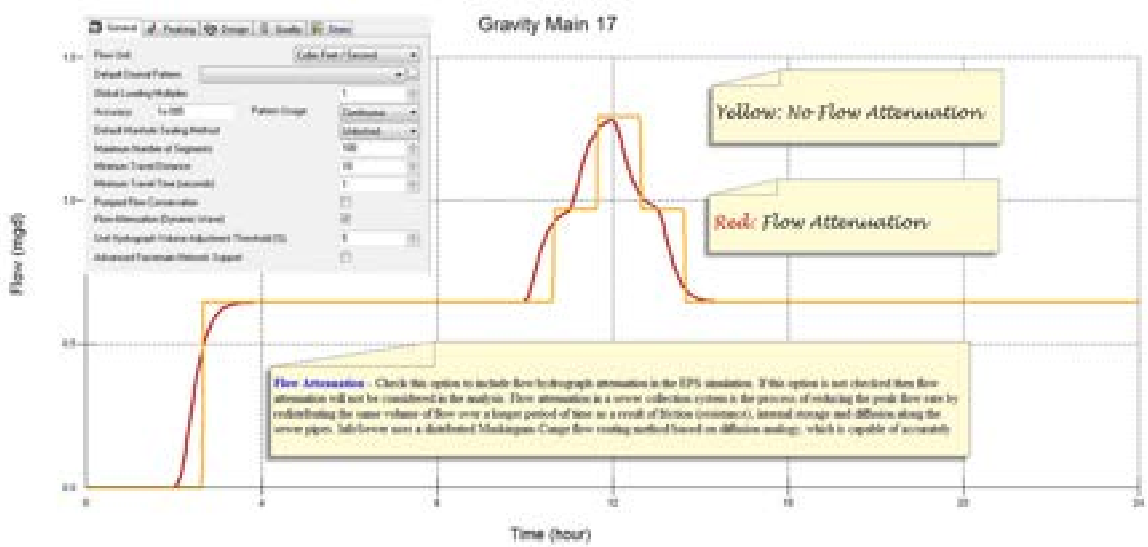


Figure 1. Effect of the Flow Attenuation Option in infoSewer and H2OMAP Sewer

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Siphon Modeling in InfoSewer and Arc Map

Siphon Modeling in InfoSewer and Arc Map

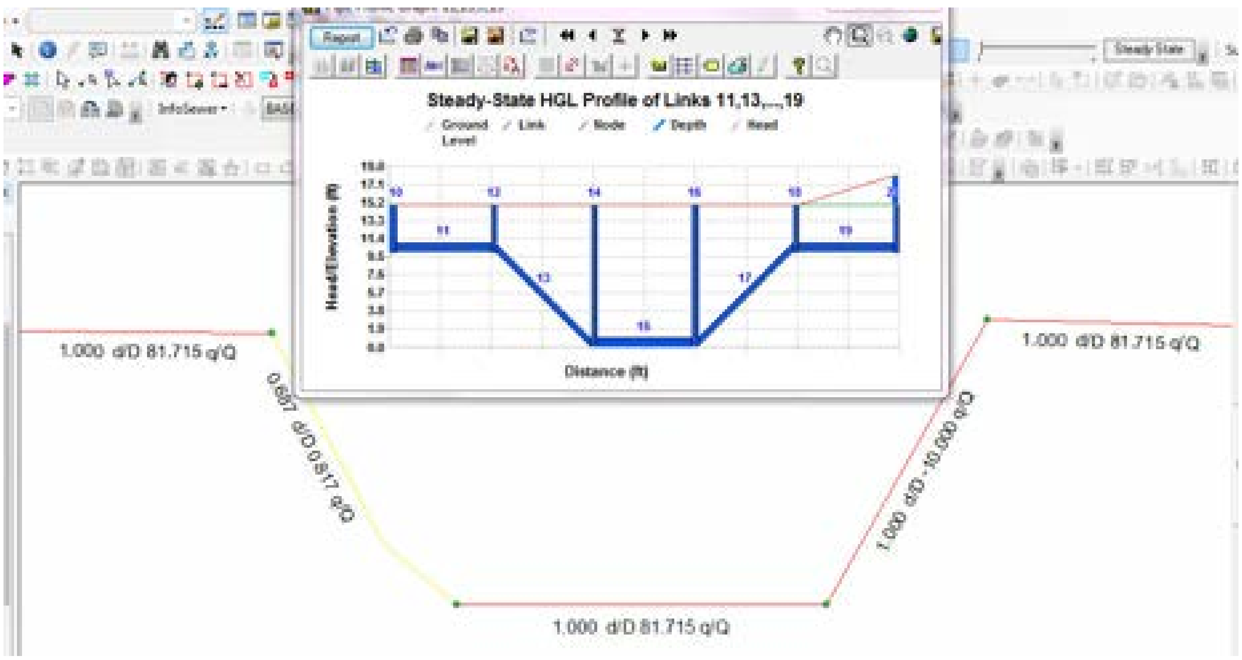
Here is at typical view of a siphon in InfoSewer, the rising links have a d/D of 1 and the dropping links typically have a d/D value less than 1. The q/Q for the rising links is based on the slope and the dropping links typically have a q/Q less than 1 as the full flow is high due to the steep slopes. You can show both the d/D and q/Q values in an Arc Map Display by using the following code. I find it easier to first plot one variable using the InfoSewer Map Display followed by editing the TOC Properties for Pipes to show more than one variable at a time in the Label properties.

Function FindLabel ([D_OVER_D], [Q_OVER_Q])

If IsNull([D_OVER_D]) then Exit Function

FindLabel ="" & FormatNumber([D_OVER_D],3) & " d/D " & "" & FormatNumber([Q_OVER_Q],3) & " q/Q"

End Function



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™
Modeling Made Easy

InfoSewer Flow Attenuation Sensitivity

InfoSewer Maximum Number of Segments Sensitivity

The three Run manager parameters, Maximum Number of Segments, Minimum Travel Distance and the Minimum Travel Distance in InfoSewer and H2OMAP Sewer affect the shape and flow attenuation of the flow in a link. The effect of altering the Maximum Number of Segments is to reduce the peak flow and spread out the flow as the number of segments increases(Figure 1).

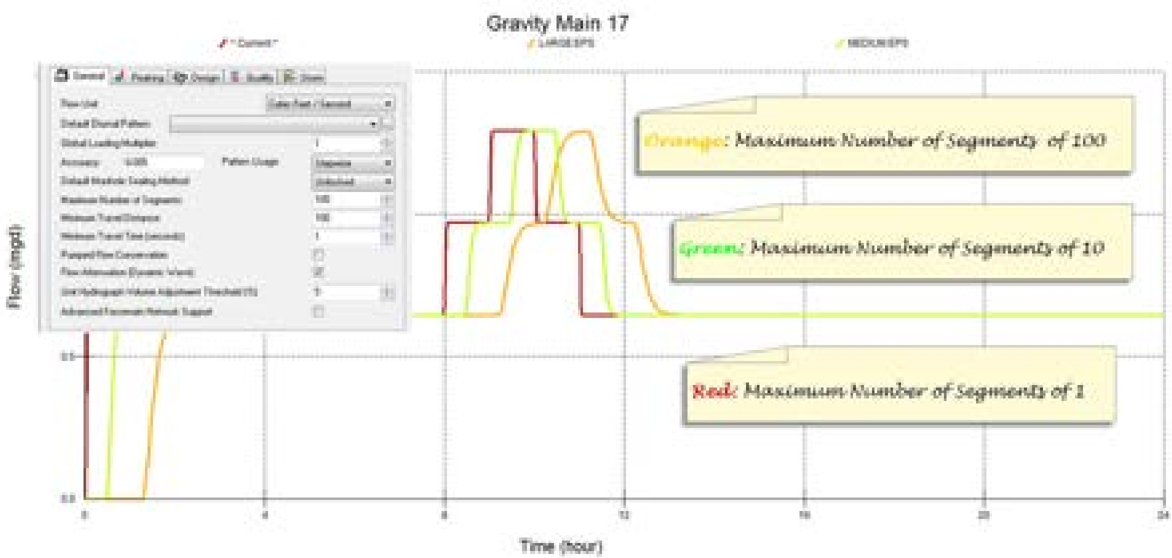


Figure 1. Effect of the Maximum Number of Segments in infoSewer and H2OMAP Sewer

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

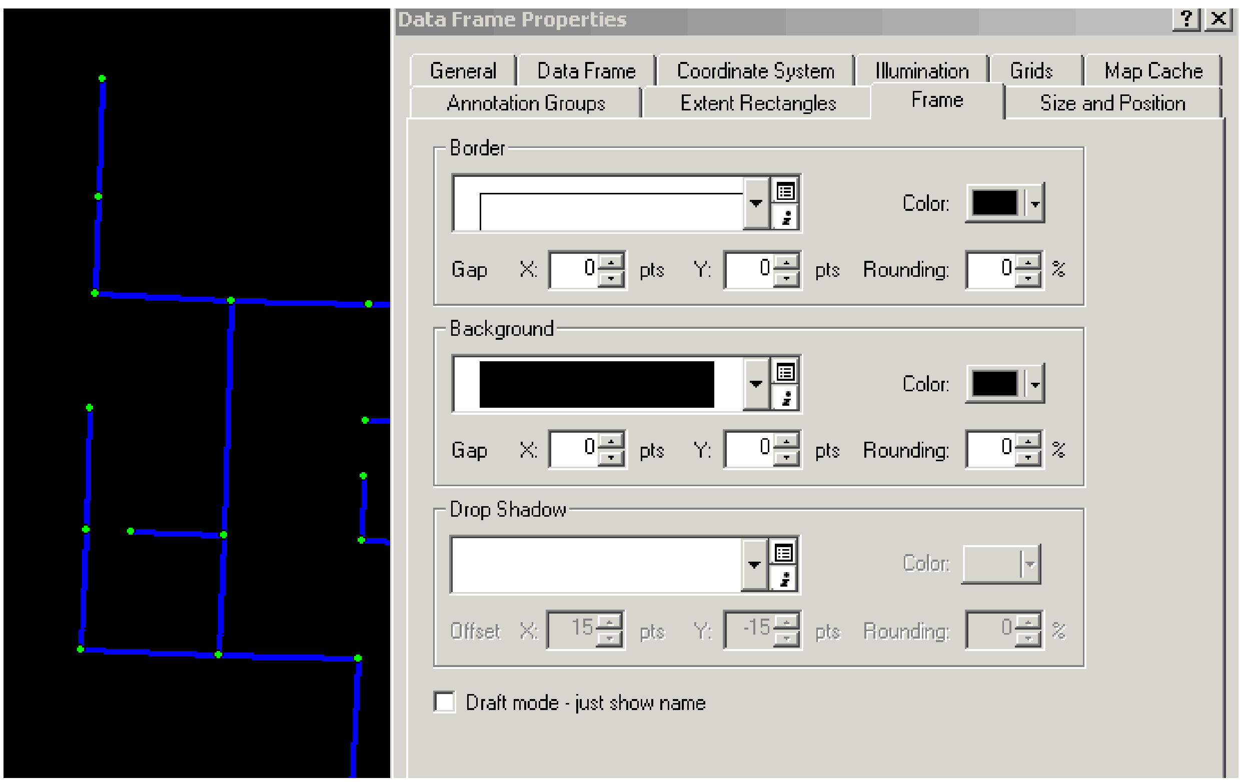
Home > InfoSewer Help File and User Guide > FAQ and Blogs > How to change the background color and data view in InfoSewer and InfoSWMM

How to change the background color and data view in InfoSewer and InfoSWMM

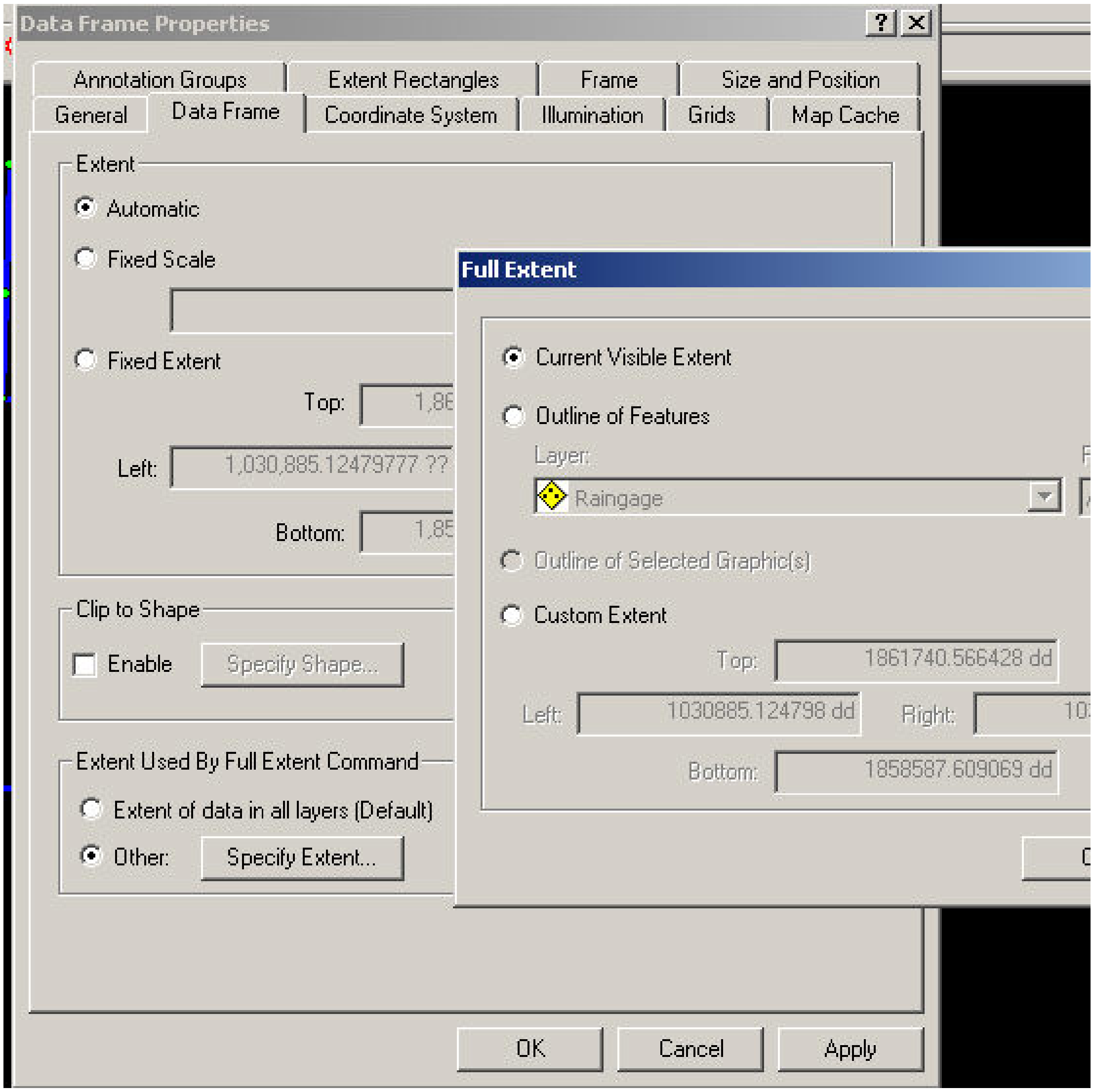
How to change the background color and data view in InfoSewer and InfoSWMM

Subject: How to change the background color and data view in InfoSewer and InfoSWMM

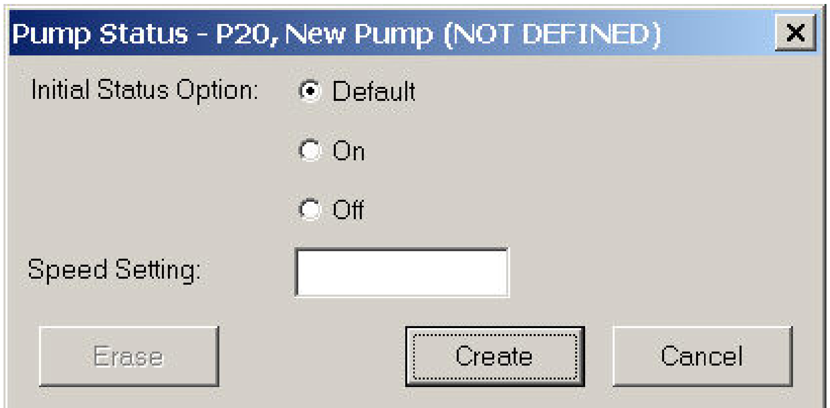
Tip 1: Use the command View> Data Frame Properties > Frame > Background (change color) to change the background color



Tip 2: Use the command View> Data Frame Properties > Data Frame > Extent to change the default view in Arc GIS. You would use this tool if you have zoom to a small point in InfoSWMM and InfoSewer.



Tip 3: Use the command Tools/Preferences/Map to change the Background color in H_2OMap SWMM and H_2OMap Sewer.



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

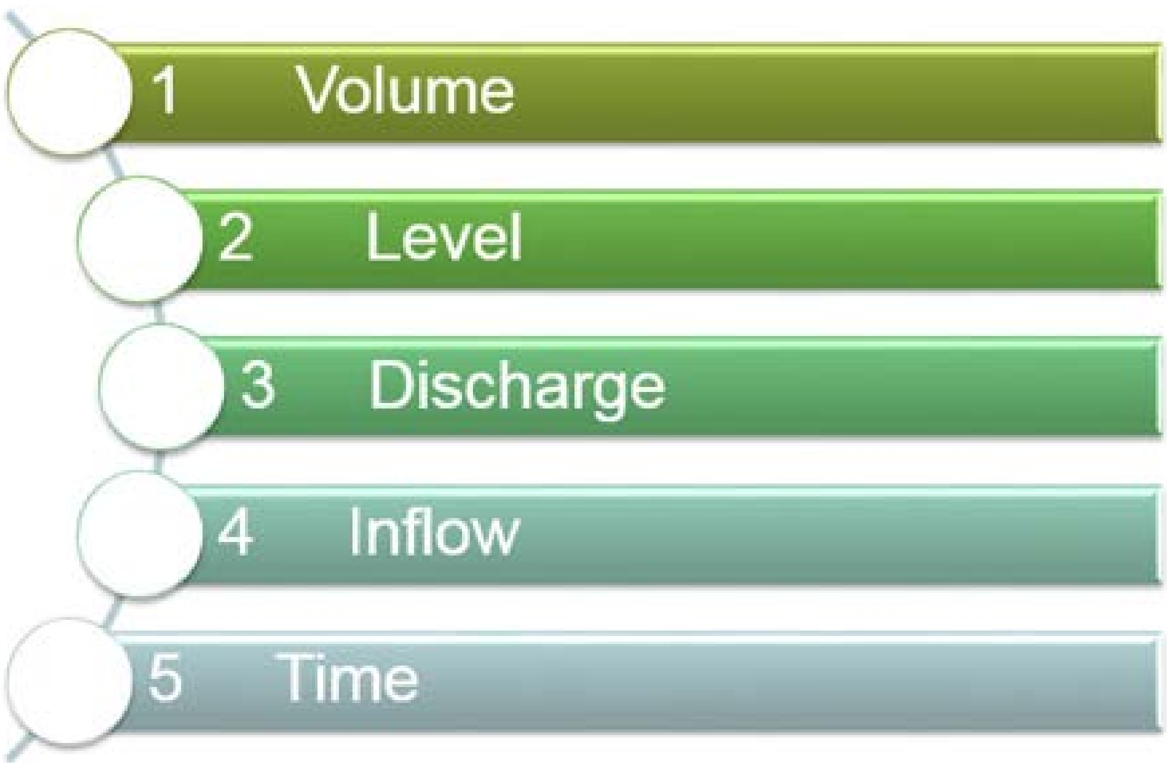
More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer Inflow Control for a Pump with a Pump Curve

InfoSewer Inflow Control for a Pump with a Pump Curve

You can control the pumps in InfoSewer and H2OMap Sewer by using a Pump Control which will control the pump based on:

1. Volume
2. Level
3. Discharge
4. Inflow
5. Time



If you use a By Inflow control the pump speed of the pump is increased or decreased to make the Upstream Wet Well Level Constant (Figure 1) for an exponential 3 point curve

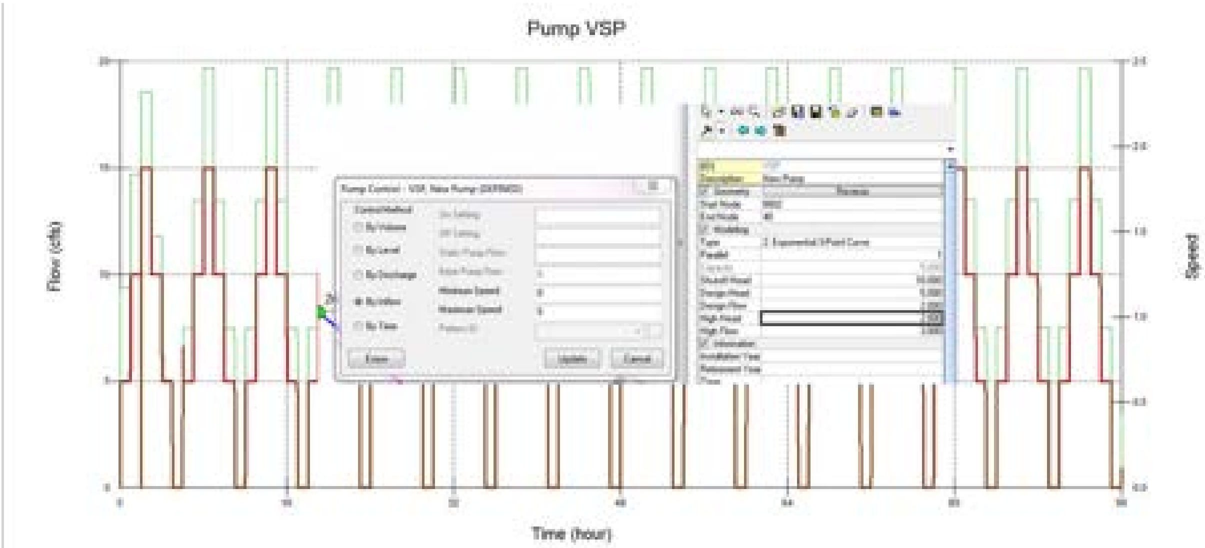


Figure 1. Inflow Control for PUMP in InfoSewer and H2OMAP Sewer will change the Pump Speed of the pump to make the Wet Well level constant

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

InfoSewer By Discharge Control for a PUMP

InfoSewer By Discharge Control for a PUMP

You can control the pumps in InfoSewer and H2OMap Sewer by using a Pump Control which will control the pump based on:

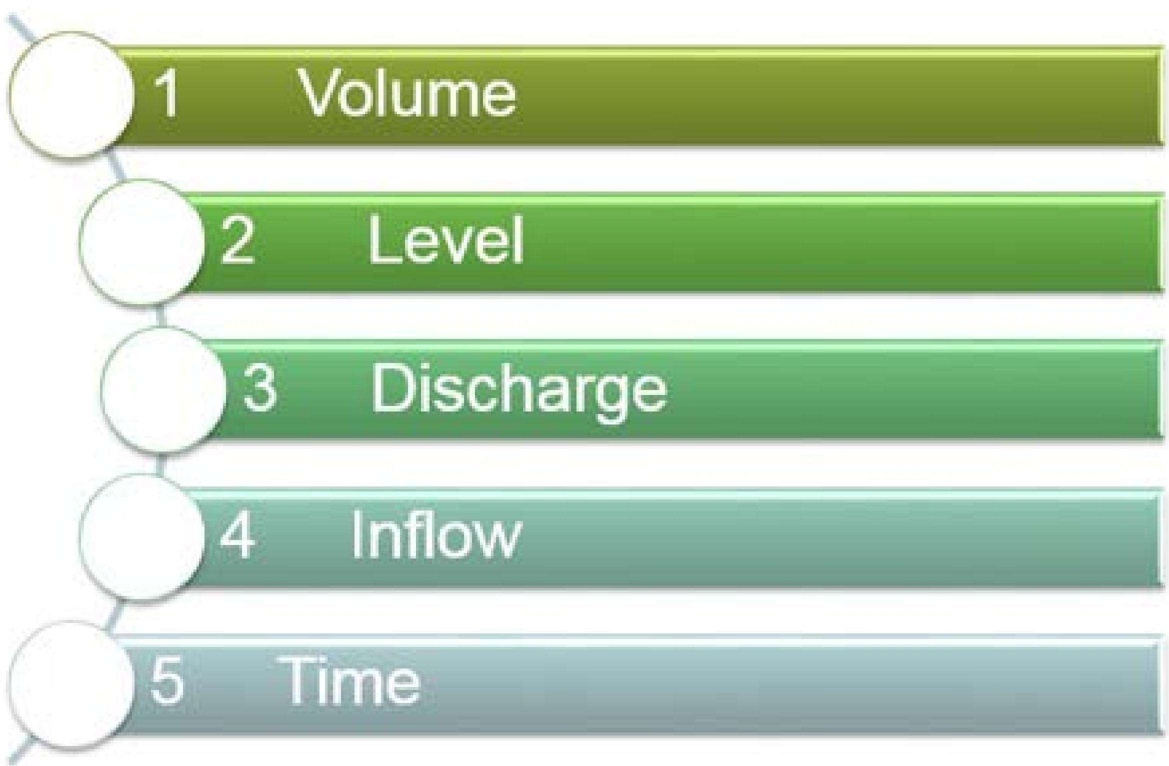
1. Volume

2. Level

3. Discharge

4. Inflow

5. Time



If you use a By Discharge control the pump speed of the pump is increased or decreased to pump the incoming Wet Well flow based on the pump rules and the geometry of the Wet Well (Figure 1).

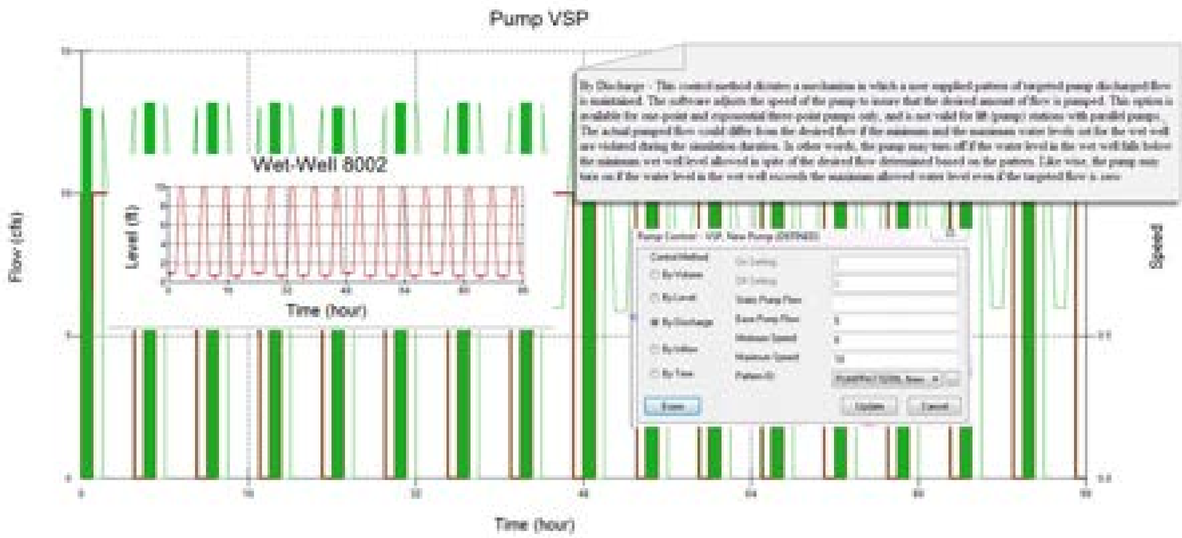


Figure 1. By Discharge Control for PUMP in InfoSewer and H2OMAP Sewer will change the Pump Speed of the pump to follow the Base Pump Flow Rules.

Tags INFOSEWER CONTROL

InfoSewer™

Modeling Made Easy

InfoSewer Inflow Control for a PUMP

InfoSewer Inflow Control for a PUMP

You can control the pumps in InfoSewer and H2OMap Sewer by using a Pump Control which will control the pump based on:

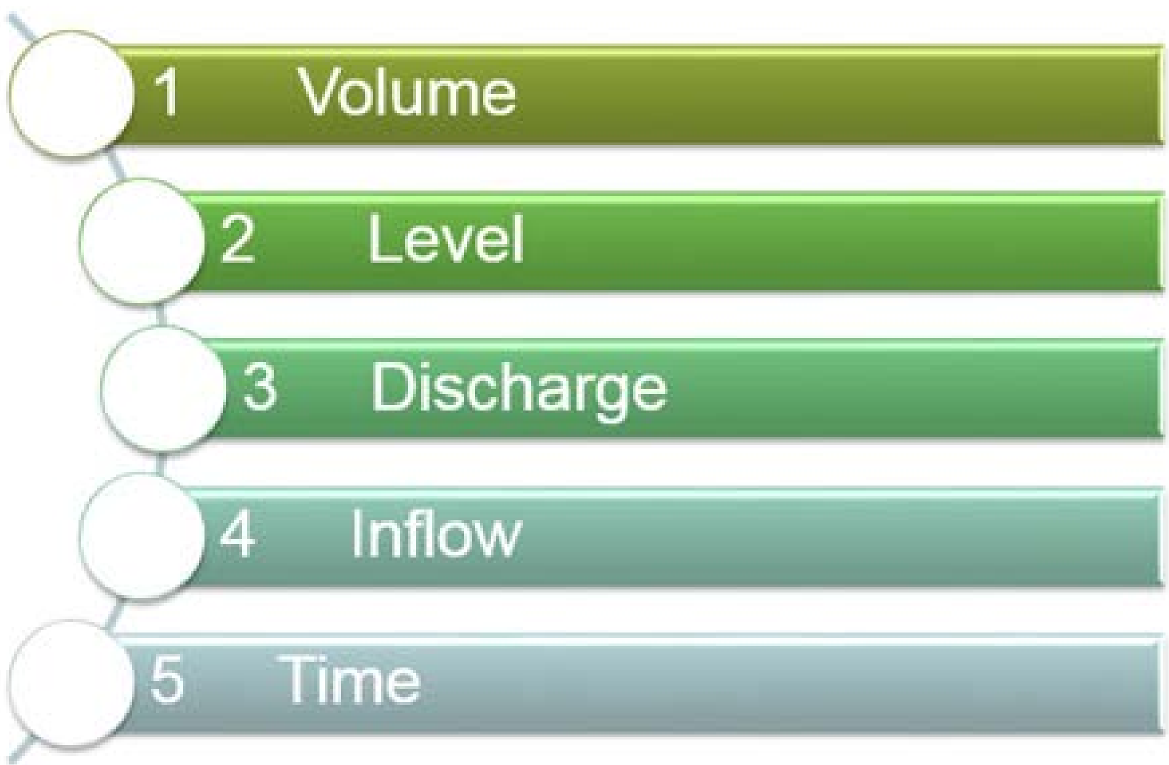
1. Volume

2. Level

3. Discharge

4. Inflow

5. Time



If you use a By Inflow control the pump speed of the pump is increased or decreased to make the Upstream Wet Well Level Constant (Figure 1).

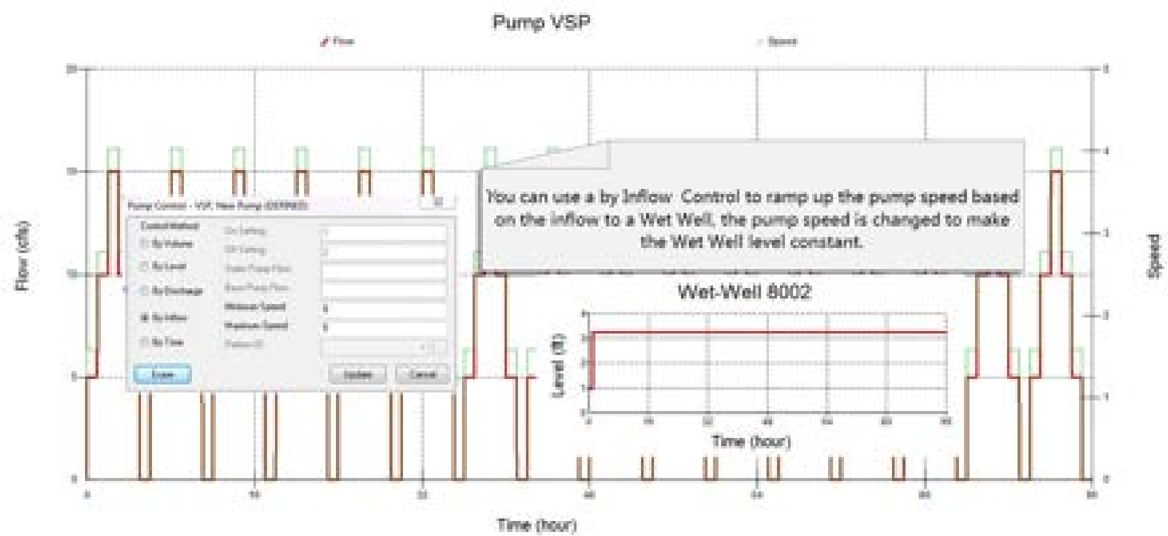


Figure 1. Inflow Control for PUMP in InfoSewer and H2OMAP Sewer will change the Pump Speed of the pump to make the Wet Well level constant

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

InfoSewer™

Modeling Made Easy

InfoSewer Minimum Travel Distance Sensitivity

InfoSewer Minimum Travel Distance Sensitivity

The three Run manager parameters, Maximum Number of Segments, Minimum Travel Distance and the Minimum Travel Distance in InfoSewer and H2OMAP Sewer affect the shape and flow attenuation of the flow in a link. The effect of decreasing the Minimum Travel Distance is to reduce the peak flow and spread out the flow as the number of segments increases **(Figure 1).** The smaller the minimum travel distance, which has the effect of increasing the number of segments in a link up the limit of the parameter Maximum Number of segments, the smaller the peak and the more attenuation of the flow in InfoSewer.

There is three ways to control attenuation in InfoSewer: (1) use the flow attenuation option, (2) increase the Maximum Number of Segments per link and (3) decrease the Minimum travel distance. You can also use all three parameters to make more segments per link for long links and only a few segments for short links.

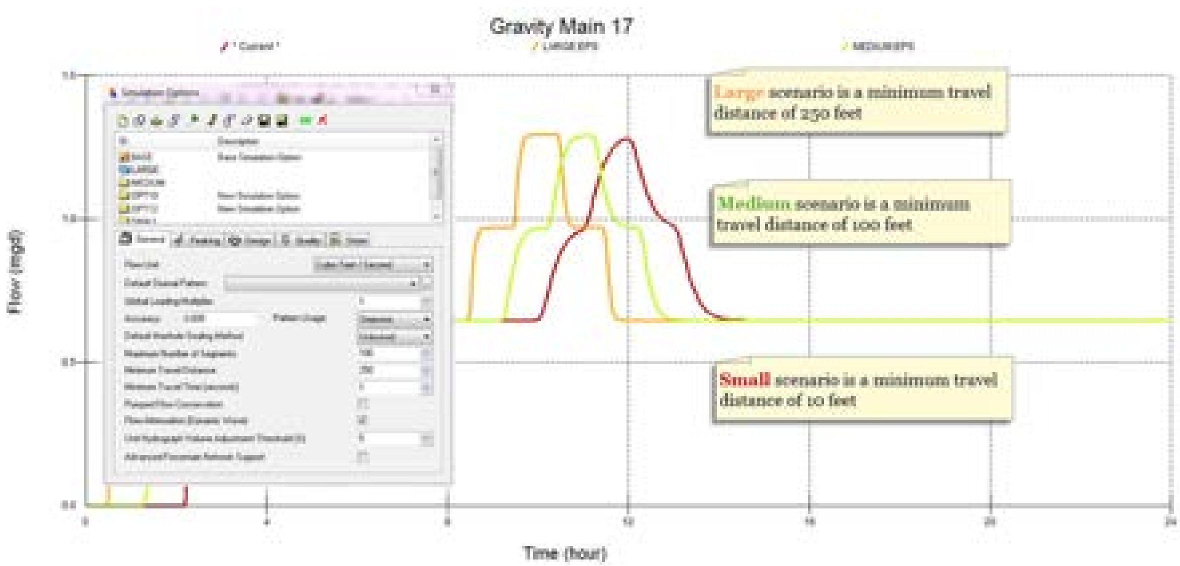


Figure 1. Effect of the Minimum Travel Distance in **InfoSewer**

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer Minimum Travel Time Sensitivity

InfoSewer Minimum Travel Time Sensitivity

The three Run manager parameters, Maximum Number of Segments, Minimum Travel Distance and the Minimum Travel Time in InfoSewer and H2OMAP Sewer affect the shape and flow attenuation of the flow in a link. The effect of decreasing the Minimum Travel Time is to reduce the peak flow and spread out the flow as the number of segments increases **(Figure 1).** The larger the Minimum Travel Time, which has the effect of decreasing the number of segments in a link up the limit of the parameter Maximum Number of segments, the smaller the peak and the more attenuation of the flow in InfoSewer.

There is three ways to control attenuation in InfoSewer: (1) use the flow attenuation option, (2) increase the Maximum Number of Segments per link and (3) decrease the Minimum travel distance. You can also use all three parameters to make more segments per link for long links and only a few segments for short links.

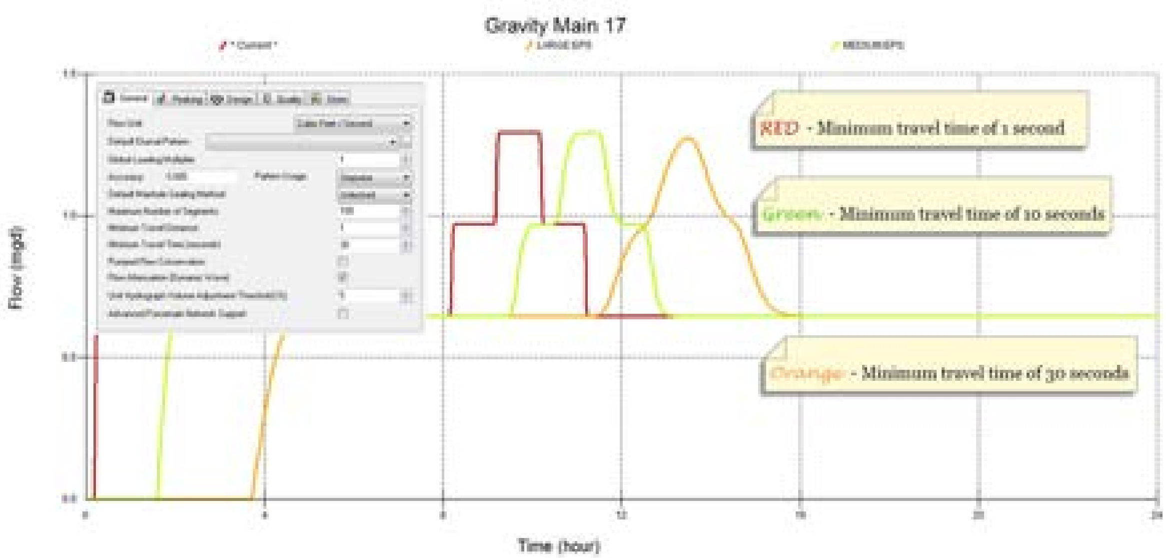


Figure 1. Effect of the Minimum Travel Time in **InfoSewer**

InfoSewer Maximum Number of Segments Sensitivity

InfoSewer Maximum Number of Segments Sensitivity

The three Run manager parameters, Maximum Number of Segments, Minimum Travel Distance and the Minimum Travel Distance in InfoSewer and H2OMAP Sewer affect the shape and flow attenuation of the flow in a link. The effect of altering the Maximum Number of Segments is to reduce the peak flow and spread out the flow as the number of segments increases **(Figure 1).**

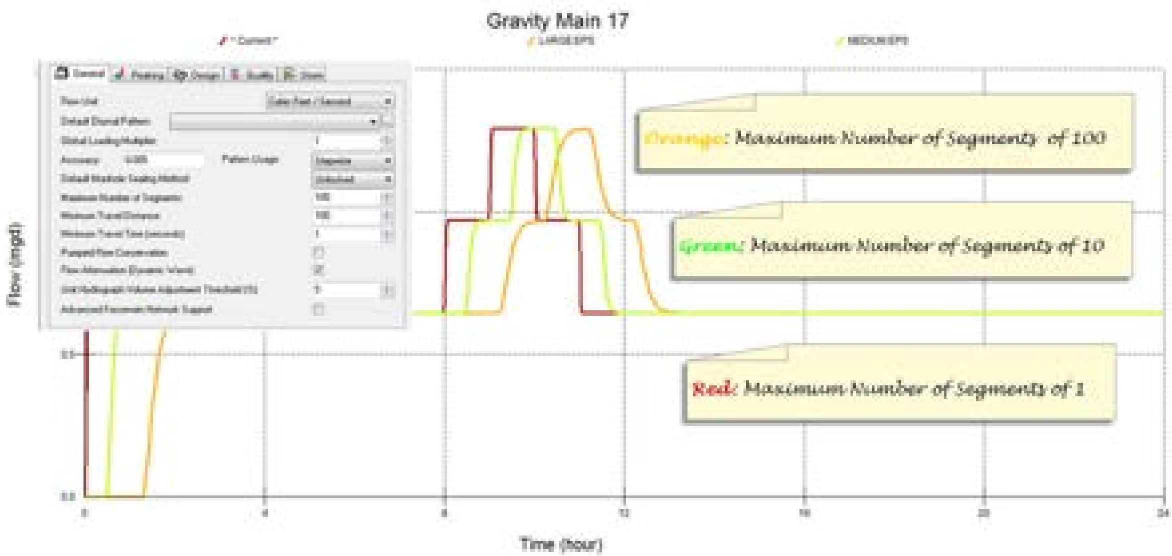


Figure 1. Effect of the Maximum Number of Segments in infoSewer and H2OMAP Sewer

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

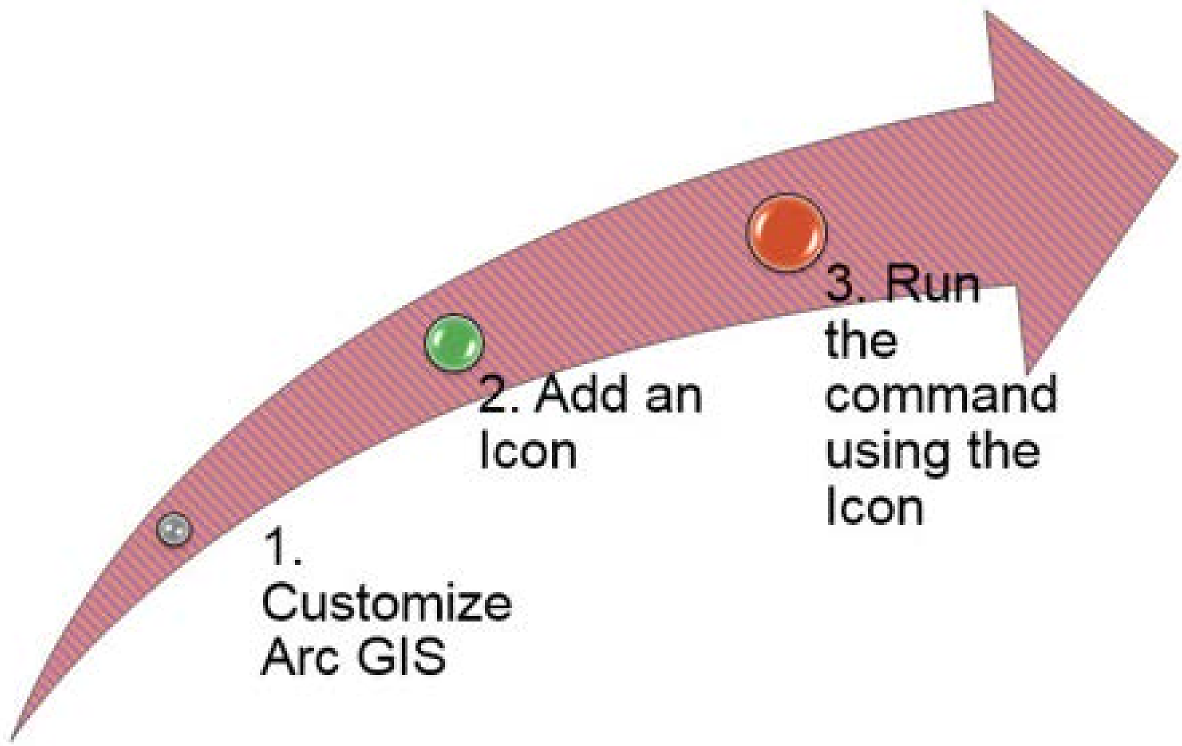
More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

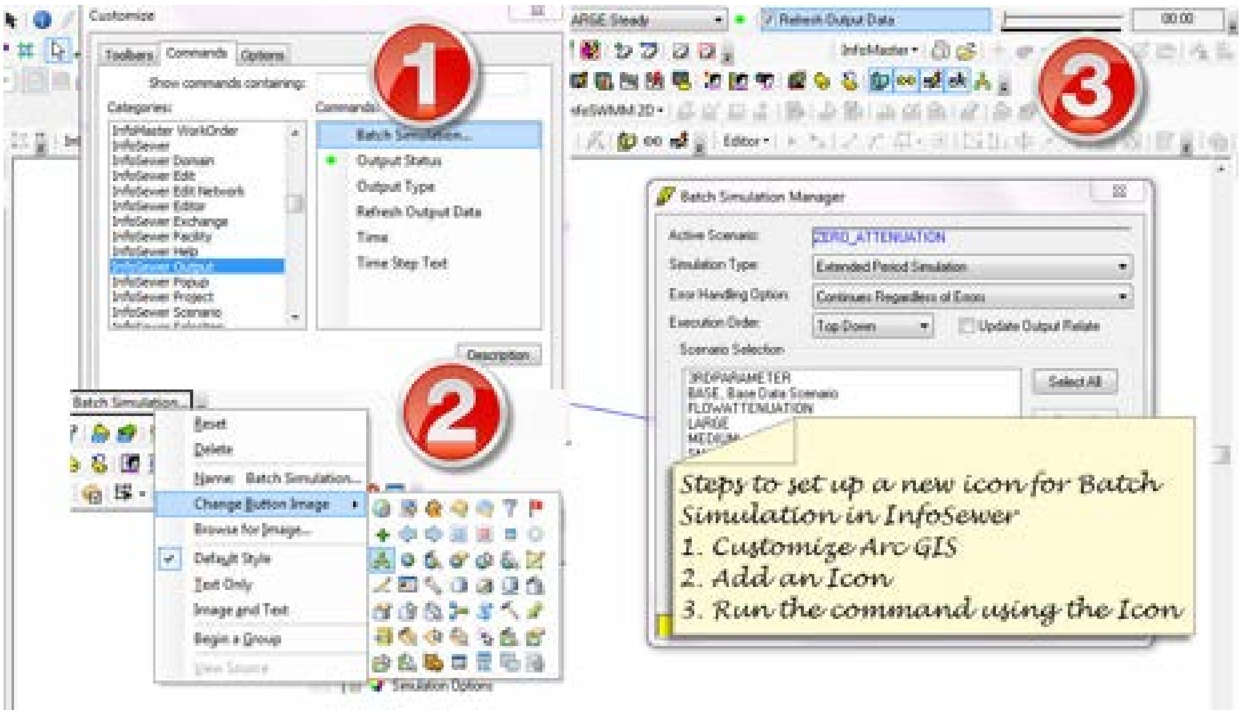
Batch Simulation in InfoSewer, InfoSWMM or InfoWater

Batch Simulation in InfoSewer, InfoSWMM or InfoWater

Steps to set up a new icon for Batch Simulation in InfoSewer, InfoSWMM or InfoWater

1. Customize Arc GIS
2. Add an Icon
3. Run the command using the Icon





InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

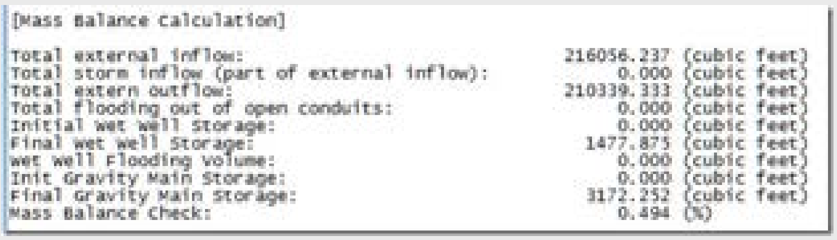
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

How to Use Trace Upstream, Domain Manager and Facility Manager in InfoSewer to Find the CE

How to Use Trace Upstream, Domain Manager and Facility Manager in InfoSewer to Find the CE

InfoSewer does not have table of node continuity errors only an overall continuity error balance. If you have a conti then you can use the process of divide and conquer to find the location of the continuity error. Start at the Outlets an Trace Upstream command, Domain Manager and Facility Manager take out whole sections of the network until you section of the network with the continuity error (Figure 1 and Figure 2).



```
[Mass Balance Calculation]

Total external inflow:                              216056.237 (cubic feet)
Total storm inflow (part of external inflow):            0.000 (cubic feet)
Total extern outflow:                               210339.333 (cubic feet)
Total flooding out of open conduits:                     0.000 (cubic feet)
Initial wet well storage:                                0.000 (cubic feet)
Final wet well storage:                               1477.875 (cubic feet)
Wet well flooding volume:                                0.000 (cubic feet)
Init Gravity Main Storage:                               0.000 (cubic feet)
Final Gravity Main Storage:                           3172.252 (cubic feet)
Mass Balance Check:                                      0.494 (%)
```

Figure 1. The Mass Balance Calculation in InfoSewer/H2OMap Sewer.

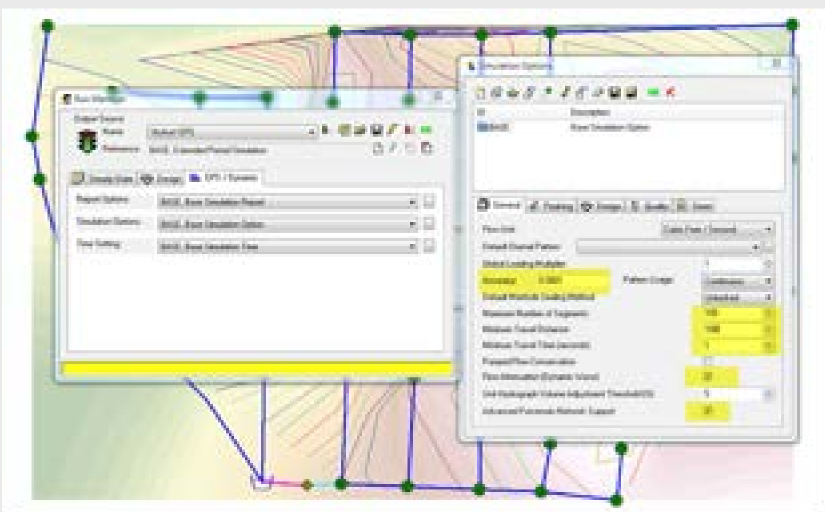


Figure 2. Control the Continuity Error with the Run Control Options in Run Manager.

Here are the steps you can take:

Step 1. Use Trace Upstream Network to find the and place in a Domain the Upstream Network (Figure 3).

Step 2. Once the upstream domain is created use the Domain Manager to add in any extra links without nodes (Figure 4)

Step 3. Make the Domain Inactive using Facility Manger (Figure 5)

Step 4. Run the network and check the overall continuity error (Figure 6)

Step 5. Continue and repeat until you isolate the area that is the main source of the Continuity Error (CE).

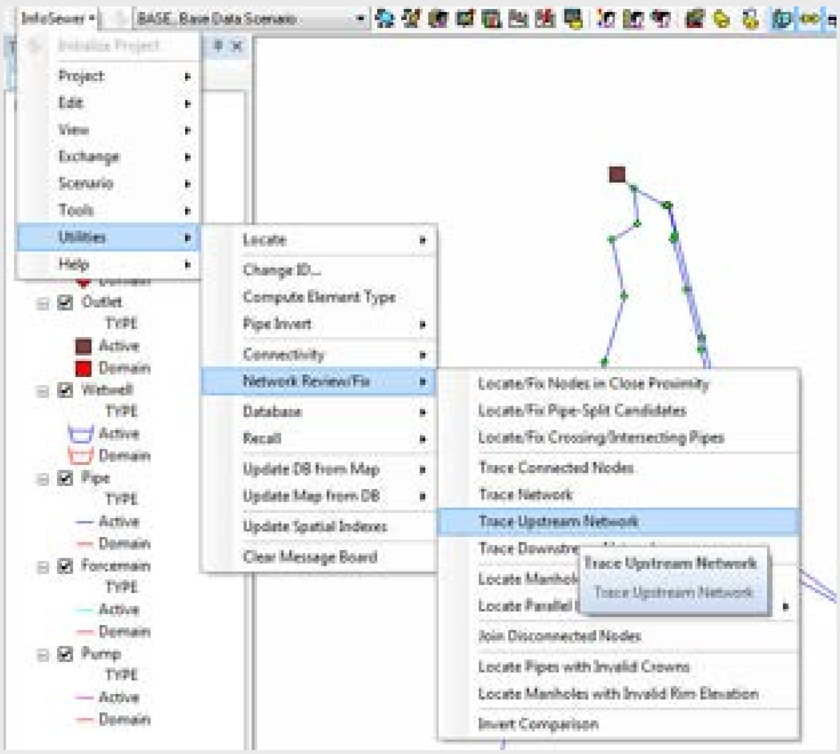


Figure 3. Trace Upstream Network and Place it in a Domain

Figure 4. Use Domain Manager to take out links without nodes

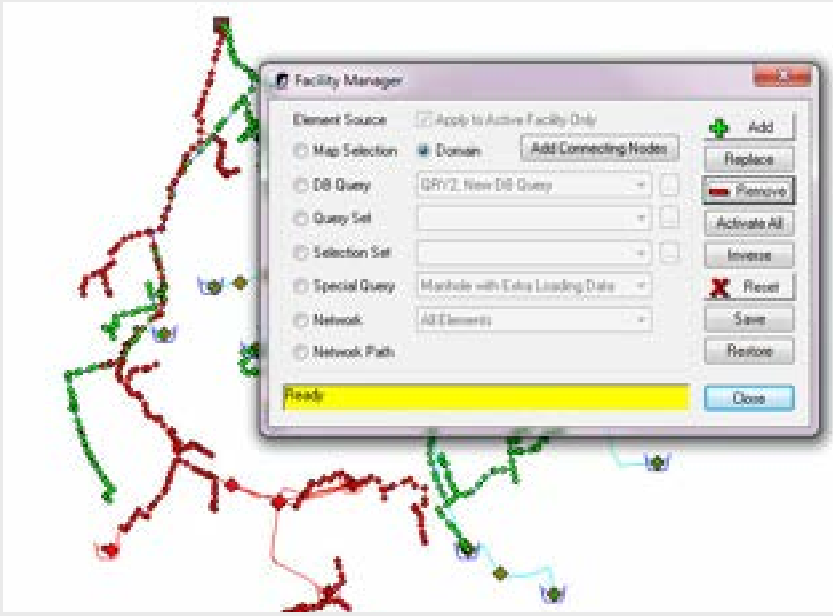


Figure 5. Use Facility Manager to Make the Domain Inactive

Figure 6. Find and Isolate the Area with the CE.

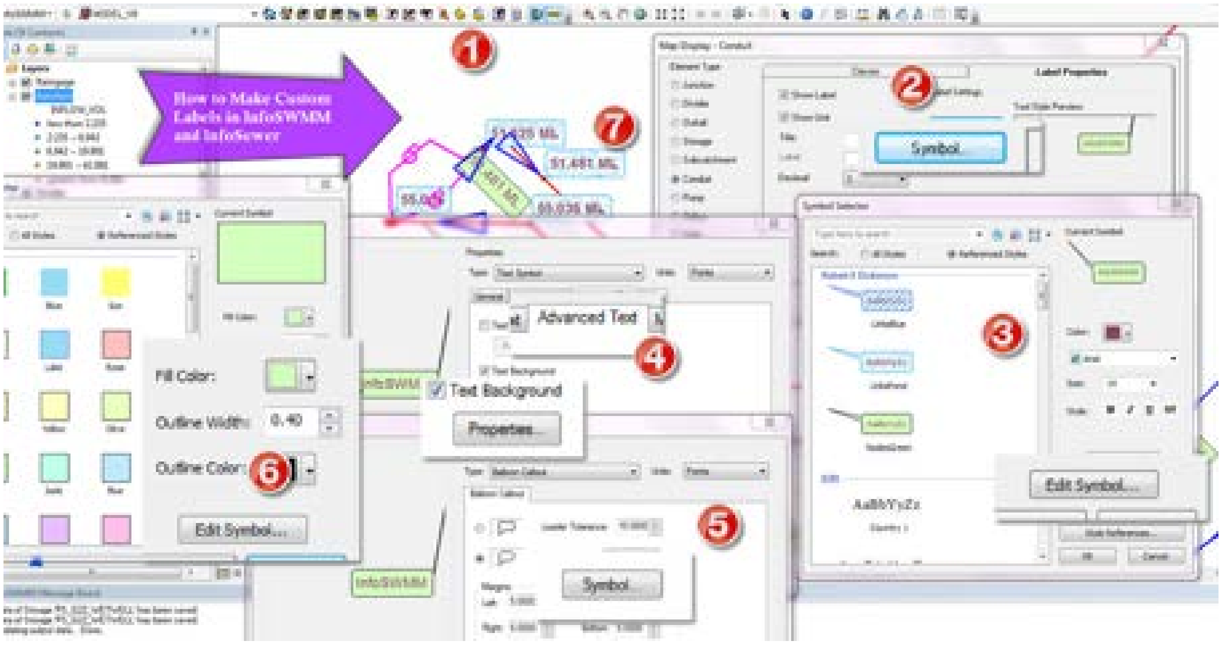
InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

---

How to Make Custom Labels in InfoSWMM and InfoSewer



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

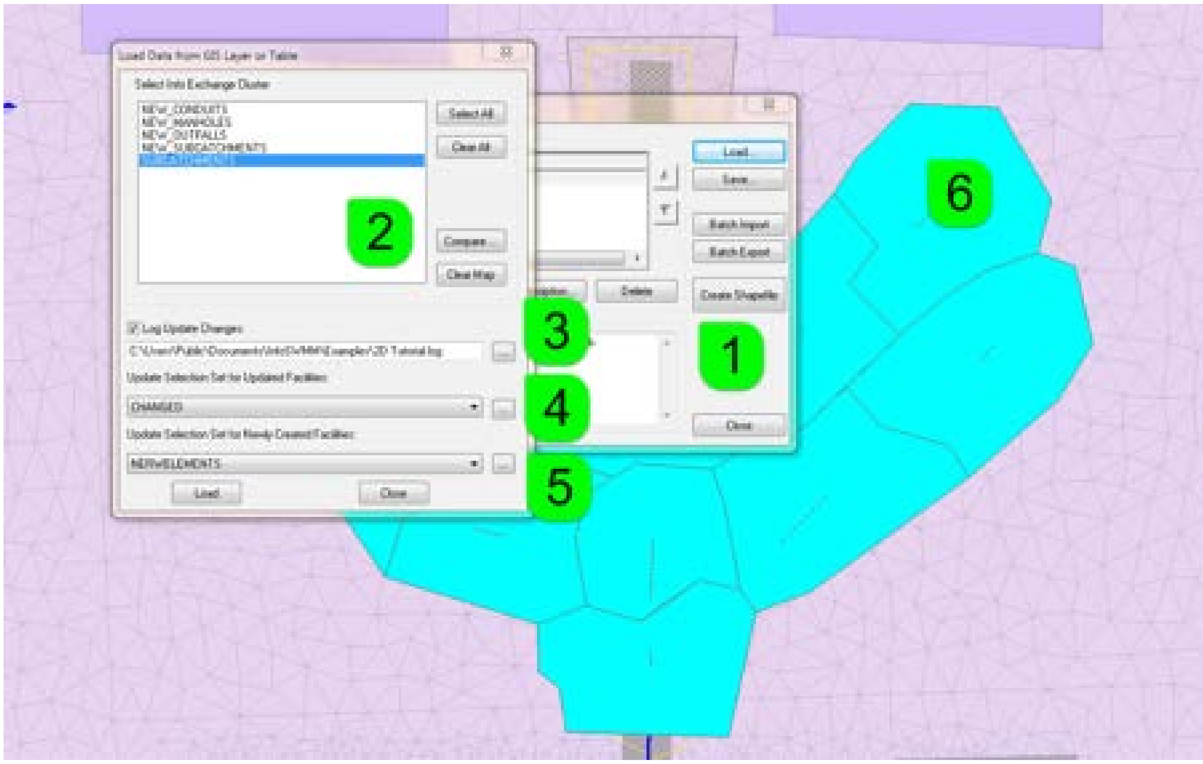
New GIS Gateway Features in InfoSewer and InfoSWMM

New GIS Gateway Features:

- New comparison function to compare data of InfoSWMM or InfoSewer and GIS.
- New logging function to allow users to track the data changes.
- Allows users to save updated data to the selection sets
- Allows users to save changed data to the selection sets
- Map of Changed Features

Image Features:

1. Create a Shapefile with Base GIS information or use GIS files directly
2. Compare Button to compare the current InfoSWMM or InfoSewer data to the GIS Layer
3. A log file is made with changed features
4. A selection set is made of Updated Facilities
5. A selection set is made of New Facilities
6. Clear Map to Clear the Changed Facilities Map


Select All
Clear All
Compare...
Clear Map
Log Update Changes
Load...
Save...
Batch Import
Batch Export
Create Shapefile
Delete
Close
Load
1
2
3
4
5
6

InfoSewer™
Modeling Made Easy

Advanced Force Main Solution and Gravity Main Attenuation in InfoSewer for better Pump, Force Main, Gravity Main Simulations

This blog is about using the Advanced Force Main Solution and Gravity Main Attenuation in **InfoSewer** for better Pump, Force Main, Gravity Main Simulations

1. Select Advanced Force Main Solution and Flow Attenuation the Run Manager
2. The overall Continuity Error will be Better
3. Gravity mains will be closer to the Force Main Flows
4. Force Main Flows will be closer to the Pump flows
5.

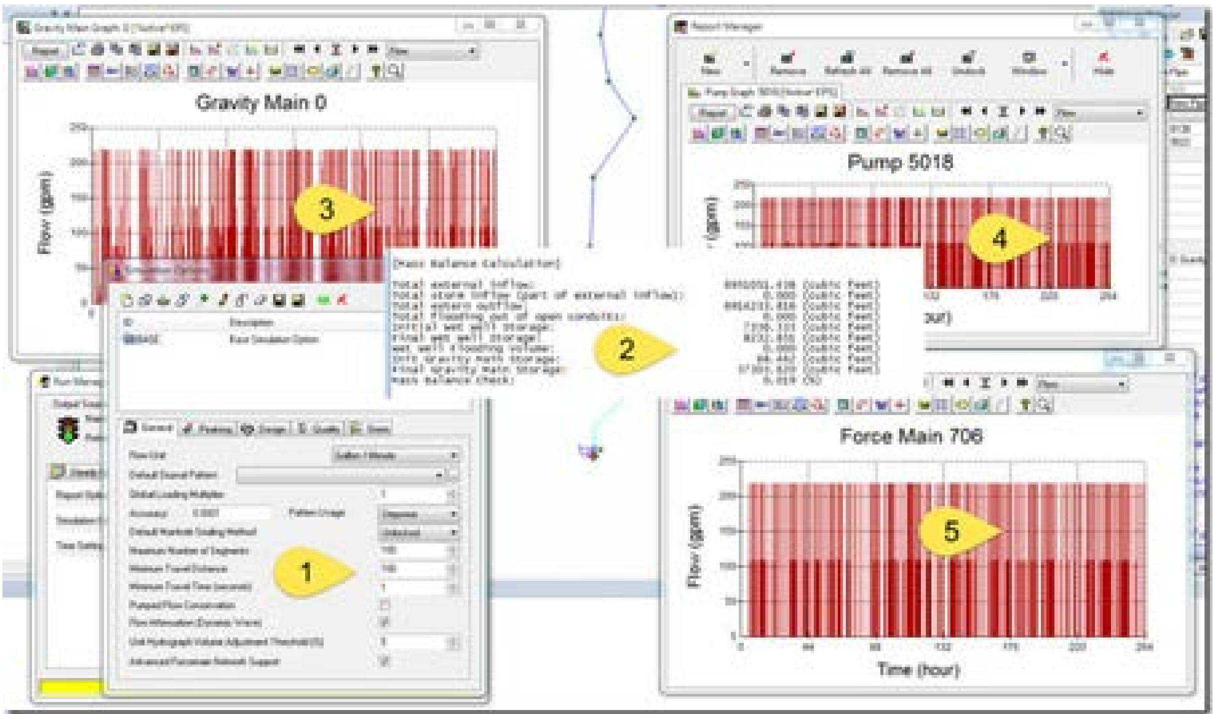
Select Advanced Force Main Solution and Flow Attenuation the Run Manager	Force Main Flows will be closer to the Pump flows
The overall Continuity Error will be Better	Gravity mains will be closer to the Force Main Flows

-

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-



Advanced Force Main Solution and Gravity Main Attenuation in **InfoSewer** for better Pump, Force Main, Gravity Main Simulations

us/support-overview

InfoSewer™

Modeling Made Easy

How to Rebuild the IEDB Folder

This is how you can fix a projection issue. Please follow the instructions below to rebuild the IEDB folder. For the instructions below, Model1 is the original model and Model2 is the rebuilt model.

1. Open the original model MXD file. Do not initialize the MXD file.

2. Save the original model MXD file to a new name (Model2).

3. Remove element layers.

4. Press the red arrow to initialize Model2 and in the Create New InfoSewer Project dialog box, select Same as active map.

5. In the Create New InfoSewer Project dialog box, we would recommend in the Project GeoDatabase Type: File GeoDatabase.

6. Press the OK button.

7. Save Model2 and close ArcMap.

8. Copy everything except the Map folder from Model1.IEDB folder into Model2.IEDB folder. (Important: do not copy the Map folder.)

9. Open Model2.mxd and initialize the model.

10. Go to the InfoSewer dropdown menu, Utilities -> Update Map from DB -> Force Full Network to redraw the network.

11. Go to the InfoSewer dropdown menu, View -> Reset Display.

12. Go to the InfoSewer dropdown menu, Utilities -> Database -> Reindex.

13. Go to the InfoSewer dropdown menu, Utilities -> Database -> Pack.

14. Go to the InfoSewer dropdown menu, Utilities -> Database -> Clean.

15. Save the model.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

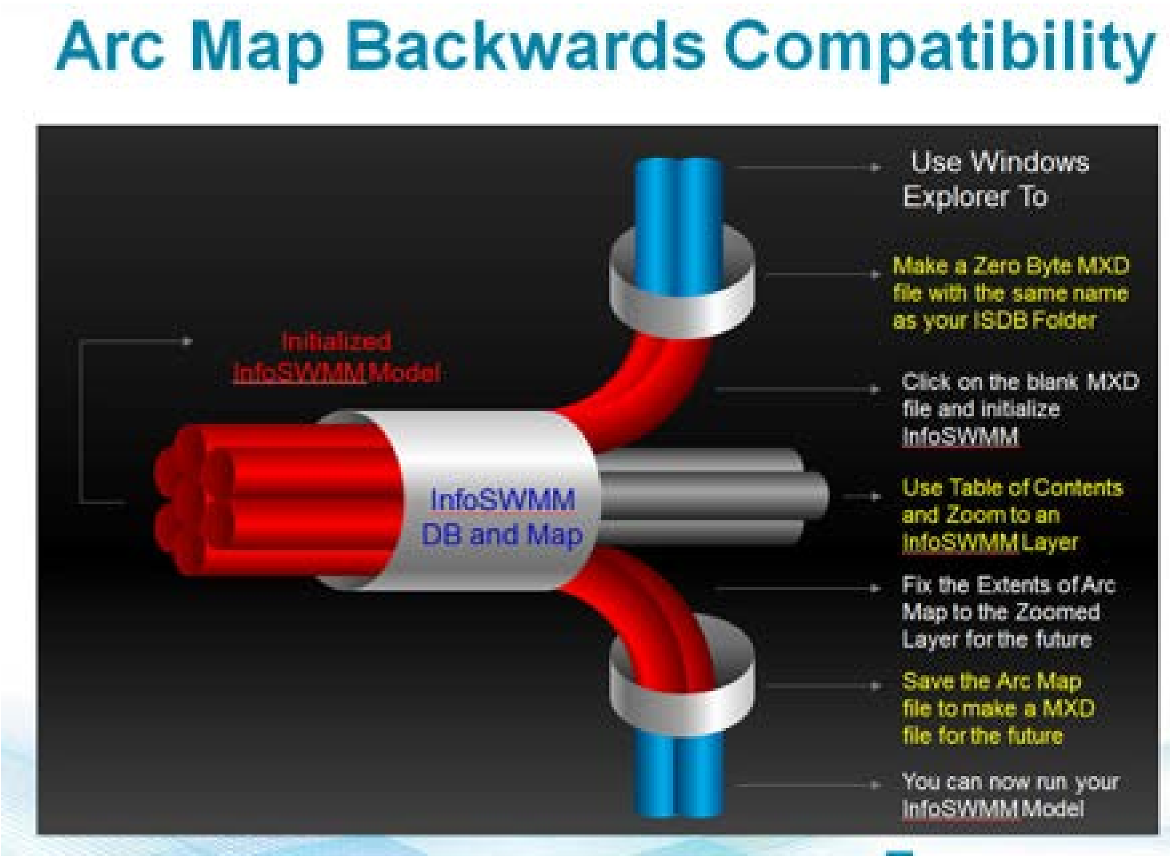
InfoSewer™

Modeling Made Easy

Arc Map Backwards Compatibility for InfoSewer

Arc Map Backwards Compatibility for InfoSWMM or InfoSewer

- Use windows explorer to, make a zero byte MXD file with the same name as your IEDB folder,
- click on the blank MXD file and initialize InfoSWMM or InfoSewer,
- Use the Table of contents to zoom to an InfoSWMM layer,
- fix the extents of the map in Arc GIS by using Data Frame,
- save the Arc Map to make a model in your version of Arc Map

1. 

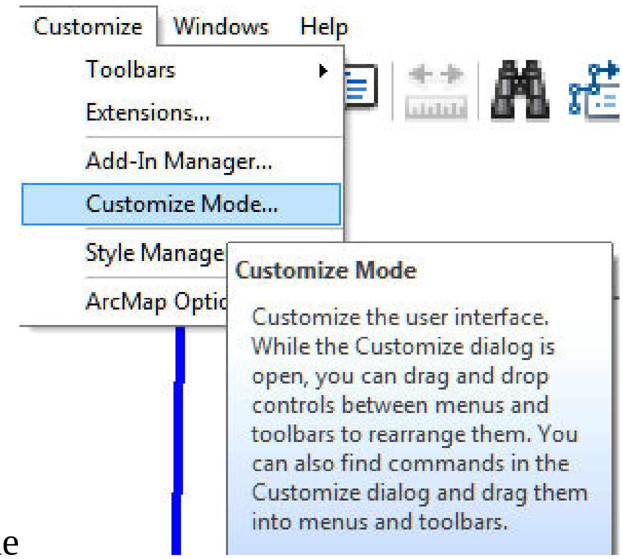
InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™
Modeling Made Easy

How to Make Large Icons in Arc GIS or Arc Map

Use the Customize mode 

in Arc GIS and pick Large Icons in the Options Tab. It really helps to see large Icons.

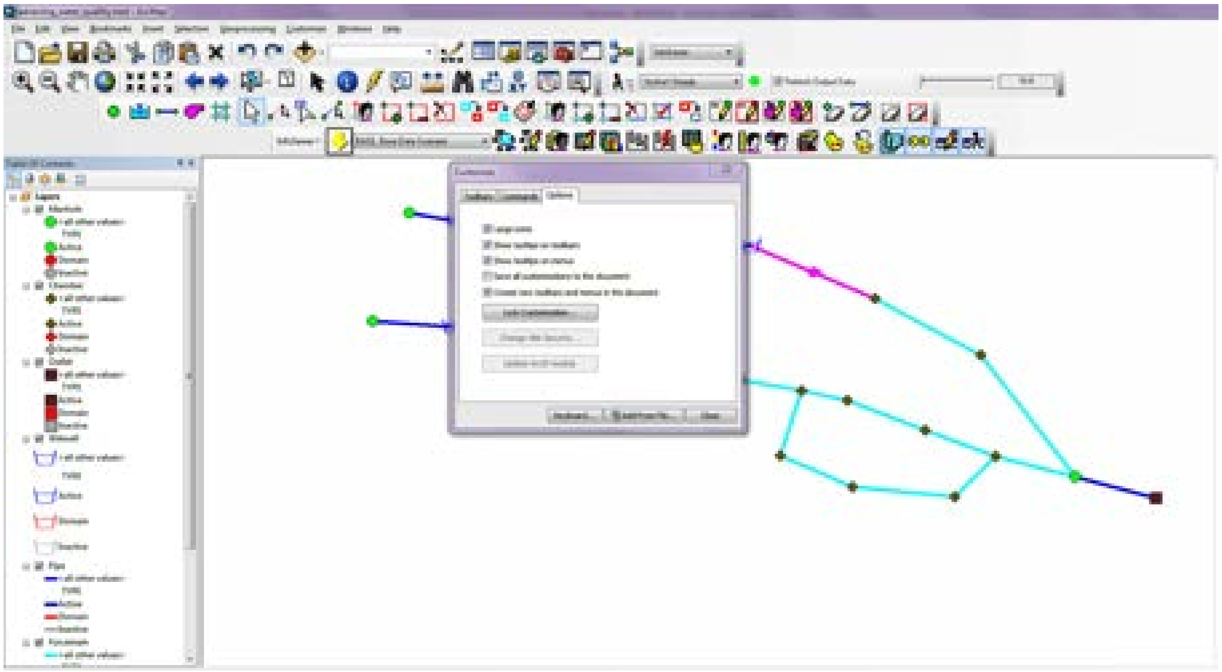
Steps:

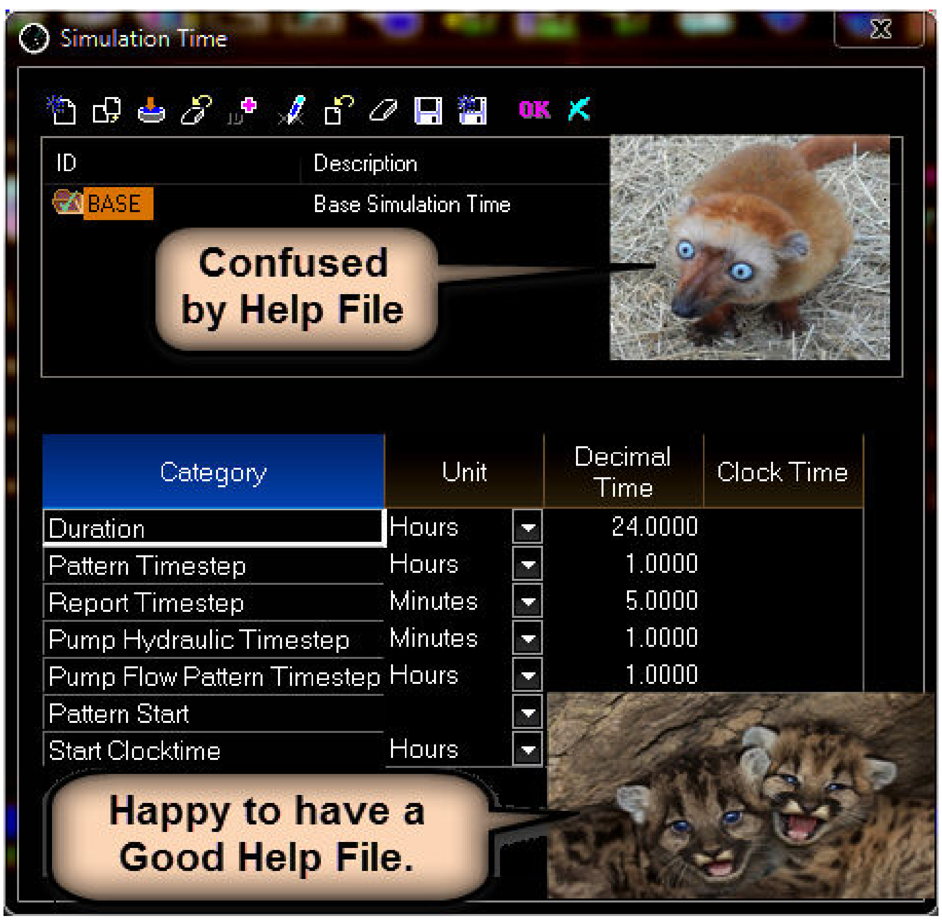
- 1. Click the Customize menu and click Customize Mode.

- 2. Click the Options tab.

- 3. Check Large icons to display large icons for a toolbar's buttons.

- 4. Click Close.




Simulation Time
OK
ID
Description
BASE
Base Simulation Time
Confused by Help File
Category
Unit
Decimal Time
Clock Time
Duration
Hours
24.0000
Pattern Timestep
Hours
1.0000
Report Timestep
Minutes
5.0000
Pump Hydraulic Timestep
Minutes
1.0000
Pump Flow Pattern Timestep
Hours
1.0000
Pattern Start
Start Clocktime
Hours
Happy to have a Good Help File.

Help File Suggestions are always Welcome! - send your idea to support@innovyze.com Clarity is important to us and we would like to hear from you if the help file can be improved or made clearer.

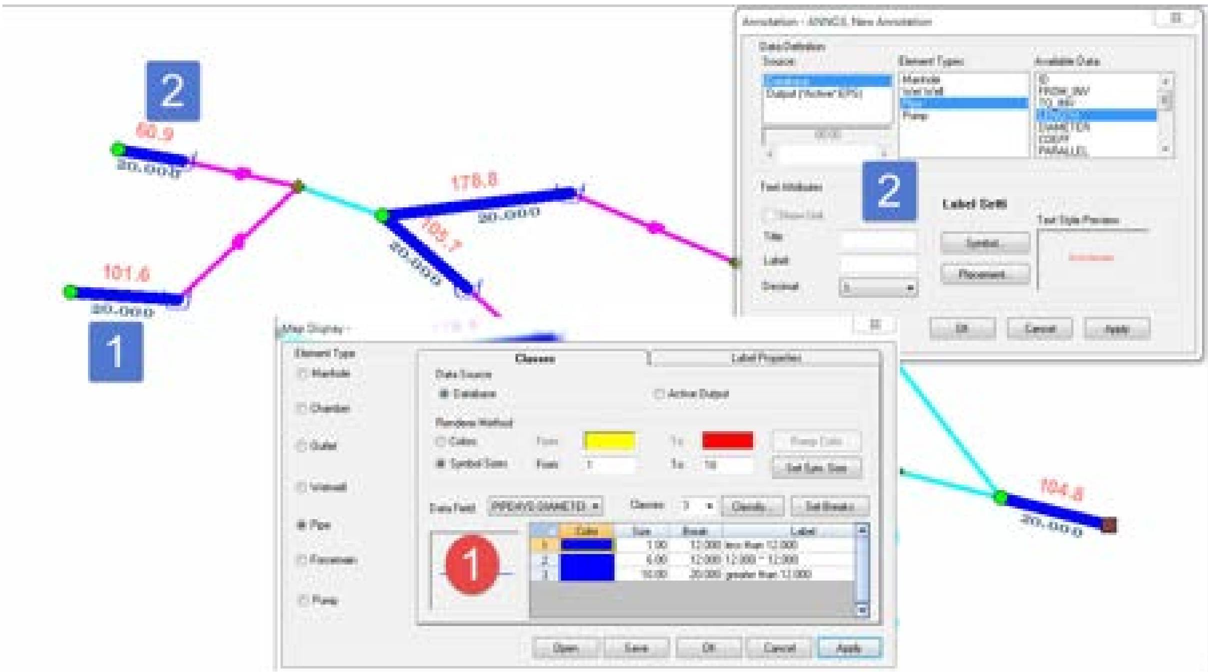
InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

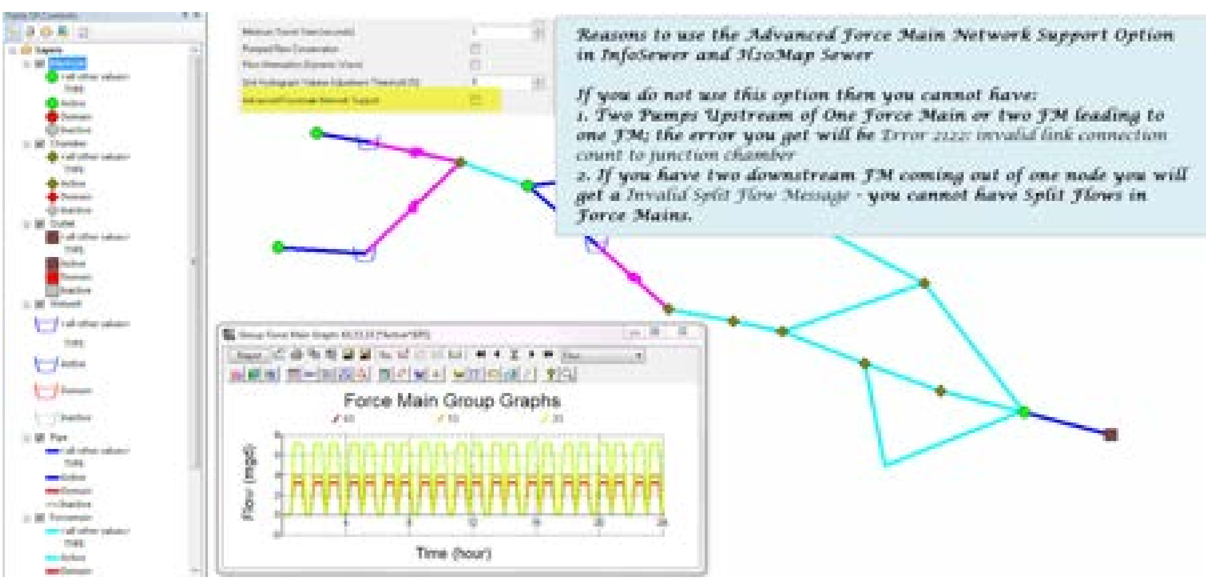
Innovyze Help File Updated March 31, 2021

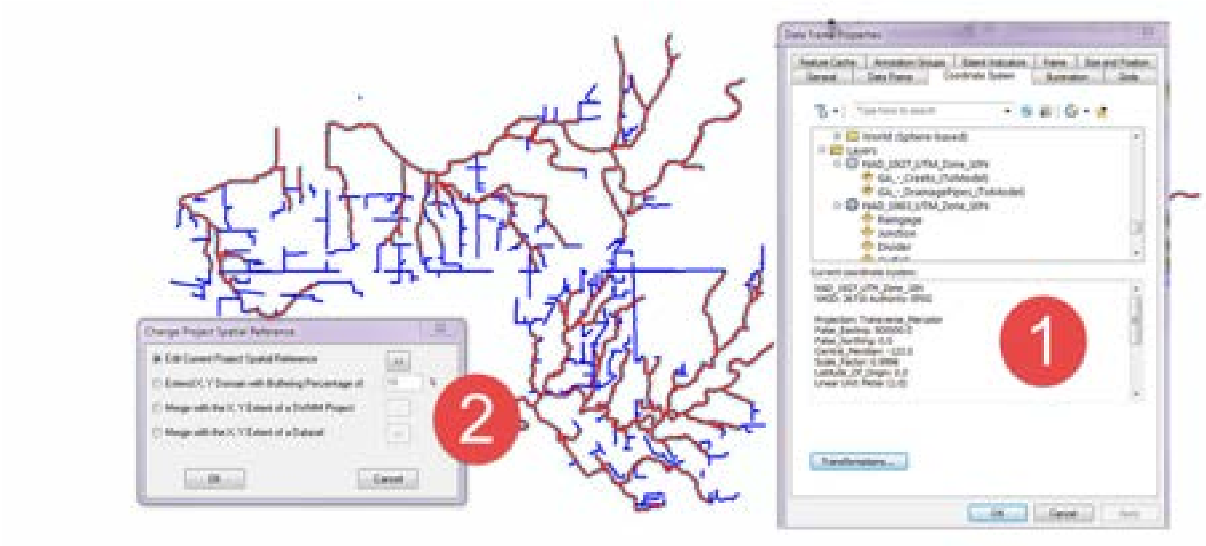
More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

#INFOSEWER

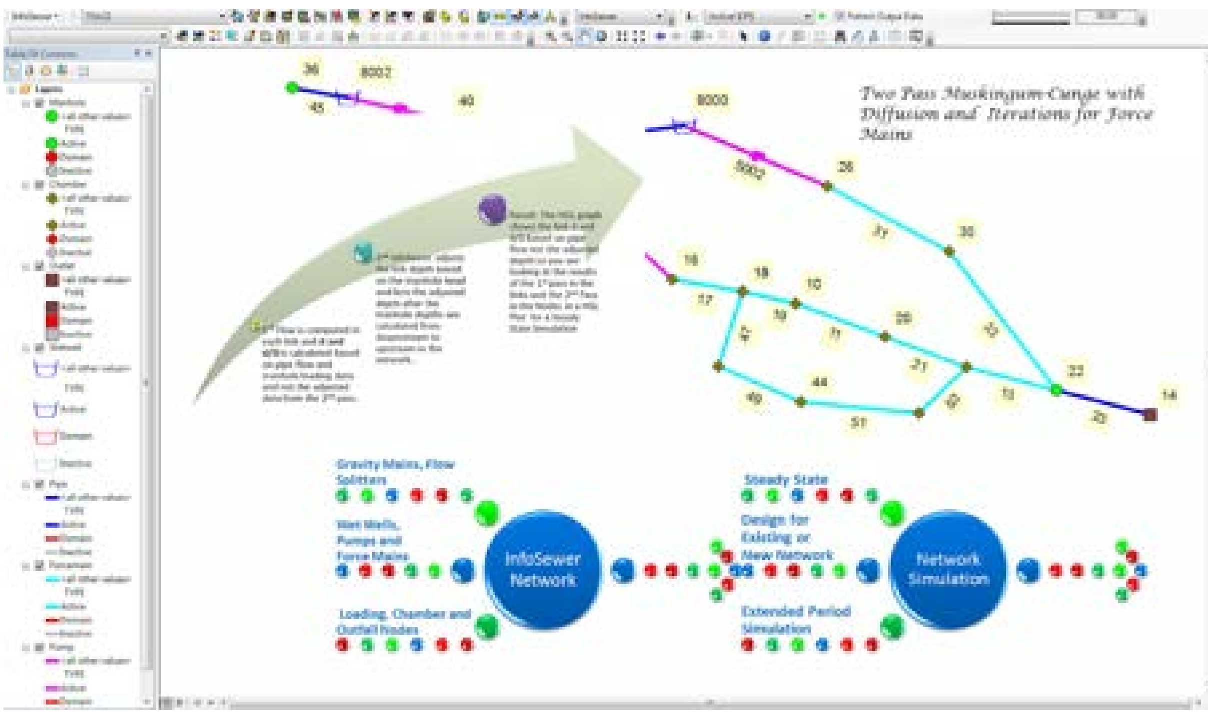
#INFOSEWER images from Twitter - in random order but if you glance at the whole list it might help clarify other sections of this Help File.





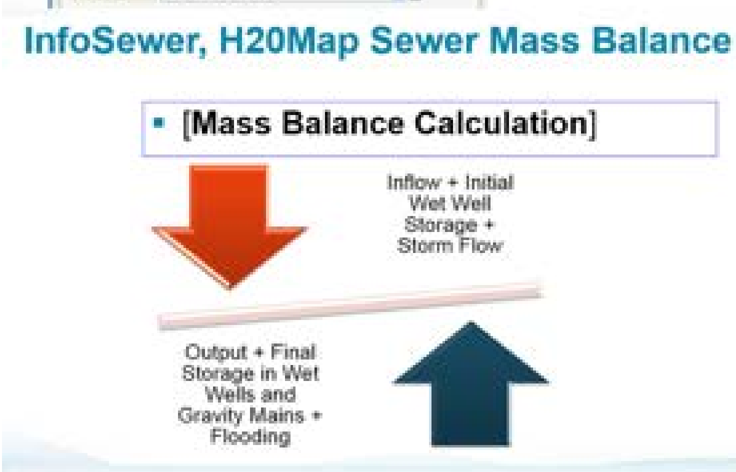


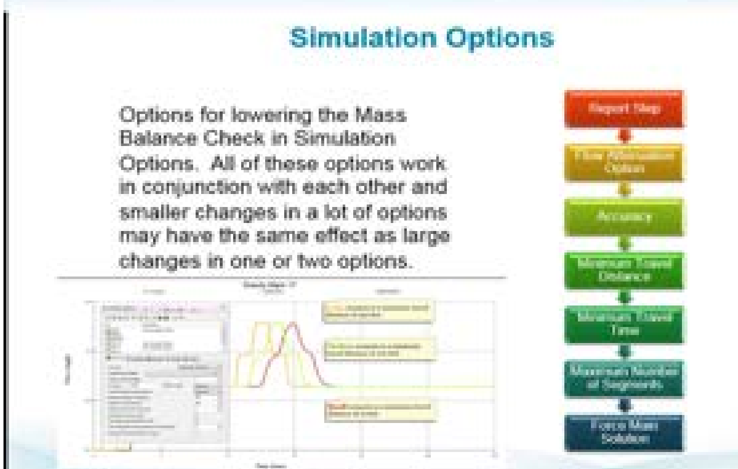
1
2

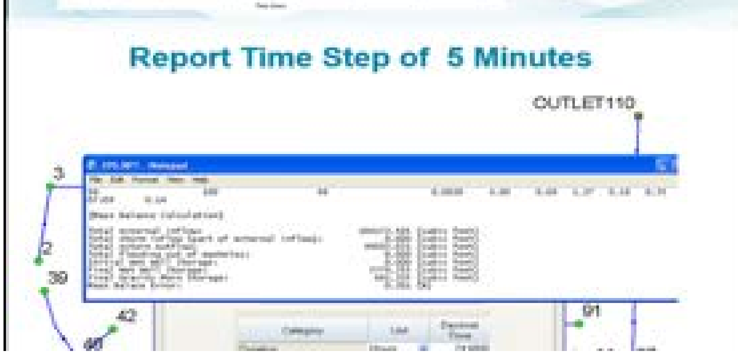


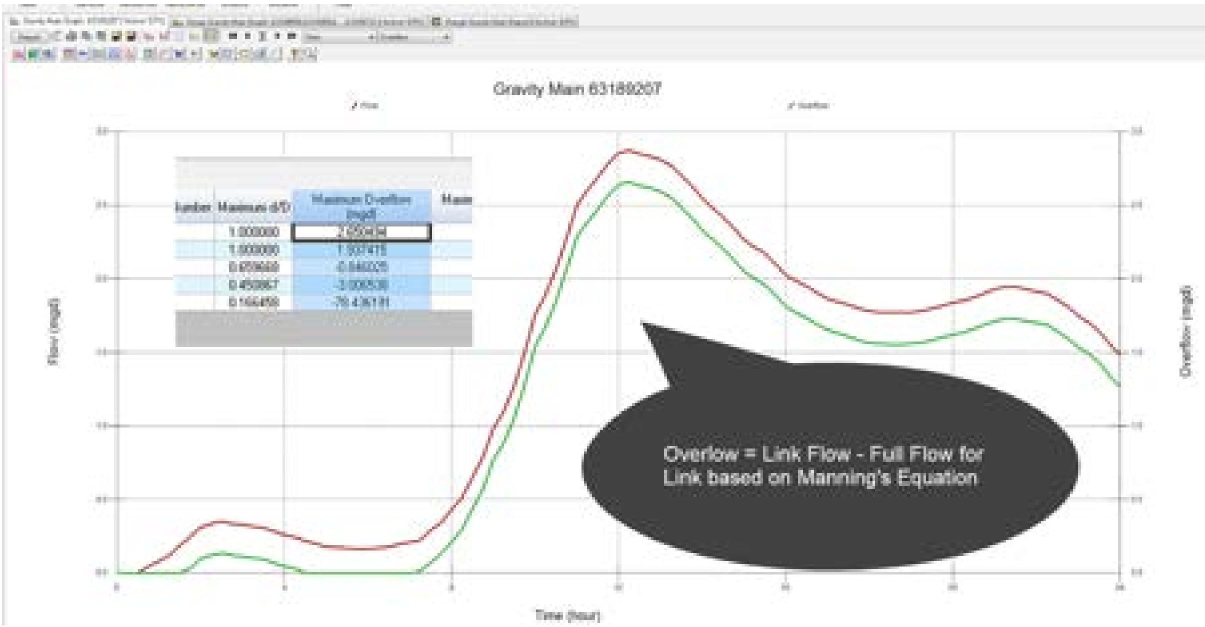
Two Pass Muskingum-Cunge with Diffusion and Iterations for Force Mains
Gravity Mains, Flow Splitters
Wet Wells, Pumps and Force Mains
Loading, Chamber and Outfall Nodes
InfoSewer Network
Steady State
Design for Existing or New Network
Extended Period Simulation
Network Simulation

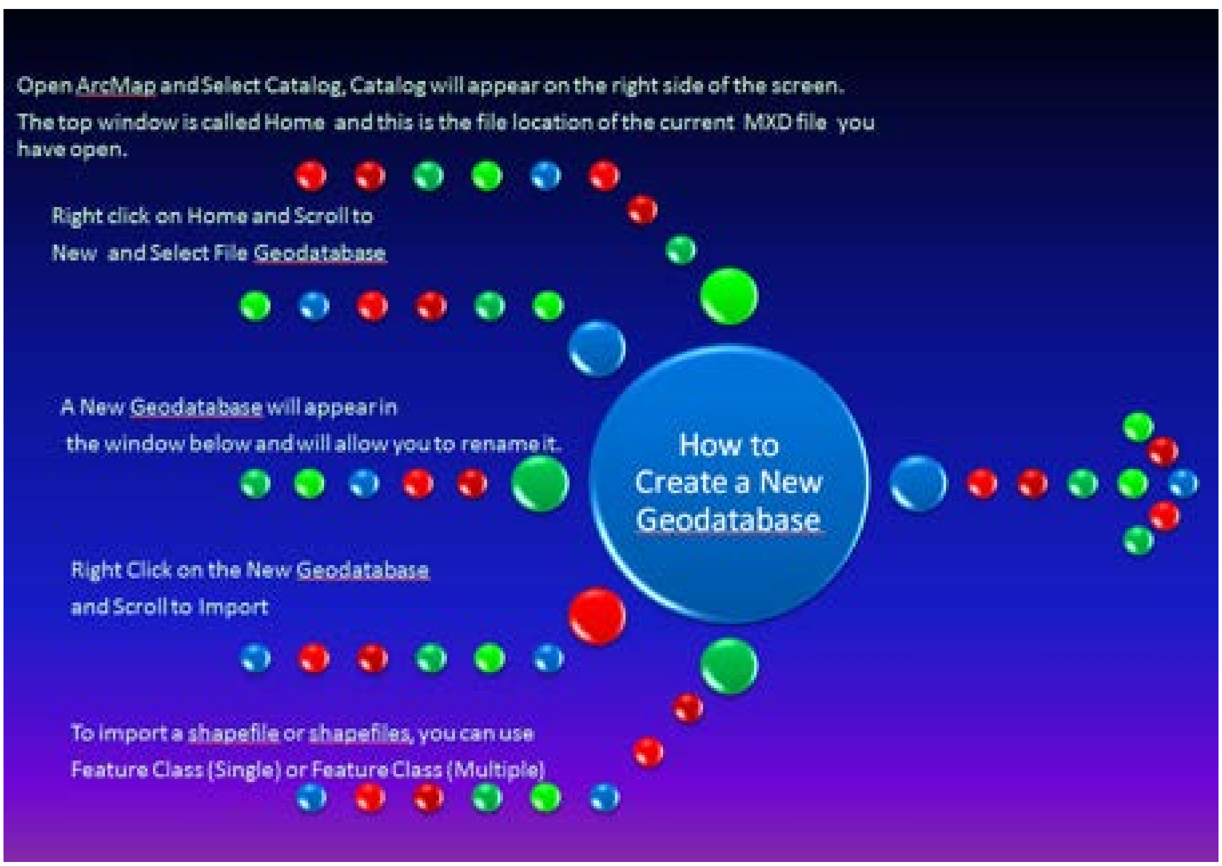

InfoSewer, H20Map Sewer EPS Mass Balance
Steady State
Design
EPS
The Mass Balance Calculations only apply to EPS Simulations – NOT design or Steady State

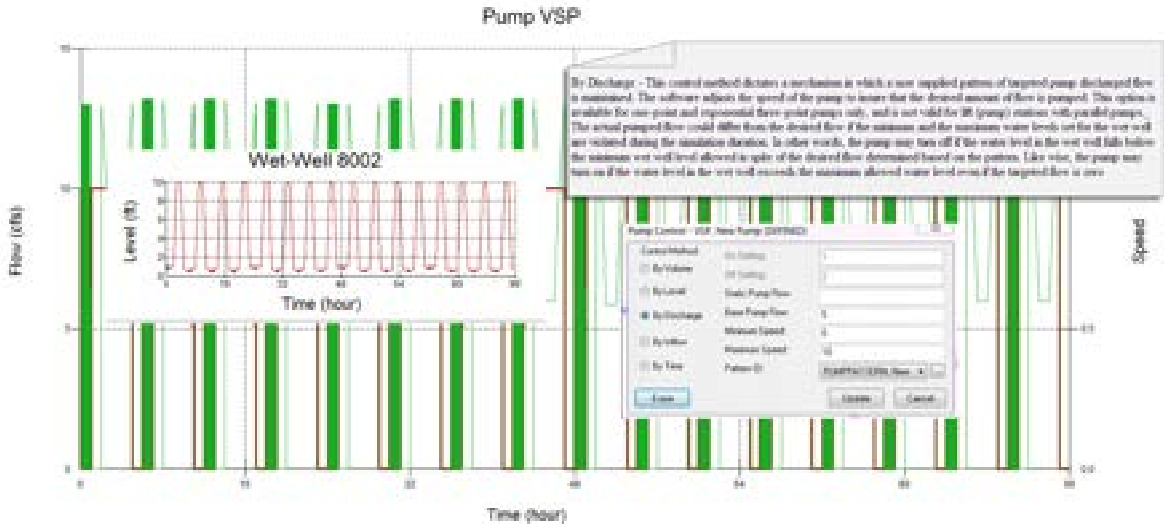

InfoSewer, H20Map Sewer Mass Balance
[Mass Balance Calculation]
Inflow + Initial Wet Well Storage + Storm Flow
Output + Final Storage in Wet Wells and Gravity Mains + Flooding

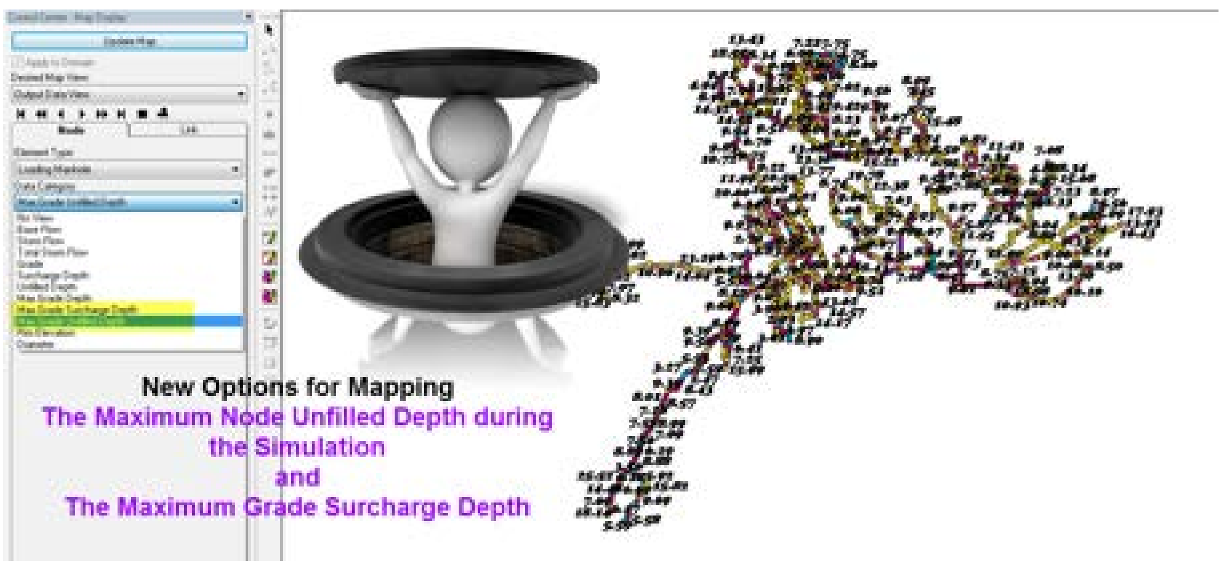

Simulation Options
Options for lowering the Mass Balance Check in Simulation Options. All of these options work in conjunction with each other and smaller changes in a lot of options may have the same effect as large changes in one or two options.
Report Step
Flow Attenuation Option
Accuracy
Minimum Travel Distance
Minimum Travel Time
Maximum Number of Segments
Force Main Solution

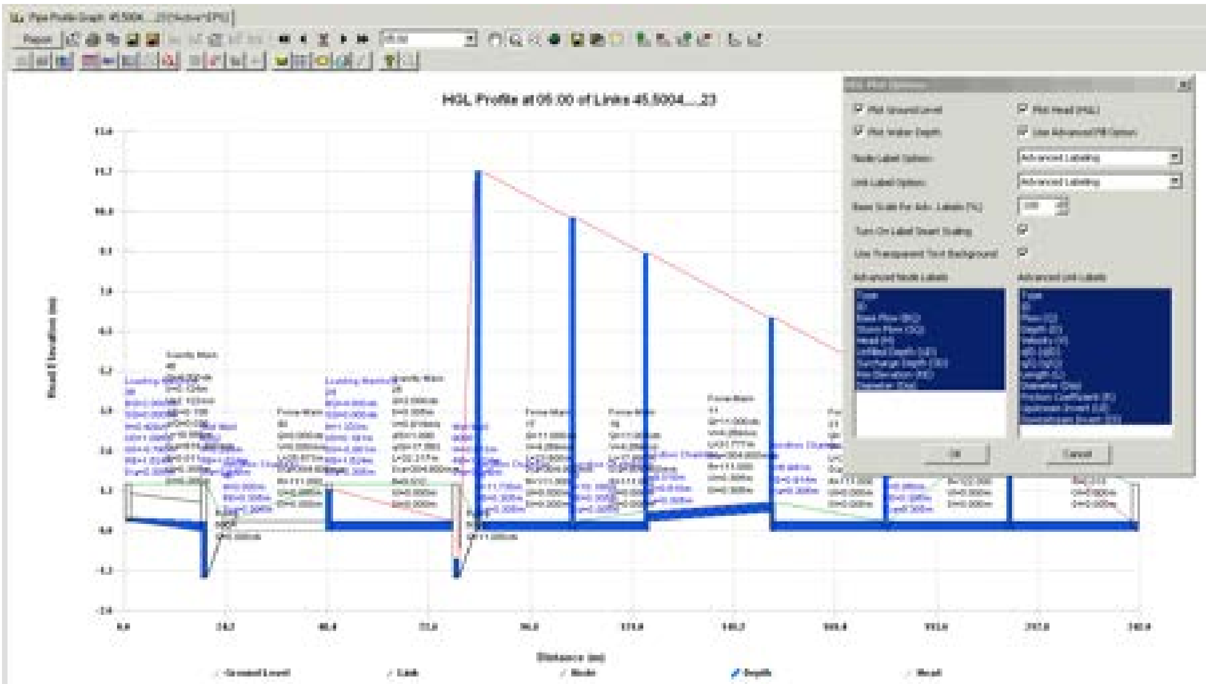

Report Time Step of 5 Minutes
OUTLET110
3
2
39
42
91

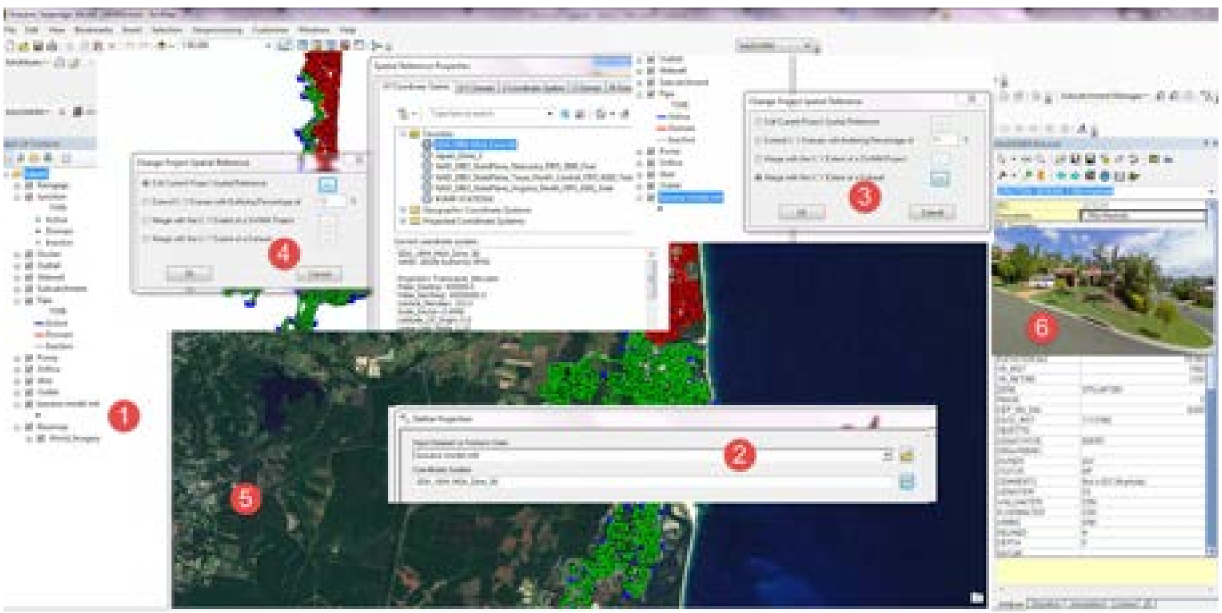

Gravity Main 63189207
Flow (mgd)
Overflow (mgd)
Time (hour)
Overlow = Link Flow - Full Flow for
Link based on Manning's Equation


Open ArcMap and Select Catalog, Catalog will appear on the right side of the screen.
The top window is called Home and this is the file location of the current MXD file you have open.
Right click on Home and Scroll to
New and Select File Geodatabase
A New Geodatabase will appear in
the window below and will allow you to rename it.
How to
Create a New
Geodatabase
Right Click on the New Geodatabase
and Scroll to Import
To import a shapefile or shapefiles, you can use
Feature Class (Single) or Feature Class (Multiple)


Pump VSP
Wet-Well 8002
Level (ft)
Time (hour)
Flow (cfs)
Speed
Time (hour)


Update Map
New Options for Mapping
The Maximum Node Unfilled Depth during the Simulation
and
The Maximum Grade Surcharge Depth





More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer vs InfoSWMM

A short comparison of the features in infoSWMM and infoSewer. , Many of the same rows apply to H2OMap SWMM and H2OMap Sewer.

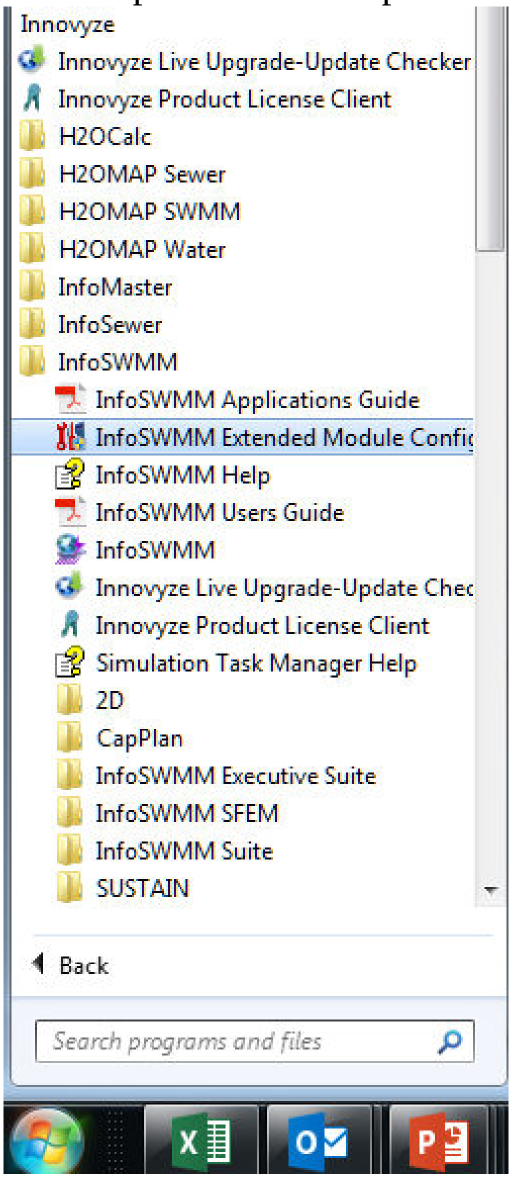
Feature Class	Feature	InfoSewer	InfoSWMM
Base Functionality	Conservative Solution as an option	✓	✓
	Public domain engine	✗	✓
	Calibrate model to flow data	✓	✓
	Calibration Data	✗	✓
	Models siphons	✓	✓
	Infers missing inverts	✓	✓
	Validation tools	✗	✓
	DWF Allocator	✓	✓
	Models flow attenuation	✓	✓

	RDII flow data processing	✔	✔
	Facility Manager	✔	✔
	Batch Simulation	✔	✔
	Scenario Manager	✔	✔
	Scenario Management: Parent-Child Inheritance Functionality	✔	✔
	Advanced Reporting (reserve capacity, freeboard, etc)	✔	✔
	Data Flagging	✘	✘
	Unlimited undo and redo	✘	✘
	Version Control: View Commit History, Compare Differences, and Automate Conflict Resolution	✘	✘

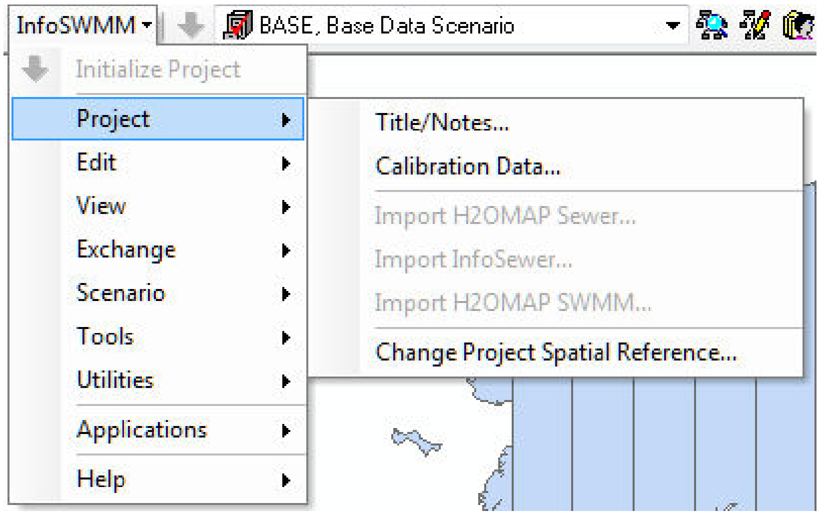
IT/Database Architecture	Multi-User Architecture: (Runs Standalone, in a Workgroup, or as an Enterprise System)	✗	✗
	64 Bit Application & Multi Core Processing	✗	✓
	Remote Simulation Capability: (Allows user to continue working on model while sim is occuring)	✗	✓
	Flexible Licensing: Mix-n-match Model Build and Sim Engine seats as required	✗	✓
2D/Advanced Hydraulic Modeling	River, Bridge, and Ancillary Modeling	✗	✗
	2D Included	✗	**Add On**

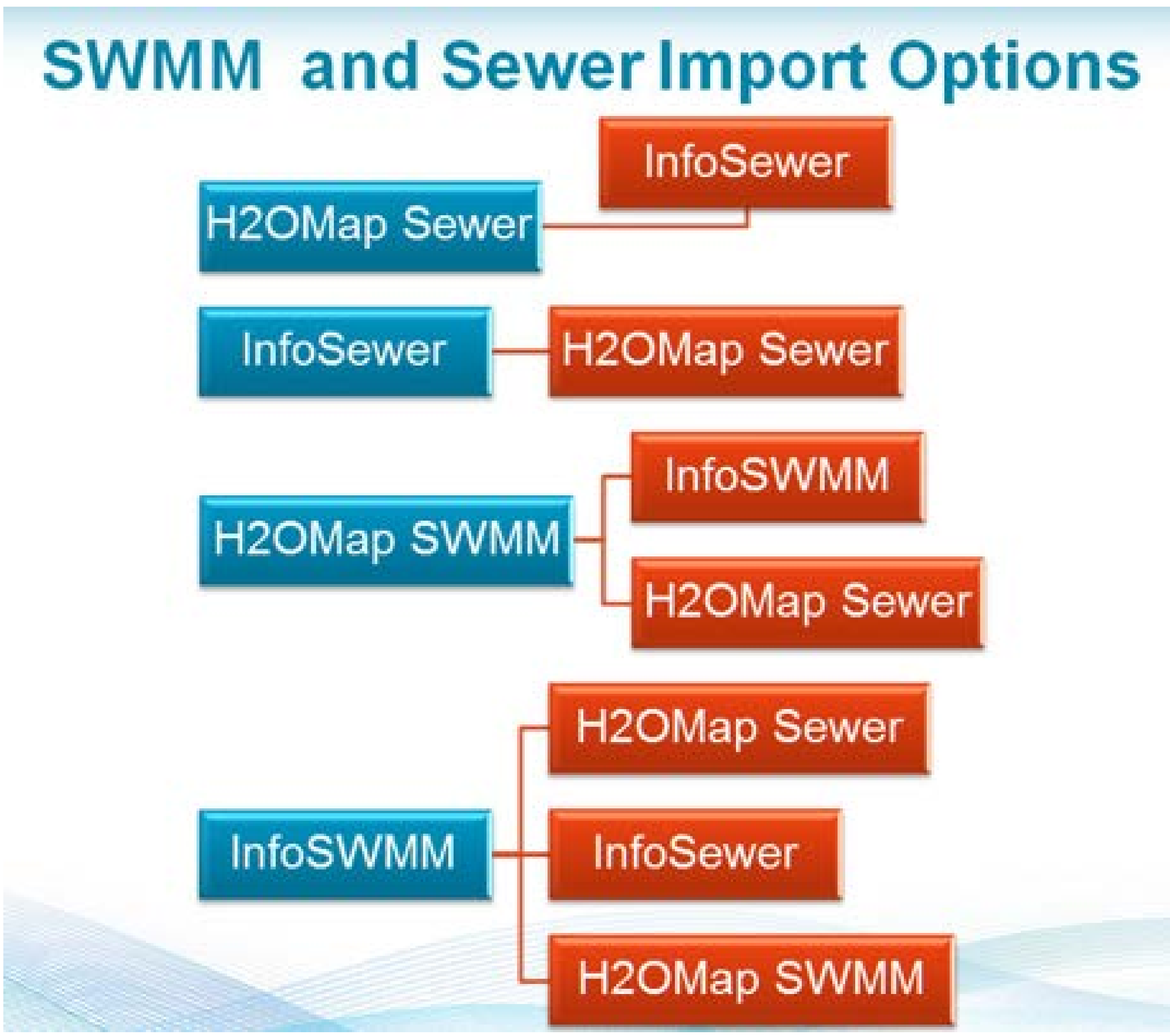
	True Representation of Open Channels: 1) No limit to x-sections. 2) Full hydraulic interaction with 2D zones along length of channel and over both river banks.		**HEC-RAS**
	Superior interaction between rivers, collection systems, and 2D meshes		
	GPU Card for Increased Sim time and Stability		
	Shock Capturing (models high velocity flows/steep embankments)		

H2OMap SWMM imports H2OMap Sewer and infoSWMM directly

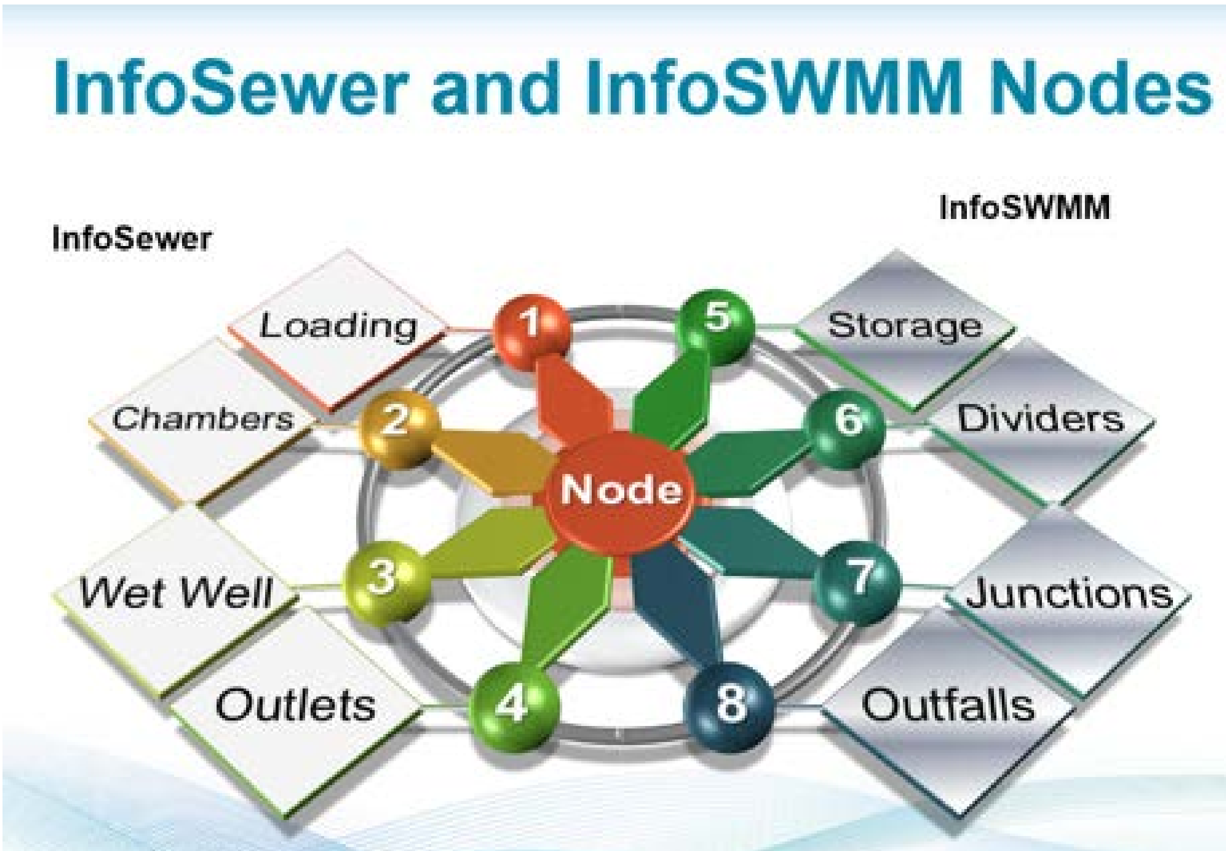


InfoSWMM imports H2OMap SWMM, H2OMap Sewer and InfoSewer directly as shown in the SWMM and Sewer Import Options figure.


InfoSWMM
BASE, Base Data Scenario
Initialize Project
Project
Edit
View
Exchange
Scenario
Tools
Utilities
Applications
Help
Title/Notes...
Calibration Data...
Import H2OMAP Sewer...
Import InfoSewer...
Import H2OMAP SWMM...
Change Project Spatial Reference...


SWMM and Sewer Import Options
H2OMap Sewer
InfoSewer
InfoSewer
H2OMap Sewer
H2OMap SWMM
InfoSWMM
H2OMap Sewer
InfoSWMM
H2OMap Sewer
InfoSewer
H2OMap SWMM

A visual comparison between Sewer and SWMM Nodes



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Innovyze St Venant Solutions for InfoSewer, InfoSWMM, H2OMap SWMM and InfoWorks ICM and InfoWorks ICM SE

This help file topic contrast the St Venant Solution for InfoSewer , InfoSewer/H_2OMap Sewer and ICM/ICM SE.

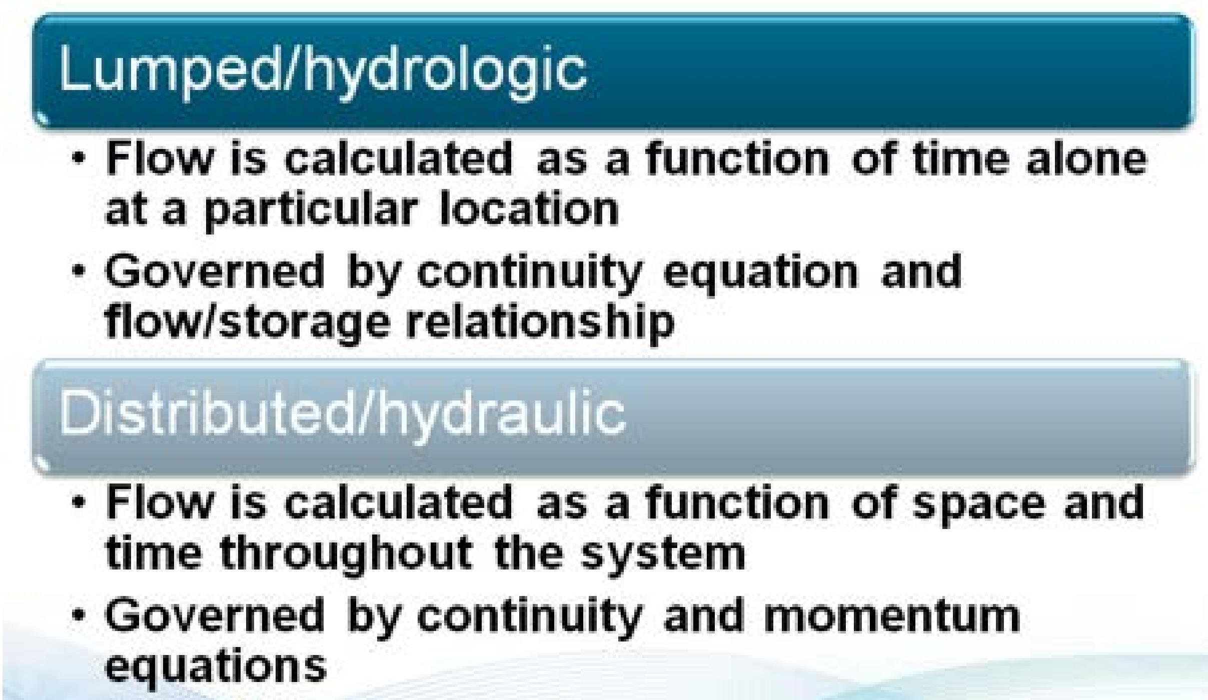
1. Assumptions for the St. Venant Equations

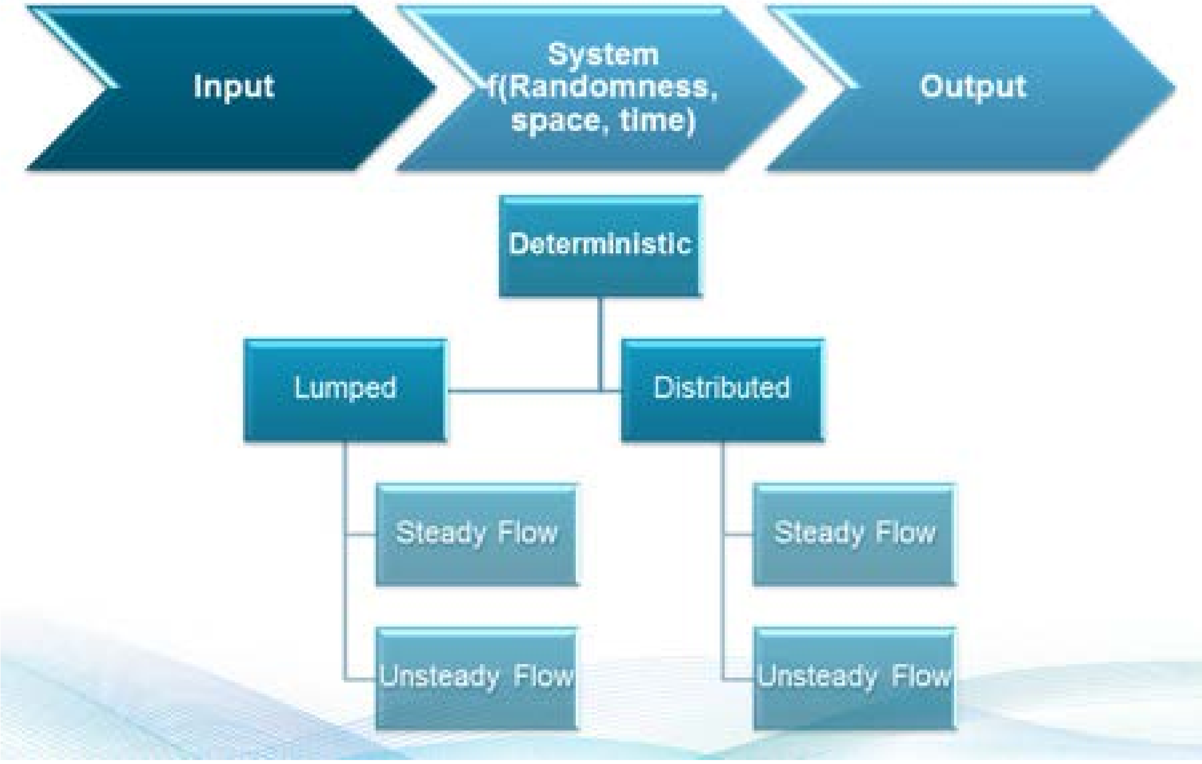
The assumptions behind Lumped and Distributed Models along with the assumptions of the St Venant Equations. InfoSewer, InfoSWMM, H2OMap SWMM, SWMM5, ICM and ICM SE are all Distributed models for Unsteady flow. InfoSWMM and InfoSewer have options for direct, steady flow. ICM and InfoSWMM can also use a quasi-steady flow solution. All of these Innovyze models use the Continuity Equation and Momentum equation for routing flows in links. The numerical solution differs between the three Innovyze main platforms:

InfoSewer and H2OMap Sewer

InfoSWMM, H2OMap SWMM and SWMM 5

ICM and ICM SE





Flow is one-dimensional

Hydrostatic pressure prevails and vertical accelerations are negligible

Streamline curvature is small.

Bottom slope of the channel is small.

Manning's equation is used to describe resistance effects

The fluid is incompressible

Continuity Equation

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0 \qquad\qquad \textit{Continuity}$$

Various Forms of the Momentum Equation

$$\frac{1}{A}\frac{\partial Q}{\partial t} + \frac{1}{A}\frac{\partial}{\partial x}\left(\frac{Q^2}{A}\right) + g\frac{\partial y}{\partial x} - g(S_o - S_f) = 0$$

Local acceleration term	Convective acceleration term	Pressure force term	Gravity force term	Friction force term

$$\frac{\partial V}{\partial t} + V\frac{\partial V}{\partial x} + g\frac{\partial y}{\partial x} - g(S_o - S_f) = 0$$

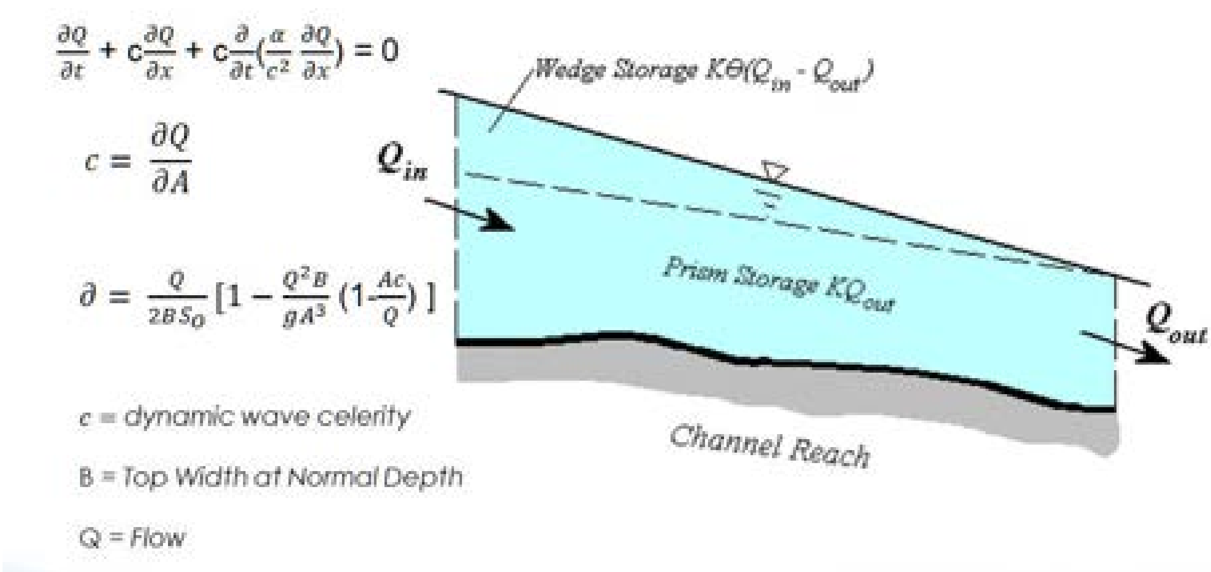
Kinematic Wave

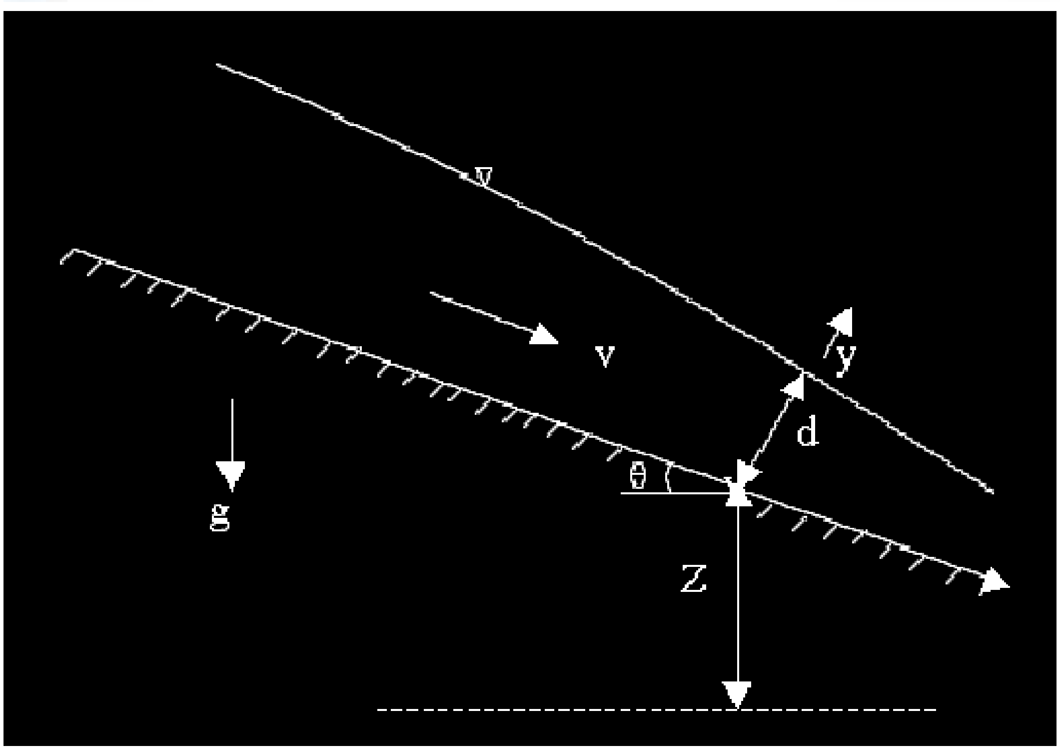
Diffusion Wave

Dynamic Wave

2. Muskingum-Cunge for InfoSewer and H2OMap Sewer

r





The continuity (mass conservation) equation is:

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0$$

$$\frac{1}{gA} \cdot \frac{\partial Q}{\partial t} + \frac{1}{gA} \cdot \frac{\partial}{\partial x}(\frac{\beta Q^2}{A}) + \cos\emptyset \frac{\partial d}{\partial x} - S_o + S_f = 0$$

where

x = distance along the pipe (longitudinal direction of sewer)

A = flow cross sectional area normal to x

y = coordinate direction normal to x on a vertical plane

d = depth of flow of the cross section, measured along y direction

Q = discharge through A

V = cross sectional average velocity along x direction

S0 = pipe slope, equal to sin θ

θ = angle between sewer bottom and horizontal plane

Sf = friction slope

g = gravitational acceleration

t = time

β = Boussinesq momentum flux correction coefficient for velocity distribution

These complete unsteady flow equations (momentum together with continuity) along with appropriate initial and boundary conditions are rather tedious and computationally expensive to solve, especially for large sewer collections systems. As a result, acceptable simplifications and improved solution methods have been proposed including non-inertial, kinematic wave and dynamic wave simplifications. Hydraulically, the dynamic wave approach is the most accurate model among the approximations. The Muskingum-Cunge explicit diffusion wave dynamic flow routing model, obtained by neglecting local acceleration term in the momentum equation, is the most commonly used dynamic wave model.

In InfoSewer /H2OMap Sewer Pro unsteady open channel (free surface) flow is simulated using Muskingum-Cunge technique whereas pressurized flow in any pipe is modeled assuming the pipe is flowing full and the energy equation is applied to the entire pipe section.

Muskingum-Cunge :

$$\frac{\partial Q}{\partial t} + c \cdot \frac{\partial Q}{\partial x} + c\,\frac{\partial}{\partial t}\left(\frac{\alpha}{c^2}\frac{\partial Q}{\partial x}\right) = 0$$

where

$$c = \frac{\partial Q}{\partial A}$$

$$\alpha = \frac{Q}{2BS_0}[1- \frac{BQ^2}{gA^3}[(1- \frac{Ac}{Q})]$$

Here c is the dynamic wave celerity and B is the top width at normal depth for discharge Q. This highly efficient and accurate flow routing algorithm is used by InfoSewer /H2OMap Sewer Pro to track the spatial and temporal variation of flows throughout the collection system.

In this method (a.k.a., one sweep explicit solution method), the network flow dynamic equations are formulated by using an explicit finite difference scheme such that the flow depth, discharge, or velocity at a given location and the current time can be solved explicitly from the known information at the previous locations at the same time level, as well as known information at the previous time level. Thus, the solution is obtained segment by segment, pipe by pipe, over a given time interval for the entire sewer network before progressing to the next interval for another sweep of individual solutions of the network flow equations for the entire network. A variable time step approach (based on the Courant number $c \cdot \frac{\Delta t}{\Delta x}$ is used to minimize numerical dispersion and ensure robustness and stability of the numerical scheme. Complex flow attenuation calculations can be explicitly carried out to more accurately simulate the movement and transformation of sanitary sewer flows in the collection system.

An excellent review and comparison between simulated and observed hydrographs of the various numerical methods for solving unsteady flow in simple and compound channels was presented by Chatila (Chatila 2003). In terms of overall performance, the Muskingum unsteady solution scheme compared favorably and proved to be a simple and reliable method avoiding complicated mathematical and numerical computations for the cases considered.

Flooding at manholes and wet-wells in InfoSewer /H2OMap Sewer Pro is not modeled during an extended period dynamic simulation. Instead, the flows at the flooded structures are conserved and are not lost by the occurrence of flooding at the manholes. In actual flooding situations, flows may be diverted away from the flooded structures and out of the sewer collection system. However, a surcharged pipe or manhole is generally an indication of poor hydraulic performance of the sewer system. InfoSewer/Pro assumes that the downstream pipes of flooded manholes are flowing full.

Sanitary sewer systems are typically designed to flow less than full and have an upper pressure limit of 4 to 6 psi. Sewer systems operating under pressurized flow condition may run the risk of violating local, state, and federal health codes. The USEPA regulations would also be in violation if raw sewage were discharged into the ground, potentially affecting groundwater. For these reasons, pressurized flows in sanitary sewers not designed to sustain pressures can be dangerous and in some cases can present an unlawful activity.

Surcharge

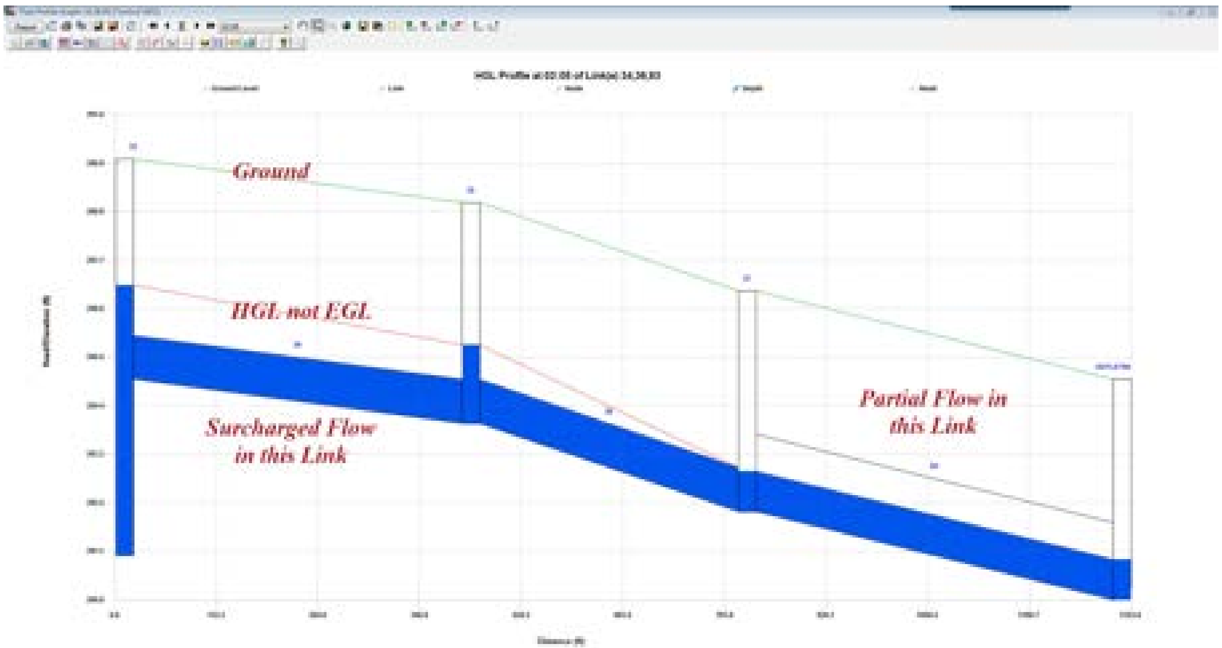
Sewer pipes can flow full with water under pressure, which is often known as surcharge flow. Surcharge flow occurs in under-designed pipes (or under extreme flows) when the flow rate Q exceeds the full pipe capacity Qf .

Flow conditions are unstable at the transition between open-channel (free surface) flow and full pipe flow. A wave or surge can induce full flow in the pipe in the unstable range. Surcharge in sewer pipes is modeled inInfoSewer /H2OMap Sewer Pro using energy and continuity principles. The energy equation between sections 1 and 2 in a pipe can be written as:

$$z_1 + d_1 + \frac{V_1^2}{2g} - H_L - z_2 - d_2 - \frac{V_2^2}{2g} = 0$$

Here z denotes the invert elevation; d represents the water depth; and H_L designates the head loss between sections 1 and 2. The energy equation is used to determine the difference in hydraulic grade line elevation (which is added at the upstream manholes) needed to pass downstream flows under the surcharge condition.

The procedure for analyzing surcharge in sewer pipes is illustrated using the figure below as a reference.



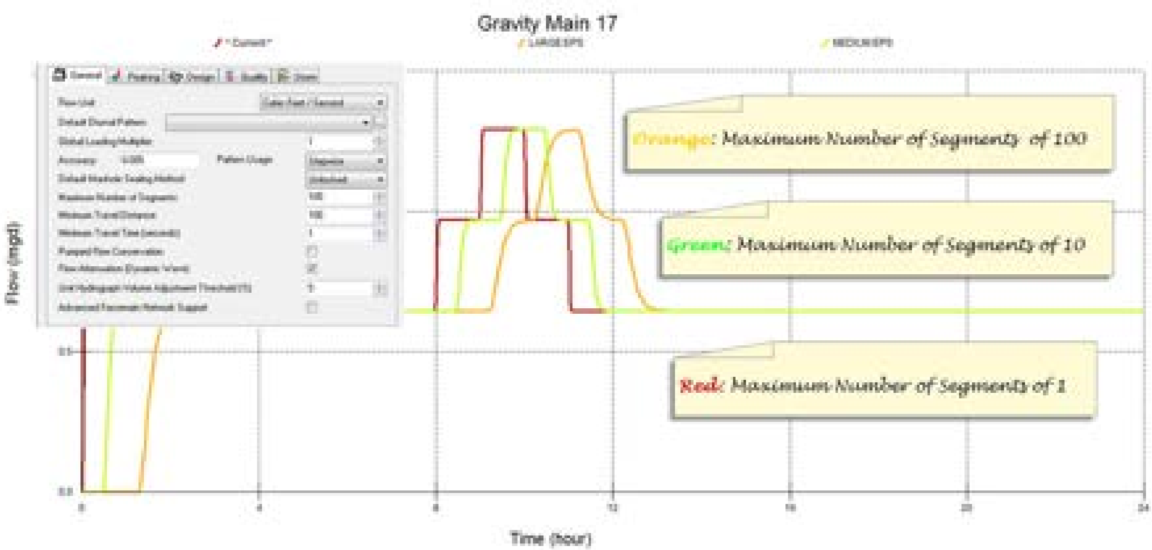
Assuming that pipe 4 (between manholes 4 and 5) is under-designed, Q4 will exceed its full flow capacity and the hydraulic grade line at manhole 4 will increase based on energy consideration to allow Q4 to pass through pipe 4 (note that water always flows from higher to lower energy) as continuity must be satisfied. This forces the hydraulic grade line at manhole 3 to increase in order for Q3 to pass through pipe 3. The procedure continues upstream until the slope of the energy grade line needed to transport the flow allows open-channel flow condition to occur in the pipe. The projected hydraulic grade line will then intersect the uniform water surface flow to complete the backwater curve.

The energy equation is also used to model the flow in siphons, which can occur in adverse pipes. InfoSewer assumes that the siphon flows full, with a continuous liquid column throughout it.

Flow Attenuation

When a flow hydrograph is injected and propagates downstream in sewer pipes the bulk of the water will normally travel slower than its induced disturbance or wave. That is, if the water is injected with a tracer then the tracer lags behind the disturbance. The speed of the disturbance depends on parameters such as depth, width and flow velocity. This disturbance will tend to flatten, or spread out, the peak flow in the downstream direction along the sewer pipes.

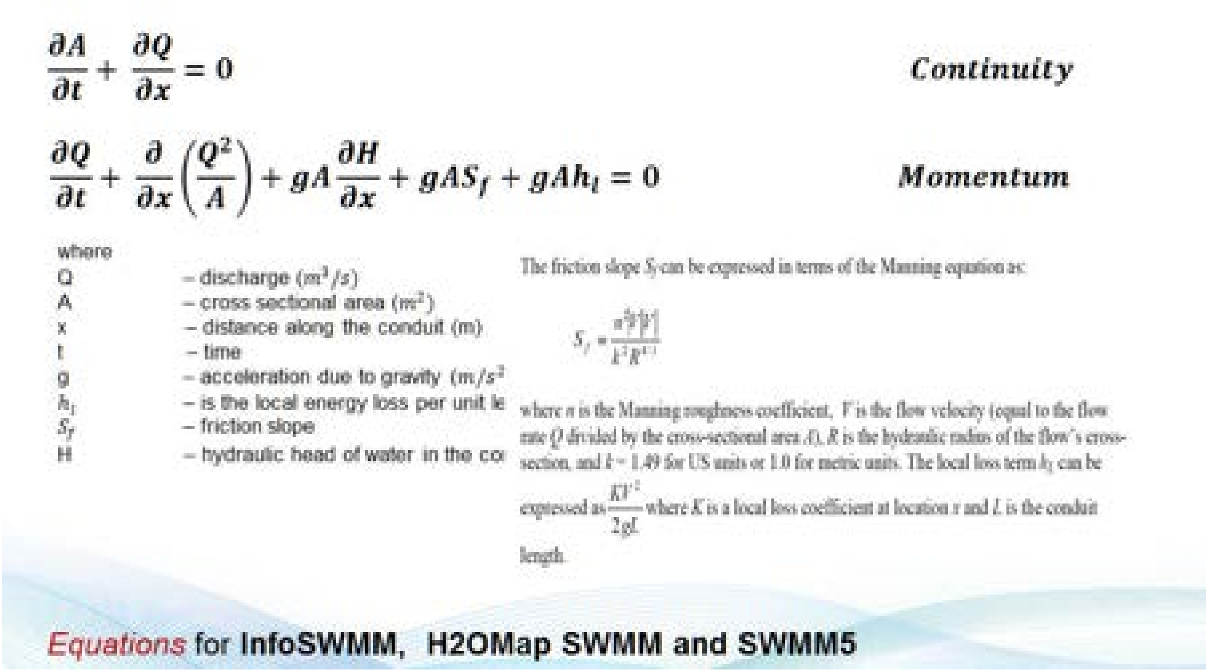
Flow attenuation in a sewer system is defined as the process of reducing the peak flow rate by redistributing the same volume of flow over a longer period of time as a result of friction (resistance), internal storage and diffusion along the sewer pipes. InfoSewer /H2OMap Sewer Pro uses the distributed Muskingum-Cunge flow routing method based on diffusion analogy, which is capable of accurately predicting hydrograph attenuation or peak flow damping effects (peak subsidence). The method is attractive since the routing parameters can be directly calculated as a function of pipe and flow properties, is applicable for a wide range of flow conditions, and does not require calibration or any iterative scheme. The Muskingum coefficients are derived from the pipe diameter, length, discharge, dynamic wave celerity, and slope of the flow. The magnitude of attenuation depends on parameters such as the peak discharge, the curvature of the hydrograph, and the width of flow. An example of flow attenuation process as a hydrograph is routed through a sewer system is illustrated in the figure below.

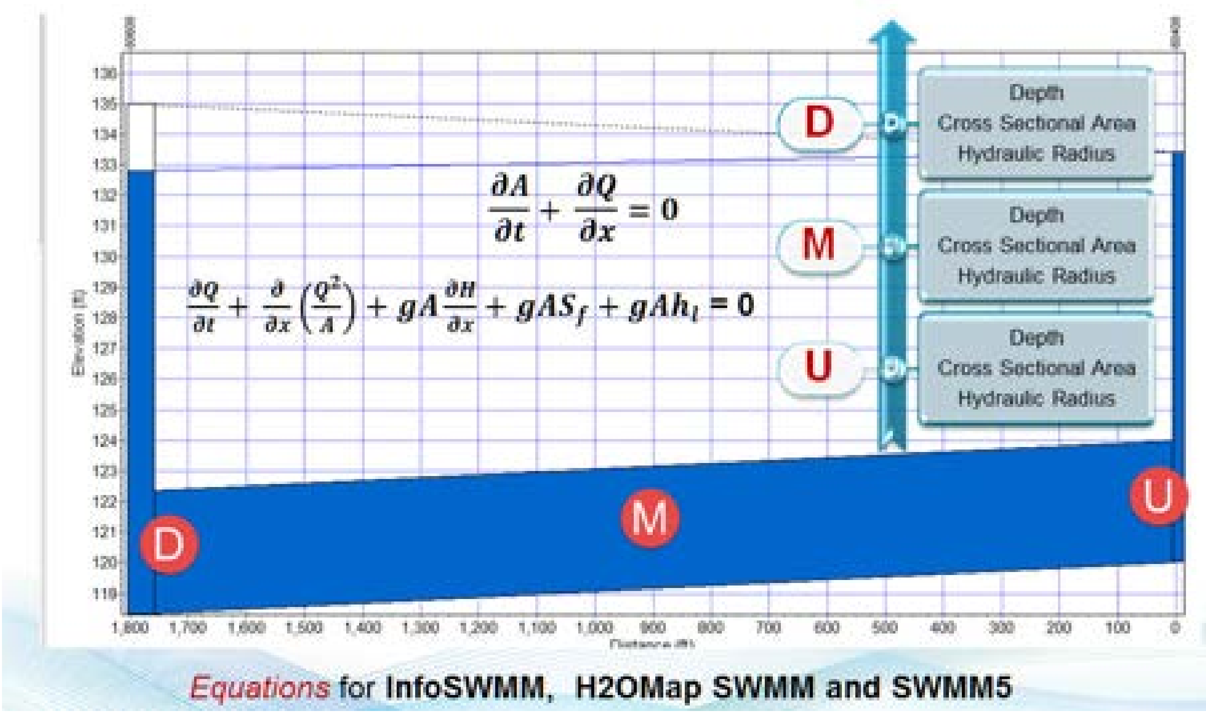


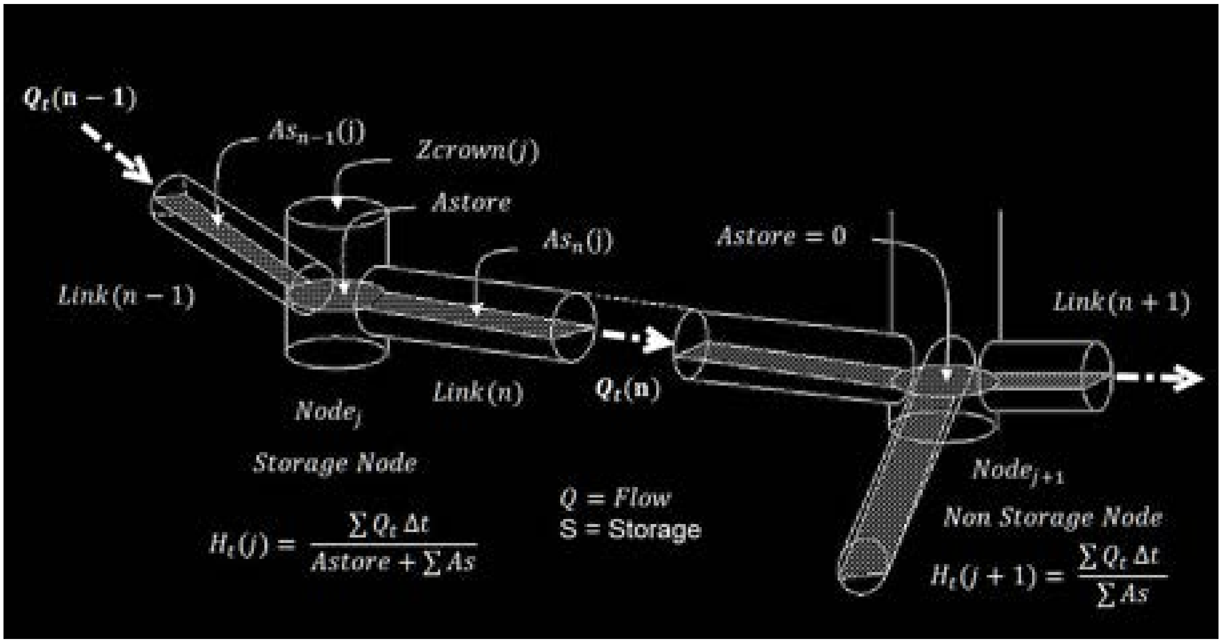
[Hydrograph Aggregation/Flow Accumulation]

Proper aggregation of multiple hydrographs with distinct time steps is essential in a sewer collection system as the flows are routed in both time and space. Aggregation normally occurs when laterals are merging around manholes and wet-wells. This can create offset of time-steps, which can affect accurate determination of flow peaks and volumes. InfoSewer /H2OMap Sewer Pro utilizes a highly accurate dynamic hydrograph aggregation method that allows preservation of both flow peaks and flow volumes when multiple hydrographs with different time steps are added. The method is Lagrangian in nature and tracks the hydrograph ordinates as they are transported along the sewer pipes and mix together at manholes and wet-wells. A variable time step is used to minimize numerical dispersion, enhance stability, and maximize computational efficiency. See the User Guide for more information on Extended Period Simulations.

3. H_2OMap SWMM and InfoSWMM

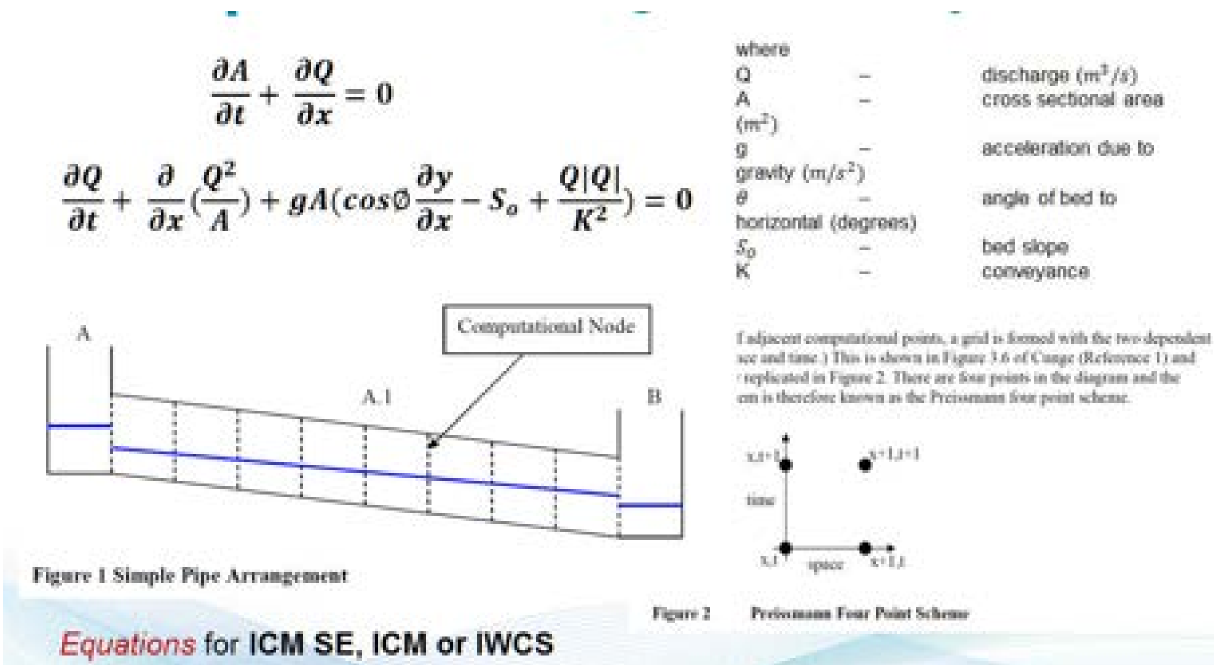






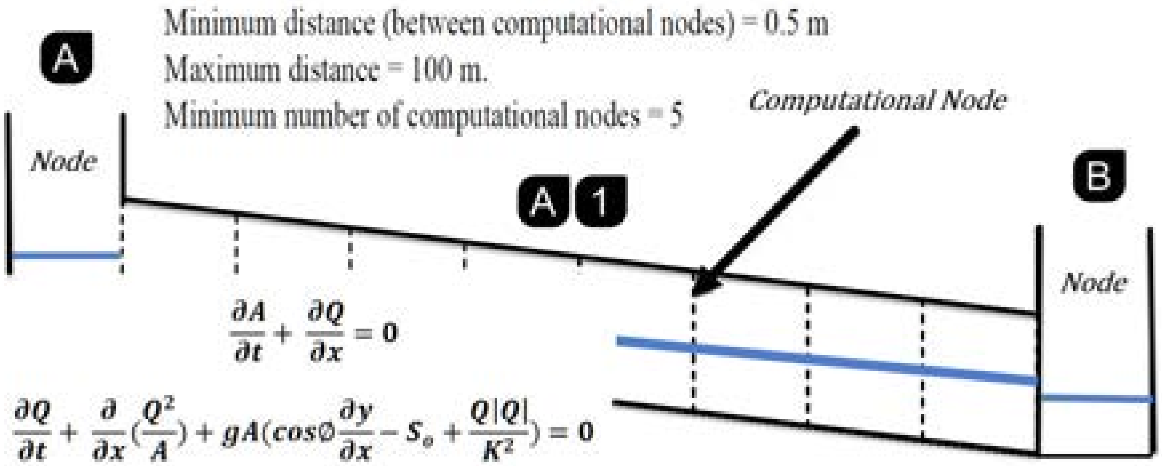
$Q_t(n-1)$
$As_{n-1}(j)$
$Zcrown(j)$
$Astore$
$As_n(j)$
$Astore = 0$
$Link\,(n-1)$
$Link\,(n+1)$
$Link\,(n)$
$Q_t(n)$
$Node_j$
Storage Node
$Node_{j+1}$
$Q = Flow$
S = Storage
Non Storage Node
$H_t(j) = \frac{\sum Q_t \Delta t}{Astore + \sum As}$
$H_t(j+1) = \frac{\sum Q_t \Delta t}{\sum As}$

4. ICM and ICM SE

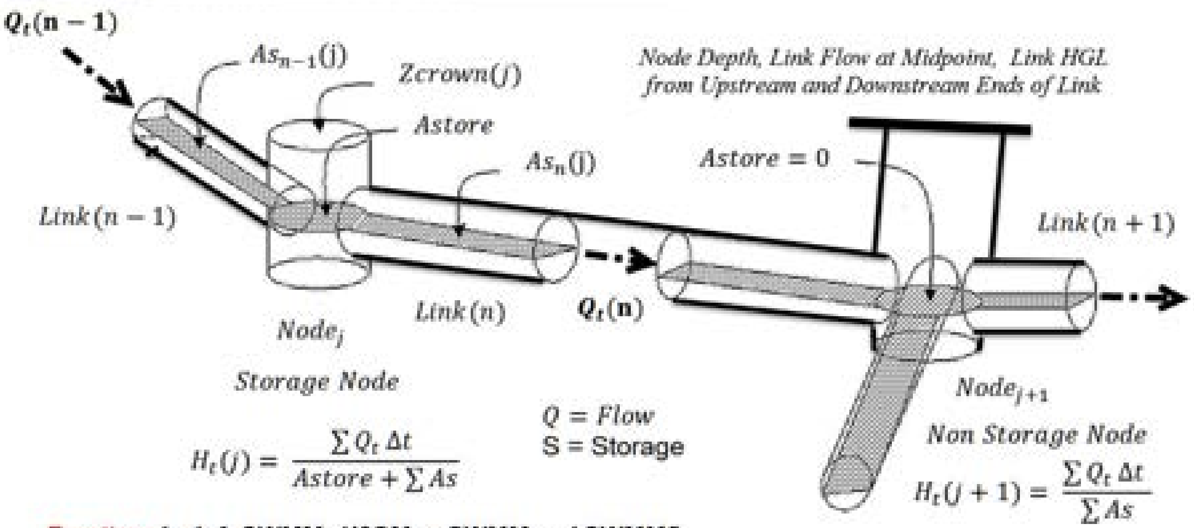


Collection System Modeling	ArcGIS-based Solutions	Stand-Alone Geospatial Solutions	Workgroup Model Management Solutions
Integrated Catchment Modeling Integrated river, sewer and overland flow modeling to accurately represent all flow paths and effectively simulate the water quality impact of polluting runoff and effluent from urban areas.			InfoWorks® ICM
Urban Drainage and Stormwater Modeling Take on the most complex and demanding sewer system with confidence and quickly determine the most cost-effective solution to flooding and pollution management.	InfoSWMM®	H_2OMAP SWMM®	InfoWorks® ICM SE
Urban Stormwater Treatment and Analysis Integrate comprehensive watershed modeling capabilities, best management practice (BMP) process simulation, and BMP cost representation within the context of a cost-benefit optimization framework.	InfoSWMM® Sustain		
Sanitary Sewer Use state-of-the-art tools to cost-effectively plan, design, analyze, rehabilitate and expand your wastewater system in record time.	InfoSewer®	H_2OMAP Sewer®	

5. A common look at the Equations for ICM, ICM SE. InfoSWMM and H2OMap SWMM



Equations for **ICM SE** and **ICM**



Equations for **InfoSWMM, H2OMap SWMM and SWMM5**

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0 \qquad \textit{Continuity}$$

$$\frac{\partial Q}{\partial t} + \frac{\partial}{\partial x}\left(\frac{Q^2}{A}\right) + gA\frac{\partial H}{\partial x} + gAS_f + gAh_l = 0 \qquad \textit{Momentum}$$

6. ICM 2D and InfoSWMM 2D Equations

ICM 2D and InfoSWMM share the same computational engine as described on the Innovyze Blog

$$\frac{\partial h}{\partial t} + \frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y} = q_{1D}$$

$$\frac{\partial (hu)}{\partial t} + \frac{\partial}{\partial x}\left(hu^2 + \frac{gh^2}{2}\right) + \frac{\partial (huv)}{\partial y} = S_{0,x} - S_{f,x} + q_{1D}u_{1d}$$

$$\frac{\partial (hv)}{\partial t} + \frac{\partial}{\partial y}\left(hv^2 + \frac{gh^2}{2}\right) + \frac{\partial (huv)}{\partial x} = S_{0,y} - S_{f,y} + q_{1D}v_{1d}$$

where,
h is the water depth,
u and v are the velocities in the x and y directions, respectively
q_{1D} is the source discharge per unit area and,
u_{1D} and v_{1D} are the velocity components of the source discharge in the x and y directions, respectively

As the scheme is an explicit solution it does not require iteration to achieve stability within defined tolerances like the ICM 1D scheme or the iterative solution in InfoSWMM. Instead, for each element, the required timestep is calculated using the Courant-Friedrichs-Lewy condition in order to achieve stability, where the Courant-Friedrichs-Lewy condition is

$$c\,\frac{\Delta x}{\Delta t} \leq 1$$

where,
c is a dimensionless Courant number
(the time step/Stability Control set in the 2D parameters (default = 0.95)

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Computational error codes

3001 : unable to allocate run time memory

3002 : unable to complete module computation 3111 : unable to complete run-time data setup 3211 : incorrect flow-split division

3212 : insufficient pump head

3213 : Pump flow is unreasonably small

3214 : Discharge and inflow based controls do not apply to parallel pumps

3215 : The maximum pump speed is too small to pump the flow 3900 : run-time error

Connectivity error codes

2111 : missing loading manhole downstream connection 2112 : invalid downstream pump station force-main connectio 2113 : invalid intermediate loading manhole in force-mains 2114 : invalid intermediate loading manhole in pump station force-main 2115 : invalid loading manhole connection to pump station 2121 : junction chamber must be connected to force-main/pump 2122 : invalid link connection count to junction chamber 2123 : junction chamber downstream link is not a force-main 2124 : missing downstream link to junction chamber 2125 : invalid upstream link type

2126 : missing upstream link to junction chambe 2131 : invalid connection to outlet

2132 : invalid downstream connection from outlet 2133 : invalid upstream link type

2141 : unsupported multiple downstream links from wet well 2142 : wet well downstream link is not a pum

2143 : wet well upstream link is not a gravity main 2211 : rim elevation less than downstream invert level 2212 : rim elevation less than downstream pipe-main level 2501 : loop found in given network

2502 : multi-loops found in given network

2511 : disconnected nodes

2512 : disconnected links

2531 : unable to organize pipe link topology

2611 : invalid split-flow configuration for non-pipe link 2612 : invalid adverse pipe from split-flow manhole

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Curve data error codes

1511 : invalid curve size data

1521 : invalid negative level (X-axis) value for wet well curve 1522 : invalid negative volume (Y-axis) value for wet well curve 1523 : wet well curve level (X-axis) values not in ascending order 1524 : wet well curve volume (Y-axis) values not in ascending order 1531 : invalid negative/zero diameter (X-axis) value for criteria curve 1532 : invalid negative d/D (Y-axis) value for criteria curve

1533 : d/D (Y-axis) value > 1.0 for criteria curve

1534 : criteria curve diameter (X-axis) values not in ascending order 1541 : invalid negative/zero diameter (X-axis) value for cost curve 1542 : invalid negative cost (Y-axis) value for cost curve

1543 : cost curve diameter (X-axis) values not in ascending order 1551 : invalid negative inflow (X-axis) value for flow-split curve 1552 : invalid negative flow-split ratio (Y-axis) value for flow-split curve

1553 : flow-split ratio (Y-axis) value > 100% for flow-split curve 1554 : flow-split curve flow (X-axis) values not in ascending order

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Design data error codes

1711 : minimum Design velocity > maximum

1721 : undefined Design Analysis curve

1722 : diameter of pipe noted below is greater than the maximum diameter value defined in the ANALYSIS Curve

1731 : undefined Design Criteria curve

1732 : undefined Design Replacement curve

1733 : undefined Design Duplication curve

1734 : maximum Replacement curve diameter > maximum Design Criteria curve diameter

1735 : maximum Duplication curve diameter > maximum Design Criteria curve diameter

InfoSewer™

Modeling Made Easy

Input Error Codes

1001 : unable to scan given input file 1002 : unable to allocate necessary memory for input data 1003 : data memory application error 1011 : undefined loading type

1012 : undefined wetwell type

1013 : undefined pump type

1014 : undefined pattern type

1015 : undefined curve type

1016 : undefined control type

1021 : undefined manhole

1022 : undefined wet well

1023 : undefined node

1024 : undefined pipe

1025 : undefined pump

1026 : undefined pattern

1027 : undefined curve

1028 : identical end nodes for given link 1031 : duplicate Manhole ID

1032 : duplicate Wet Well ID

1033 : duplicate Pipe ID

1034 : duplicate Pump ID

1035 : duplicate Pattern ID

1036 : duplicate Curve ID

1041 : invalid loading pattern type 1042 : invalid curve application type 1051 : invalid Pattern section order 1052 : invalid Curve section order 1053 : invalid Loading Manhole section order 1054 : invalid Junction Chamber section order 1055 : invalid Outlet Manhole section order 1056 : invalid

Wet Well section order 1057 : invalid Pipe section order 1058 : invalid Force-Main section order 1059 : invalid Pump section order 1071 : unrecognized data field

1081 : missing default constant pattern 1082 : missing default criteria curve 1083 : missing default cost curve 1091 : invalid data fields

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Link data error codes

1311 : given pipe upstream manhole is an outlet 1312 : given gravity main has upstream invert <= downstream invert 1313 : invalid pipe length data

1314 : invalid pipe diameter data

1315 : invalid pipe friction/resistance data 1316 : invalid parallel pipe parallel count data 1331 : invalid Manning coefficient for gravity main 1332 : flow-split pipes do not apply to non-loading manhole nodes 1333 : adverse/flat pipes do not apply to flow-splits 1351 : invalid down-stream outlet node for given force-main 1352 : invalid split-flow configuration for given force-main 1355 : invalid force-main connection at node junction 1361 : invalid force-main group data object 1362 : invalid parallel force-mains

1363 : invalid split force-mains

1364 : invalide intermediate node type for the force-main group 1365 : invalid from node type for the leading force-main 1366 : invalid to node type for the last force-main 1391 : model does not contain any pipe

1411 : invalid parallel pump count data

1412 : invalid parallel pump definition

1421 : invalid pump capacity data

1422 : invalid pump design head data

1423 : invalid pump design flow data

1424 : invalid pump cutoff head data

1425 : invalid pump design head data

1426 : invalid pump design flow data

1427 : invalid pump high head data

1428 : invalid pump high flow data

1431 : downstream manhole for pump must be a junction chamber 1441 : insufficient pump head for pump

1442 : inappropriate pump curve coefficients for pump 1461 : invalid split-flow connection downstream to pumping station 1462 : invalid downstream connection to pumping station

Node data error codes

1111 : invalid loading manhole diameter data 1112 : invalid loading manhole coverage count data 1113 : invalid negative manhole loading factor data 1114 : invalid negative manhole loading pattern data 1115 : unable to add loading to manhole

1119 : given manhole is not a loading manhole 1121 : invalid junction chamber diameter data 1131 : invalid outlet manhole diameter data 1191 : model does not contain any loading manhole 1192 : model does not contain any outlet

1212 : wet well maximum level < 0.0

1213 : wet well minimum level < 0.0

1214 : wet well initial level < 0.0

1221 : wet well minimum level > maximum

1222 : wet well initial level outside minimum/maximum range 1223 : invalid wet well diameter data

InfoSewer™

Modeling Made Easy

Additional Node/Link data error codes

1611 : invalid split-flow percentage range 1621 : invalid infiltration flow type data 1622 : invalid infiltration flow count data 1631 : invalid loading manhole HGL head type Control data error codes

1651 : control ON volume <= OFF volume

1652 : control ON volume outside wet well volume range 1653 : control OFF volume outside wet well volume range

Option data error codes

1821 : peaking factor K < 0.0

1822 : peaking factor P < 0.0

1823 : peaking parameter A < 0.0

1824 : peaking parameter B < 0.0

1825 : peaking parameter C < 0.0

1826 : peaking parameter D < 0.0

1831 : minimum global Design velocity < 0.0

1832 : maximum global Design velocity <= 0.0

1833 : minimum global Design velocity > maximum 1841 : minimum pipe segment distance <= 0.0

1842 : flow tolerance <= 0.0

1843 : Design Manning coefficient <= 0.0

1851 : invalid flow unit definition

1852 : invalid coverage loading unit definition

InfoSewer™

Modeling Made Easy

InfoSewer™

Modeling Made Easy

Output error codes

4001 : unable to allocate output memory 4002 : unable to output module data

4111 : unable to compute loading manhole grade data 4112 : unable to compute junction chamber grade data 4113 : unable to compute outlet grade data File based error codes

5001 : no input file given

5002 : no report file given

5003 : no output file given

5011 : unable to open input data file

5012 : unable to open report file

5021 : unable to open working files for EPS run

InfoSewer™ Modeling Made Easy

Time data error codes

1811 : invalid time field data

1812 : simulation start time < 0.0

1813 : simulation duration <= 0.0

1814 : report time step <= 0.0

1815 : pattern time step <= 0.0

1816 : minimum pipe travelling time <= 0.0

1817 : minimum pump time step <= 0.0

Error Messages & Warnings

After a model run, errors or warnings may be encountered. When you run a simulation in InfoSewer, the simulation may complete successfully. In these cases the Run Status Indicator (stop light) on the Run Manager dialog box shows Green. In other cases, the simulation may complete, but several warnings are issued. In other cases, one or more errors are present in the model input data preventing InfoSewer from properly compiling required data to run a simulation or several errors occur during the simulation.

If the simulation completed with warnings:

Error Codes

If a simulation fails with warnings or errors, you may want to have InfoSewer generate a standard output report as it runs the simulation. This report lists important simulation options and parameters and can report on system performance at each reporting timestep. This report can be used to help you debug any problems. Standard output report options are specified with the Standard Report Manager command prior to running the simulation.

InfoSewer™

Modeling Made Easy

[Error Messages in InfoSewer

During a hydraulic simulation, one or more of the following error statements may be produced. Locate the error number to determine the error being encountered by InfoSewer .

Input data based error codes

1001 : unable to scan given input file

1002 : unable to allocate necessary memory for input data 1003 : data memory application error

1011 : undefined loading type

1012 : undefined wetwell type

1013 : undefined pump type

1014 : undefined pattern type

1015 : undefined curve type

1016 : undefined control type

1021 : undefined manhole

1022 : undefined wet well

1023 : undefined node

1024 : undefined pipe

1025 : undefined pump

1026 : undefined pattern

1027 : undefined curve

1028 : identical end nodes for given link

1031 : duplicate Manhole ID

1032 : duplicate Wet Well ID

1033 : duplicate Pipe ID

1034 : duplicate Pump ID

1035 : duplicate Pattern ID

1036 : duplicate Curve ID

1041 : invalid loading pattern type

1042 : invalid curve application type

1051 : invalid Pattern section order

1052 : invalid Curve section order

1053 : invalid Loading Manhole section order 1054 : invalid Junction Chamber section order 1055 : invalid Outlet Manhole section order 1056 : invalid Wet Well section order

1057 : invalid Pipe section order

1058 : invalid Force-Main section order

1059 : invalid Pump section order

1071 : unrecognized data field

1081 : missing default constant pattern

1082 : missing default criteria curve

1083 : missing default cost curve

1091 : invalid data fields

Node data error codes

1111 : invalid loading manhole diameter data 1112 : invalid loading manhole coverage count data 1113 : invalid negative manhole loading factor data 1114 : invalid negative manhole loading pattern data 1115 : unable to add loading to manhole

1119 : given manhole is not a loading manhole 1121 : invalid junction chamber diameter data 1131 : invalid outlet manhole diameter data 1191 : model does not contain any loading manhole 1192 : model does not contain any outlet

1212 : wet well maximum level < 0.0

1213 : wet well minimum level < 0.0

1214 : wet well initial level < 0.0

1221 : wet well minimum level > maximum 1222 : wet well initial level outside minimum/maximum range 1223 : invalid wet well diameter data

Link data error codes

1311 : given pipe upstream manhole is an outlet 1312 : given gravity main has upstream invert <= downstream invert 1313 : invalid pipe length data

1314 : invalid pipe diameter data

1315 : invalid pipe friction/resistance data 1316 : invalid parallel pipe parallel count data 1331 : invalid Manning coefficient for gravity main 1332 : flow-split pipes do not apply to non-loading manhole nodes 1333 : adverse/flat pipes do not apply to flow-splits 1351 : invalid down-stream outlet node for given force-main 1352 : invalid split-flow configuration for given force-main 1355 : invalid force-main connection at node junction 1361 : invalid force-main group data object 1362 : invalid parallel force-mains

1363 : invalid split force-mains

1364 : invalide intermediate node type for the force-main group 1365 : invalid from node type for the leading force-main 1366 : invalid to node type for the last force-main 1391 : model does not contain any pipe

1411 : invalid parallel pump count data

1412 : invalid parallel pump definition

1421 : invalid pump capacity data

1422 : invalid pump design head data

1423 : invalid pump design flow data

1424 : invalid pump cutoff head data

1425 : invalid pump design head data

1426 : invalid pump design flow data

1427 : invalid pump high head data

1428 : invalid pump high flow data

1431 : downstream manhole for pump must be a junction chamber 1441 : insufficient pump head for pump

1442 : inappropriate pump curve coefficients for pump 1461 : invalid split-flow connection downstream to pumping station 1462 : invalid downstream connection to pumping station

Curve data error codes

1511 : invalid curve size data

1521 : invalid negative level (X-axis) value for wet well curve 1522 : invalid negative volume (Y-axis) value for wet well curve 1523 : wet well curve level (X-axis) values not in ascending order 1524 : wet well curve volume (Y-axis) values not in ascending order 1531 : invalid negative/zero diameter (X-axis) value for criteria curve 1532 : invalid negative d/D (Y-axis) value for criteria curve 1533 : d/D (Y-axis) value > 1.0 for criteria curve 1534 : criteria curve diameter (X-axis) values not in ascending order 1541 : invalid negative/zero diameter (X-axis) value for cost curve 1542 : invalid negative cost (Y-axis) value for cost curve 1543 : cost curve diameter (X-axis) values not in ascending order 1551 : invalid negative inflow (X-axis) value for flow-split curve 1552 : invalid negative flow-split ratio (Y-axis) value for flow-split curve

1553 : flow-split ratio (Y-axis) value > 100% for flow-split curve 1554 : flow-split curve flow (X-axis) values not in ascending order

Additional Node/Link data error codes

1611 : invalid split-flow percentage range 1621 : invalid infiltration flow type data 1622 : invalid infiltration flow count data 1631 : invalid loading manhole HGL head type

Control data error codes

1651 : control ON volume <= OFF volume

1652 : control ON volume outside wet well volume range 1653 : control OFF volume outside wet well volume range

Design data error codes

1711 : minimum Design velocity > maximum 1721 : undefined Design Analysis curve

1722 : diameter of pipe noted below is greater than the maximum diameter value defined in the ANALYSIS Curve

1731 : undefined Design Criteria curve

1732 : undefined Design Replacement curve

1733 : undefined Design Duplication curve

1734 : maximum Replacement curve diameter > maximum Design Criteria curve diameter

1735 : maximum Duplication curve diameter > maximum Design Criteria curve diameter

Time data error codes

1811 : invalid time field data

1812 : simulation start time < 0.0

1813 : simulation duration <= 0.0

1814 : report time step <= 0.0

1815 : pattern time step <= 0.0

1816 : minimum pipe travelling time <= 0.0

1817 : minimum pump time step <= 0.0

Option data error codes

1821 : peaking factor K < 0.0

1822 : peaking factor P < 0.0

1823 : peaking parameter A < 0.0

1824 : peaking parameter B < 0.0

1825 : peaking parameter C < 0.0

1826 : peaking parameter D < 0.0

1831 : minimum global Design velocity < 0.0

1832 : maximum global Design velocity <= 0.0

1833 : minimum global Design velocity > maximum 1841 : minimum pipe segment distance <= 0.0

1842 : flow tolerance <= 0.0

1843 : Design Manning coefficient <= 0.0

1851 : invalid flow unit definition

1852 : invalid coverage loading unit definition

Connectivity error codes

2111 : missing loading manhole downstream connection 2112 : invalid downstream pump station force-main connectio 2113 : invalid intermediate loading manhole in force-mains 2114 : invalid intermediate loading manhole in pump station force-main 2115 : invalid loading manhole connection to pump station 2121 : junction chamber must be connected to force-main/pump 2122 : invalid link connection count to junction chamber 2123 : junction chamber downstream link is not a force-main 2124 : missing downstream link to junction chamber 2125 : invalid upstream link type

2126 : missing upstream link to junction chambe 2131 : invalid connection to outlet

2132 : invalid downstream connection from outlet 2133 : invalid upstream link type

2141 : unsupported multiple downstream links from wet well 2142 : wet well downstream link is not a pum 2143 : wet well upstream link is not a gravity main 2211 : rim elevation less than downstream invert level 2212 : rim elevation less than downstream pipe-main level 2501 : loop found in given network

2502 : multi-loops found in given network

2511 : disconnected nodes

2512 : disconnected links

2531 : unable to organize pipe link topology 2611 : invalid split-flow configuration for non-pipe link 2612 : invalid adverse pipe from split-flow manhole

Computational error codes

3001 : unable to allocate run time memory

3002 : unable to complete module computation 3111 : unable to complete run-time data setup 3211 : incorrect flow-split division

3212 : insufficient pump head

3213 : Pump flow is unreasonably small 3214 : Discharge and inflow based controls do not apply to parallel pumps

3215 : The maximum pump speed is too small to pump the flow

3900 : run-time error

Output error codes

4001 : unable to allocate output memory

4002 : unable to output module data

4111 : unable to compute loading manhole grade data 4112 : unable to compute junction chamber grade data 4113 : unable to compute outlet grade data

File based error codes

5001 : no input file given

5002 : no report file given

5003 : no output file given

5011 : unable to open input data file

5012 : unable to open report file

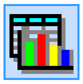
5021 : unable to open working files for EPS run

InfoSewer™

Modeling Made Easy

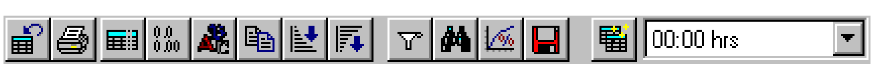
Available Report Types

The following report types are available from the Output Report Manager.

 Choose a report for more information:

Steady State Analysis	*Steady State Design*	*EPS*
Static Loading Manhole Report	Pipe Design Report	EPS Loading Manhole Report
Static Chamber Manhole Report		EPS Chamber Manhole Report
Static Outlet Report		EPS Outlet Report
Static Wet Well Report		EPS Wet Well Report
Static Gravity Main Report		EPS Gravity Main Report
Static Force Main Report		EPS Force Main Report
Static Pump Report		EPS Pump Report
		Range Loading Manhole Report
		Range Gravity Main Report

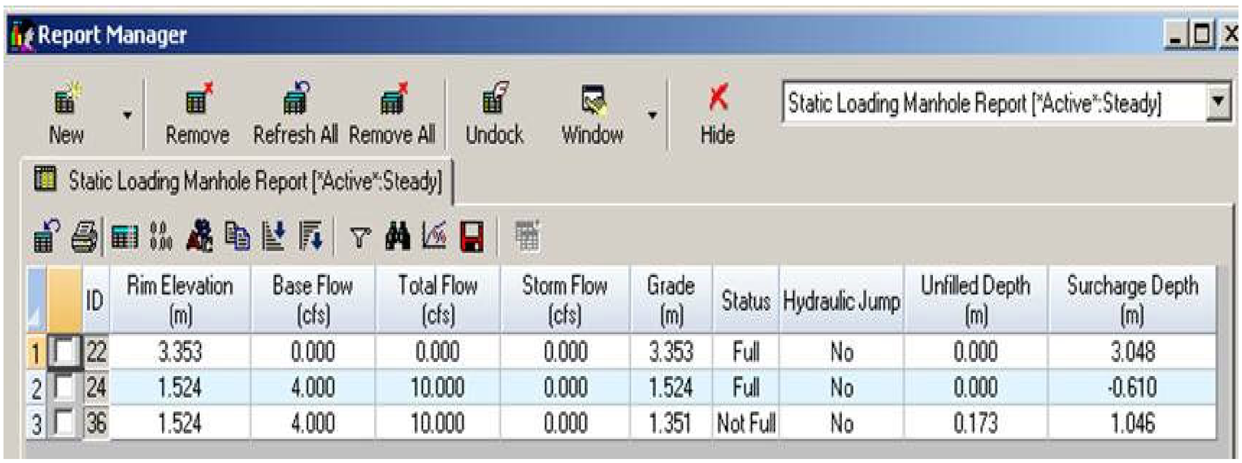
Click on the Report Modification buttons below to learn more about each icon.



Static Loading Manhole Report

Shows steady state simulation results for all manholes in tabular format. The report displays one record for each manhole in the current InfoSewer project. Manhole report columns include the Node Identifier, Rim Elevation, Load, Overload and Grade, surcharge status, occurrence of a hydraulic jump across the node and the unfilled and surcharged depth.

The following variables are displayed on the Static Loading Manhole Report in the Output Report Manager for all or selected manholes:



		ID	Rim Elevation (m)	Base Flow (cfs)	Total Flow (cfs)	Storm Flow (cfs)	Grade (m)	Status	Hydraulic Jump	Unfilled Depth (m)	Surcharge Depth (m)
1		22	3.353	0.000	0.000	0.000	3.353	Full	No	0.000	3.048
2		24	1.524	4.000	10.000	0.000	1.524	Full	No	0.000	-0.610
3		36	1.524	4.000	10.000	0.000	1.351	Not Full	No	0.173	1.046

1. **ID** - Manhole node identifier.
2. **Rim Elevation** - Manhole node elevation, ft (m).
3. **Base Flow** - The base loading applied to the manhole (before peaking), flow units.
4. **Total Flow** - The calculated flow (after peaking), inserted into the manhole, flow units.
5. **Storm Flow** - Peak storm load at the manhole, flow units
6. **Grade** - Manhole node hydraulic grade for the steady state simulation, ft (m).
7. **Status** - Surcharge status of the manhole.
8. **Hydraulic Jump** – Was there a Hydraulic Jump between the incoming and outgoing pipe of the node? The Froude Number defines the

subcritical flow or Supercritical flow. **Froude Number: Fr = V/√(gL) Where: Fr = Froude number V = Velocity g = gravity L = depth of flow** A Froude number greater than 1 is a Supercritical flow whereas a Froude number less than 1 is a subcritical flow. In order to have a hydraulic jump the Froude number needs to be greater than or equal to 1 in a Sewer link. A hydraulic jump occurs when the flow goes from Supercritical flow (Fr > 1) to Subcritical flow (Fr < 1) or from an unstable flow to a stable flow. A hydraulic jump will not occur when a flow goes from subcritical flow (Fr < 1) to a Supercritical flow (Fr > 1). **Sewer flags a node as having a jump if the upstream link (at the upstream end of the link) is Supercritical and the downstream link is Subcritical based on the link Froude Number.**

9. **Unfilled Depth** – Depth between the node Rim Elevation and the Node Grade. A zero value indicates it is full.
10. **Surcharge Depth** - Is the difference of "The Depth of Water of Manhole" and "The Crown of the Highest Connecting Conduits". A positive Surcharge Depth means the node water surface elevation is above the highest pipe crown, a negative depth means that the node depth is below the highest pipe crown.

Static Chamber Manhole Report

Shows steady state simulation results for all chamber manholes in tabular format. The report displays one record for each chamber manhole in the current InfoSewer project. Chamber manhole report columns include the Node Identifier and the Hydraulic Grade.

The following variables are displayed on the Static Chamber Manhole Report in the Output Report Manager for all or selected chamber manholes:

ID	Grade (ft)
JC48	296.6983
JC5	299.0547

1. **ID** - Chamber manhole node identifier.
2. **Grade** - Chamber manhole node hydraulic grade for the steady state simulation, ft (m).

Static Outlet Report

Shows steady state simulation results for all outlets in tabular format. The report displays one record for each outlet in the current InfoSewer project. Outlet report columns include the Node Identifier, Flow and Hydraulic Grade.

The following variables are displayed on the Static Outlet Report in the Output Report Manager for all or selected outlets:

ID	Flow (cfs)	Grade (ft)
OUTLET110	0.7250	278.2744
OUTLET94	2.0250	280.5537

1. **ID** - Outlet node identifier.
2. **Flow** - The peaked flow exiting the system at the outlet structure, flow units.
3. **Grade** - The hydraulic grade at the outlet structure for the steady state simulation, ft (m).

Static Wet Well Report

Shows steady state simulation results for all wet wells in tabular format. The report displays one record for each wet well in the current InfoSewer project. Wet well report columns include the Node Identifier and Hydraulic Grade.

The following variables are displayed on the Static Wet Well Report in the Output Report Manager for all or selected wet wells:

ID	Grade (ft)
WW48	266.0000
WW5	265.0000

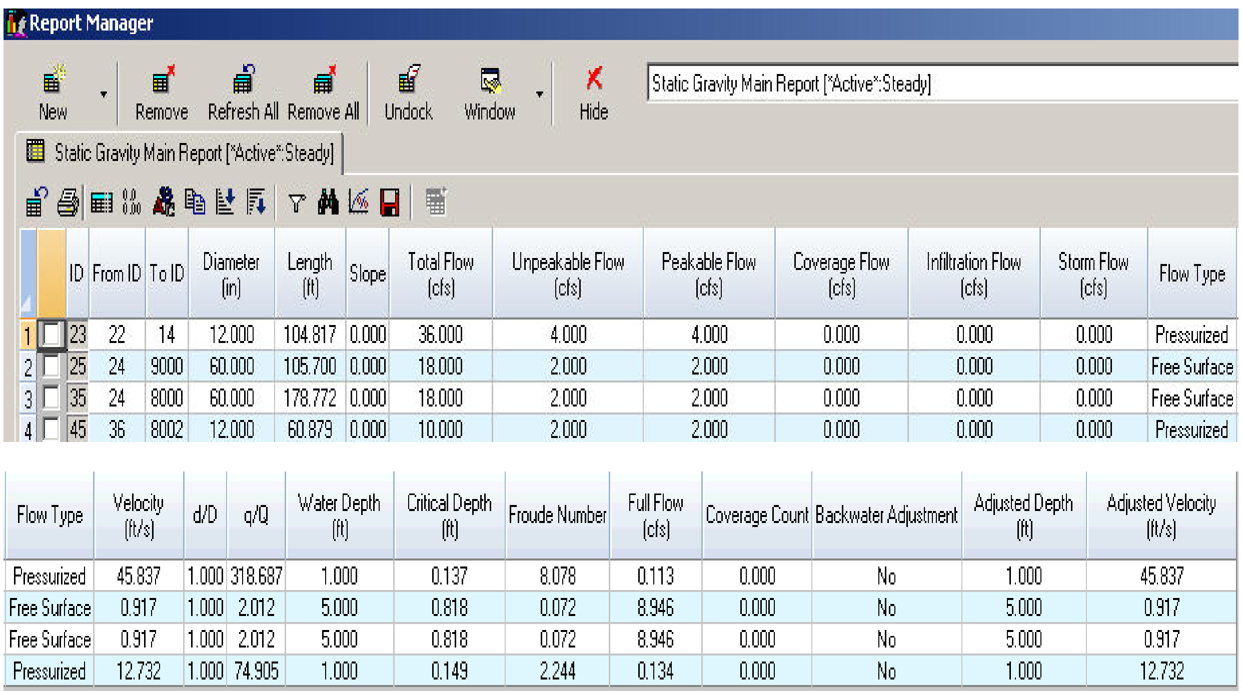
1. **ID** - Wet well node identifier.

2. **Grade** - The hydraulic grade at the outlet structure for the steady state simulation, ft (m).

Static Gravity Main Report

Shows steady state simulation results for all gravity mains in tabular format. The report displays one record for each gravity main in the current InfoSewer project. Gravity main report columns include the Node Identifier, Total Flow, Unpeakable Flow, Peakable Flow, Coverage Flow, Infiltration Flow, Storm Flow, Flow Type, Velocity, d/D, q/Q, Water Depth, Critical Depth, Full Flow, Coverage Count, Backwater Adjustment, Adjusted Depth and Adjusted Velocity.

The following variables are displayed on the Static Gravity Main Report in the Output Report Manager for all or selected gravity mains:



		ID	From ID	To ID	Diameter (in)	Length (ft)	Slope	Total Flow (cfs)	Unpeakable Flow (cfs)	Peakable Flow (cfs)	Coverage Flow (cfs)	Infiltration Flow (cfs)	Storm Flow (cfs)	Flow Type
1	☐	23	22	14	12.000	104.817	0.000	36.000	4.000	4.000	0.000	0.000	0.000	Pressurized
2	☐	25	24	9000	60.000	105.700	0.000	18.000	2.000	2.000	0.000	0.000	0.000	Free Surface
3	☐	35	24	8000	60.000	178.772	0.000	18.000	2.000	2.000	0.000	0.000	0.000	Free Surface
4	☐	45	36	8002	12.000	60.879	0.000	10.000	2.000	2.000	0.000	0.000	0.000	Pressurized

Flow Type	Velocity (ft/s)	d/D	q/Q	Water Depth (ft)	Critical Depth (ft)	Froude Number	Full Flow (cfs)	Coverage Count	Backwater Adjustment	Adjusted Depth (ft)	Adjusted Velocity (ft/s)
Pressurized	45.837	1.000	318.687	1.000	0.137	8.078	0.113	0.000	No	1.000	45.837
Free Surface	0.917	1.000	2.012	5.000	0.818	0.072	8.946	0.000	No	5.000	0.917
Free Surface	0.917	1.000	2.012	5.000	0.818	0.072	8.946	0.000	No	5.000	0.917
Pressurized	12.732	1.000	74.905	1.000	0.149	2.244	0.134	0.000	No	1.000	12.732

1. **ID** - Gravity main identifier.
2. **From Node** - ID of upstream node.
3. **To Node** - ID of downstream node.
4. **Diameter** - Inside pipe diameter for circular channels, in (mm).

5. **Channel Depth** - Maximum depth of a conduit (for non-circular channels only), in (mm).
6. **Channel Width** - Top/Bottom width of a conduit (for non-circular channels only), in (mm).
7. **Channel Left Slope** - Left side slope of a conduit (for trapezoidal and triangular channels only).
8. **Channel Right Slope** - Right side slope of a conduit (for trapezoidal and triangular channels only).
9. **Length** - Pipe length, ft (m).
10. **Slope** - Ratio of the change in vertical distance to the change in horizontal distance, unitless.
11. **Total Flow** - The summation of all flow types, flow unit.
12. **Unpeakable Flow** - The flow to which no peaking is applied, flow unit.
13. **Peakable Flow** - The flow derived based on the Federov peaking equation, flow unit.
14. **Coverage Flow** - The flow derived based on the Harman and Babbitt peaking equation in reference to the contributing population, flow unit.
15. **Infiltration Flow** - The volume of groundwater entering the sewer system from the soil through defective joints, broken or cracked pipes, improper connections, or manhole walls, flow unit.
16. **Storm Flow** - Peak storm load in the pipe, flow units.
17. **Flow Type** - Indicates if the flow is pressurized or free surface.
18. **Velocity** - The speed with which the water is traveling through the pipe, in ft/s (m/s).
19. **d/D** - The ratio of actual flow depth to the diameter of the pipe (full flow depth), unitless.

20. **q/Q** - The ratio of actual flow to the full flow as derived based on Manning's Equation, unitless.
21. **Water Depth** - The depth of water as it is flowing through the pipe, ft (m).
22. **Critical Depth** - The depth of water resulting when the Froude Number is equal to 1.0, ft (m). See Critical Depth
23. **Full Flow** - The capacity of the pipe as evaluated based on Manning's Equation (d/D = 1.0), flow units.
24. **Coverage Count** - The population parameter used in the Harman and Babbitt equations.
25. **Backwater Adjustment** – If the downstream head of the link is greater than the flow depth + the downstream pipe invert then the adjusted depth is one half of the sum of the water surface depth at the upstream and downstream ends of the link.
26. **Adjusted Depth** – The adjusted depth is the average of the upstream plus the downstream adjusted depth, where the upstream adjusted depth is the upstream head minus the upstream pipe invert elevation and the downstream adjusted depth is the downstream head minus the downstream pipe invert elevation. The adjusted depth is the minimum of the pipe diameter and the computed pipe adjusted depth.
27. **Adjusted Velocity** – the adjusted depth is used to calculate the wet area and adjusted velocity = flow/wet area.

Static Force Main Report

Shows steady state simulation results for all force mains in tabular format. The report displays one record for each force main in the current InfoSewer project. Force main report columns include the Node Identifier, from ID, to ID, Diameter, Length, Flow, Velocity and Headloss.

The following variables are displayed on the Static Force Main Report in the Output Report Manager for all or selected force mains:

ID	From ID	To ID	Diameter (in)	Length (ft)	Flow (cfs)	Velocity (ft/s)	Headloss (ft)
FM20	JC5	21	6.0000	240.0000	1.5000	7.6394	8.5547
FM51	JC48	52	6.0000	230.0000	1.5000	7.6394	8.1983

1. **ID** - Force main identifier.
2. **From ID** - ID of upstream node.
3. **To ID** - ID of downstream node.
4. **Diameter** - Inside pipe diameter, in (mm).
5. **Length** - Pipe length, ft (m).
6. **Flow** - The flow passing through the force main (as derived from the upstream pump), flow units. identifier.
7. **Velocity** - The velocity of flow passing through the force main, flow units.

Static Pump Report

Shows steady state simulation results for all pumps in tabular format. The report displays one record for each pump in the current InfoSewer project. Pump report columns include the Node Identifier, from ID, to ID, Flow, Head Increase, Usage and Pump Speed.

The following variables are displayed on the Static Pump Report in the Output Report Manager for all or selected pumps:

		ID	From ID	To ID	Flow (cfs)	Head Increase (ft)	Power (hp)	Usage	Speed
1	☐	5000	9000	16	11.000	32.485	40.541	1.000	1.000
2	☐	5002	8000	26	11.000	23.763	29.657	1.000	1.000
3	☐	5004	8002	40	11.000	10.505	13.110	1.000	1.000

1. **ID** - Pump identifier.
2. **From ID** - ID of upstream node.
3. **To ID** - ID of downstream node.

4. **Flow** - The flow rate passing through the pump, flow units.
5. **Head Increase** - The hydraulic head increase the pump is lifting sewage flows, ft (m).
6. **Power** - The power imparted by the pump, hp (kw)
7. **Usage** - The number of pumps in the simulation.
8. **Speed** – Pump Speed based on the pump affinity laws in the controls.

Pipe Design Report

Shows results for a steady state design simulation in tabular format. The report displays one record for each gravity main in the current InfoSewer project. Pipe design report columns include the Node Identifier, from ID, to ID, diameter, length, original velocity, d/D Ratio, Analysis Flow, Analysis Excess, Analysis d/D Ratio, Design Flow, Design Excess, Design d/D Ratio, Replacement Diameter, Replacement Velocity, Replacement d/D Ratio, Replacement Cost, Parallel Diameter, Parallel Velocity, Parallel d/D Ratio and Parallel Cost.

Click here for details.

EPS Loading Manhole Report

Shows extended period simulation results for all manholes in tabular format at a specified time step. The report displays one record for each manhole in the current InfoSewerproject. Manhole report columns include the Node Identifier, Rim Elevation, Load, Storm Flow, Total Storm Flow, Grade, Surcharge Status, occurrence of a hydraulic jump across the node, Unfilled Depth, Surcharged Depth and Quality.

The following variables are displayed on the EPS Loading Manhole Report in the Output Report Manager for all or selected manholes:

		ID	Rim Elevation (ft)	Load (cfs)	Storm Flow (cfs)	Total Storm Flow (cfs)	Grade (ft)	Status	Hydraulic Jump	Unfilled Depth (ft)	Surcharge Depth (ft)	H2S (mg/l)
1	☐	22	11.000	0.000	0.000	0.000	1.000	Not Full	No	10.000	0.000	0.000
2	☐	24	5.000	4.000	0.000	0.000	5.000	Full	No	0.000	-2.000	0.922
3	☐	36	5.000	4.000	0.000	0.000	5.000	Full	No	0.000	4.000	0.903

1. **ID** - Manhole node identifier.
2. **Rim** Elevation - Manhole node elevation, ft (m).
3. **Load** - The base loading applied to the manhole at that particular time step, flow units.
4. **Storm Flow** - The storm flow entering the system at the manhole, flow units
5. **Total Storm Flow** -Storm load at the manhole including contributions from upstream subcatchments, flow units.
6. **Grade** - Manhole node hydraulic grade for the steady state simulation, ft (m).
7. **Status** - Surcharge status of the manhole.
8. **Hydraulic Jump** – Was there a Hydraulic Jump between the incoming and outgoing pipes of the node?
9. **Unfilled Depth** – depth between the node Rim Elevation and the Node Grade. A zero value indicates it is full.
10. **Surcharge Depth** - is the difference of “The Depth of Water of Manhole” and “The Crown of the Highest Connecting Conduits”. A positive Surcharge Depth means the node water surface elevation is above the highest pipe crown, a negative depth means that the node depth is below the highest pipe crown.
11. **Quality** - Reports the simulated water quality. The model supporting simulation of seven different water quality modules including (hydrogen sulfide, corrosion, sediment transport and deposition, BOD, pollutant (contaminant) transport, time of concentration (flow age), and trace).

EPS Chamber Manhole Report

Shows extended period simulation results for all chamber manholes in tabular format at a specific timestep. The report displays one record for each chamber manhole in the current InfoSewer project. Chamber manhole report columns include the Node Identifier, Hydraulic Grade and Quality.

The following variables are displayed on the EPS Chamber Manhole Report in the Output Report Manager for all or selected chamber manholes:

		ID	Grade (ft)	H2S (mg/l)
1	☐	10	1.000	0.000
2	☐	12	0.000	0.000
3	☐	16	0.000	0.000
4	☐	18	0.000	0.000
5	☐	20	0.000	0.000
6	☐	26	0.000	0.000
7	☐	30	0.000	0.000
8	☐	40	0.000	0.000

1. **ID** - Chamber manhole node identifier.
2. **Grade** - Chamber manhole node hydraulic grade for the extended period simulation timestep, ft (m).
3. **Quality** - Reports the simulated water quality. The model supporting simulation of seven different water quality modules including (hydrogen sulfide, corrosion, sediment transport and deposition, BOD, pollutant (contaminant) transport, time of concentration (flow age), and trace).

EPS Outlet Report

Shows extended period simulation results for all outlets in tabular format at a specific timestep. The report displays one record for each outlet in the current InfoSewer project. Outlet report columns include the Node Identifier, Flow, Hydraulic Grade and Quality.

The following variables are displayed on the EPS Outlet Report in the Output Report Manager for all or selected outlets:

		ID	Flow (cfs)	Grade (ft)	H2S (mg/l)
1	☐	14	22.000	1.000	0.000

1. **ID** - Outlet node identifier.

2. **Flow** - The peaked flow exiting the system at the outlet structure, flow units.

3. **Grade** - The hydraulic grade at the outlet structure for the extended period simulation timestep, ft (m).

4. **Quality** - Reports the simulated water quality. The model supporting simulation of seven different water quality modules including (hydrogen sulfide, corrosion, sediment transport and deposition, BOD, pollutant (contaminant) transport, time of concentration (flow age), and trace).

EPS Wet Well Report

Shows extended period simulation results for all wet wells in tabular format at a specific timestep. The report displays one record for each wet well in the current InfoSewer project. Wet well report columns include the Node Identifier, Bottom Elevation, Water Level, Hydraulic Grade, Stored Volume, Percent Full, Overflow and Quality.

The following variables are displayed on the EPS Wet Well Report in the Output Report Manager for all or selected wet wells:

		ID	Bottom Elevation (ft)	Level (ft)	Grade (ft)	Volume (MG)	% Volume (%)	Overflow (MG)	H2S (mg/l)
1	☐	9000	-5.000	3.268	-1.732	0.002	32.683	0.000	0.921
2	☐	8000	-5.000	3.268	-1.732	0.002	32.683	0.000	0.921
3	☐	8002	-5.000	6.820	1.820	0.004	68.205	0.000	0.942

1. **ID** - Wet well node identifier.

2. **Bottom Elevation** - Elevation of the wet well above datum, ft (m).

3. **Level** - Water level above the bottom elevation of the wet well, ft (m).

4. **Grade** - The hydraulic grade at the outlet structure for the steady state simulation, ft (m).

5. **Volume** - The total volume of sewage being stored in the wet well, volume units.

6. **% Volume** - Percent full status of the wet well as calculated between the high and low water levels specified for the wet well, %.

7. **Overflow** - Amount of water stored in the wet well that exceeds the storage capacity of the wet well. This field will only be populated when the wet well is full and when flows entering the wet well exceed the pumping capacity of the lift station.
8. **Quality** - Reports the simulated water quality. The model supporting simulation of seven different water quality modules including (hydrogen sulfide, corrosion, sediment transport and deposition, BOD, pollutant (contaminant) transport, time of concentration (flow age), and trace).

EPS Gravity Main Report

Shows extended period simulation results for all gravity mains in tabular format at a specific timestep. The report displays one record for each gravity main in the current InfoSewer project. Gravity main report columns include the Node Identifier, from ID, to ID, Diameter, Length, Slope, Flow, Flow Type, Velocity, Water Depth, Froude Number, Overflow, Backwater Adjustment, Adjusted Depth, Adjusted Velocity, d/D ratio, q/Q ratio and Quality.

The following variables are displayed on the EPS Gravity Main Report in the Output Report Manager for all or selected gravity mains:

		ID	From ID	To ID	Diameter (in)	Length (ft)	Slope	Flow (cfs)	Flow Type	Velocity (ft/s)	Water Depth (ft)	Froude Number	Overflow (MG)	Backwater Adjustment	Adjusted Depth (ft)	Adjusted Velocity (ft/s)	d/D	q/Q	H2S (mg/l)
1	☐	23	22	14	12.000	104.817	0.000	22.000	Pressurized	28.011	1.000	4.936	0.000	No	1.000	28.011	1.000	194.753	0.000
2	☐	25	24	9000	60.000	105.700	0.000	7.500	Free Surface	0.510	3.503	0.050	0.000	No	3.503	0.510	0.701	0.838	0.927
3	☐	35	24	8000	60.000	178.772	0.000	7.500	Free Surface	0.510	3.503	0.050	0.000	No	3.503	0.510	0.701	0.838	0.927
4	☐	45	36	8002	12.000	60.879	0.000	4.000	Pressurized	5.093	1.000	0.898	0.000	Yes	1.000	5.093	1.000	29.962	0.961

	ID	From ID	To ID	Diameter (mm)	Channel Depth (mm)	Channel Width (mm)	Channel Left Slope	Channel Right Slope	Length (m)	Slope	Total Flow (cfs)
☐	23	22	14	304.800					31.948	0.000	36.000
☐	25	24	9000		304.800	304.800	0.000	0.000	32.217	0.000	18.000
☐	35	24	8000	1,524.000					54.490	0.000	18.000
☐	45	36	8002	914.400					18.556	0.016	10.000

1. **ID** - Gravity main identifier.
2. **From Node** - ID of upstream node.
3. **To Node** - ID of downstream node.
4. **Diameter** - Inside diameter of circular pipes, in (mm).

5. **Channel Depth** - Maximum depth of non-circular conduits, in (mm).
6. **Channel Width** - Top/Bottom width of non-circular conduits, in (mm).
7. **Channel left Slope** - Left side slope of trapezoidal and triangular channels.
8. **Channel Right Slope** - Right side slope of trapezoidal and triangular channels.
9. **Length** - Pipe length, ft (m).
10. **Slope** - The calculated slope of the pipe (upstream - downstream)/length, unitless.
11. **Flow** - The calculated flow at the specified timestep, flow units.
12. **Flow Type** - Indicates if the flow type is pressurized or open channel (free surface).
13. **Velocity** - The velocity of flow at the specified timestep, ft/sec (m/sec).
14. **Water Depth** - The depth of water at the specified timestep, ft (m).
15. **Overflow** - Accumulated volume of wastewater that is in excess of full flow capacity of the channel, volume units. This parameter is calculated only for channels that are open to atmosphere (e.g., open rectangular, open parabolic, open trapezoidal, and open triangular).
16. **Backwater Adjustment** – If the downstream head of the link is greater than the flow depth + the downstream pipe invert then the adjusted depth is one half of the sum of the water surface depth at the upstream and downstream ends of the link.
17. **Adjusted Depth** – The adjusted depth is the average of the upstream plus the downstream adjusted depth, where the upstream adjusted depth is the upstream head minus the upstream pipe invert elevation and the downstream adjusted depth is the downstream head minus the downstream pipe invert elevation. The adjusted depth is the minimum of the pipe diameter and the computed pipe adjusted depth.
18. **Adjusted Velocity** – the adjusted depth is used to calculate the wet area and adjusted velocity = flow/wet area.

19. **d/D** - The d/D ratio of the flow at the specified timestep, unitless.
20. **q/Q** - The q/Q ration of the flow rate at the specified timestep, unitless.
21. **Quality** - Reports the simulated water quality. The model supporting simulation of seven different water quality modules including (hydrogen sulfide, corrosion, sediment transport and deposition, BOD, pollutant (contaminant) transport, time of concentration (flow age), and trace).

EPS Force Main Report

Shows extended period simulation results for all force mains in tabular format at a specified timestep. The report displays one record for each force main in the current InfoSewer project. Force main report columns include the Node Identifier, from ID, to ID, Diameter, Length, Flow, Velocity and Headloss.

The following variables are displayed on the Static Force Main Report in the Output Report Manager for all or selected force mains:

ID	From ID	To ID	Diameter (in)	Length (ft)	Flow (cfs)	Velocity (ft/s)	Headloss (ft)
FM20	JC5	21	6.0000	240.0000	0.0000	0.0000	0.0000
FM51	JC48	52	6.0000	230.0000	0.0000	0.0000	0.0000

1. **ID** - Force main identifier.
2. **From ID** - ID of upstream node.
3. **To ID** - ID of downstream node.
4. **Diameter** - Inside pipe diameter, in (mm).
5. **Length** - Pipe length, ft (m).
6. **Flow** - The flow passing through the force main (as derived from the upstream pump), flow units. identifier.
7. **Velocity** - The velocity of flow passing through the force main, flow units.
8. **Headloss** - The total headloss experienced across the force main, ft (m).

EPS Pump Report

Shows extended period simulation results for all pumps in tabular format at a specified timestep. The report displays one record for each pump in the current InfoSewer project. Pump report columns include the Node Identifier, from ID, to ID, Flow, Head Increase, Usage and Pump Speed.

The following variables are displayed on the EPS Pump Report in the Output Report Manager for all or selected pumps:

		ID	From ID	To ID	Flow (cfs)	Head Increase (ft)	Power (hp)	Usage	Speed
1	☐	5000	9000	16	11.000	32.485	40.541	1.000	1.000
2	☐	5002	8000	26	11.000	23.763	29.657	1.000	1.000
3	☐	5004	8002	40	11.000	10.505	13.110	1.000	1.000

1. **ID** - Pump identifier.
2. **From ID** - ID of upstream node.
3. **To ID** - ID of downstream node.
4. **Flow** - The flow rate passing through the pump, flow units.
5. **Head Increase** - The hydraulic head increase the pump is lifting sewage flows, ft (m).
6. **Power** - The power imparted by the pump, hp (kw)
7. **Usage** - The number of pumps in the simulation.
8. **Speed** – Pump Speed based on the pump affinity laws in the controls.

Range Loading Manhole Report

The Range Loading Manhole Report displays the maximum value of the load node grade (compared across the entire simulation period), and values of some other essential load node results corresponding to the time of occurrence of the maximum grade.

Ranges allow the user to see, in a report format, the maximum values experienced for any loading manhole in the system over the EPS. The reported variables include Maximum Grade, Maximum Grade Time, Load, Status, and Depth.

		ID	Maximum Grade (ft)	Maximum Grade Time	Load (cfs)	Status	Depth (ft)
1		44	278.0000	02:00 hr	1.5000	Full	2.0000
2		52	293.0000	00:00 hr	0.5000	Full	0.8000
3		54	292.9000	01:00 hr	1.0000	Full	1.9000
4		901	278.4396	14:00 hr	0.5000	Not Full	0.4396
5		903	281.7786	02:00 hr	1.5000	Not Full	0.3136
6		917	284.0000	13:00 hr	2.0000	Full	0.8000
7		918	280.5329	02:00 hr	1.5000	Not Full	0.5329
8		919	274.7835	14:00 hr	0.5000	Not Full	0.3835
9		920	273.4011	14:00 hr	0.5000	Not Full	0.4011
10		921	296.5000	02:00 hr	1.5000	Full	0.7000

Range Gravity Main Report

The Range Gravity Main Report displays the maximum values for all pipes during the entire extended period simulation One record is displayed for each component of the selected component type.

Ranges allow the user to see, in a report format, the maximum values experienced for any gravity main in the system over the EPS. Gravity Main Reports include Maximum Flow, Maximum Flow Time, Maximum Velocity, Maximum Water Depth, Maximum d/D,Maximum Overflow, Maximum q/Q, Maximum Critical Depth, and maximum quality.

		ID	From ID	To ID	Maximum Flow	Maximum Flow Time	Maximum Velocity	Maximum Water Depth	Maximum d/D	Maximum Overflow	Maximum q/Q	Maximum Critical Depth	H2S (mg/l)
1		53	52	54	4.0315	13:05 hr	3.2252	1.0000	1.0000	0.0000	1.1687	0.6861	0.2152
2		911	54	OUTLET94	11.6724	13:59 hr	14.8618	1.0000	1.0000	0.0000	5.3359	0.9967	0.2013
3		913	901	44	6.8075	13:59 hr	3.5242	0.7188	0.7188	0.0000	0.4898	0.6838	0.0000
4		917	903	918	3.5194	13:59 hr	4.2552	0.6431	0.6431	0.0000	0.3079	0.7191	0.1224
5		929	917	903	2.0670	13:00 hr	5.9216	0.6667	1.0000	0.0000	1.3667	0.6308	0.1168
6		930	918	901	4.8892	13:59 hr	3.8169	0.8003	0.8003	0.0000	0.5521	0.8202	0.0000
7		934	921	922	4.1599	01:42 hr	4.9968	0.9990	0.9990	9.3158	0.9998	1.0000	0.0000
8		935	922	923	6.1538	13:59 hr	4.9230	1.0000	1.0000	0.0000	1.2369	0.9096	0.1119
9		936	923	924	7.3450	13:59 hr	5.8760	1.0000	1.0000	0.0000	1.6506	1.0000	0.1109
10		937	924	54	5.7439	01:10 hr	4.5974	0.9995	0.9995	3.3018	0.9999	0.8688	0.0000

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Pumps

A pump is a link that imparts energy to a fluid thereby raising its hydraulic head. The relationship describing the head imparted to a fluid as a function of its flow rate through the pump is termed the pump characteristic curve.

In a sewer collection system, pumps are only allowed in conjunction with wet wells. Therefore, a wet well and chamber (manhole) must first be digitized prior to creating a pump.

How do I...

- **Create a pump (types of pumps)?**
- **Edit a pump?**
- **Delete a pump?**
- **Recall a deleted pump?**
- **Redraw a pump?**
- **Create Pumps in Parallel?**

Create a Pump

To create a pump, click on the add pump link icon or from the **Create** menu, click **Add Pump**.

Once the command is initiated, a pencil cursor will appear. Select the *Wet Well* for which the pump is to be placed and begin digitizing the location of the pump. Intermediate shape-defining vertices can be placed by clicking the mouse in any location. Configure the pump accordingly and double-click the mouse on top of the *Chamber* to end the digitization process.

Once the pump has been located, a dialog box will become present requesting the ID number of the pump, a description and the type of pump being created. Once these data are entered, press OK to end the command.

Types of Pumps

There are 3 different types of pumps the user is able to input into H2OMAP Sewer. Click on any pump to learn more.

0. ***Fixed Capacity***
1. ***Design Point Curve***
2. ***Exponential 3-Point Curve***

Edit a Pump

Graphic Edit - To edit the pump graphically, select the Edit Link Vertex icon and click on the pump or from the **Edit** menu, select the **Edit Link Vertex** command. To add a vertex, click on the pump two times - one to select the pump, the other to add a vertex. Once a vertex is added the selected vertex will turn blue. By clicking on the blue box and holding down the left mouse button, the user can drag the vertex to its new location.

Data Edit - To edit the data related to a pump, first select the pump using the select tool. Once selected, the user is able to edit the database data related to the pump under the Link tab of the Attribute Browser dialog box.

Delete a Pump

To delete a pump, from the **Edit** menu, select **Delete Link**. From here the user selects the desired pump to be deleted and is asked by H2OMAP Sewer to confirm the deletion of this pump. This pump is deleted graphically but can still be recalled from the database using the recall command. However, if Auto Database Packing is enabled (Tools -> Preferences), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Pump

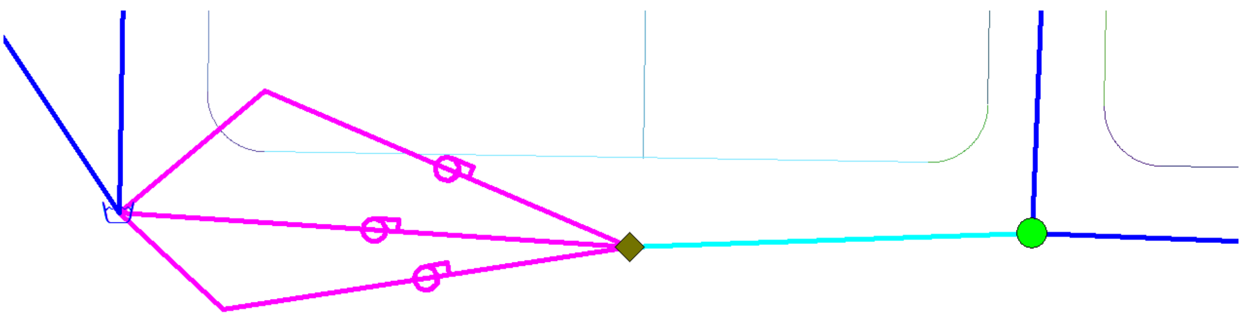
To recall a deleted pump, go to **Utilities** menu, point to **Recall**, then select **Link**. In order to perform the recall, the user must know the ID of the pump that was deleted. If the ID is unknown, select the Show Deleted Links command from the Recall menu to see a list of ID's that have been deleted.

Redraw a Pump

Redrawing a pump is the same as modifying its geometry. To redraw a pump, select the Redraw Link icon or from the **Edit** menu, select the **Redraw Link** command. Once initialized, the mouse pointer will change to a crosshair. At this point the user is able to redigitize the new location of the pump currently highlighted. This command is dependent upon highlighting the pump in question first, then changing its geometry via this command.

Create Pumps in Parallel

H2OMAP Sewer allows you to model an unlimited number of pumps in parallel as shown in the figure below. Each pump may have unique properties.



Pumps in parallel of the same type (i.e. all fixed capacity pumps or pumps described by characteristic curves) can be represented with one pump by specifying the number of parallel pumps in the *PARALLEL* field of the attribute browser. Pumps described by characteristic curves can be of exponential or design point type.

For Extended Period and Dynamic Simulations (EPS/Dynamic), InfoSewer allows you to control the on-off status of each pump based on the upstream wet-well levels or volumes.

Other Related Topics - Data Elements, Manhole, Pipes, Pumps, Wet Well

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

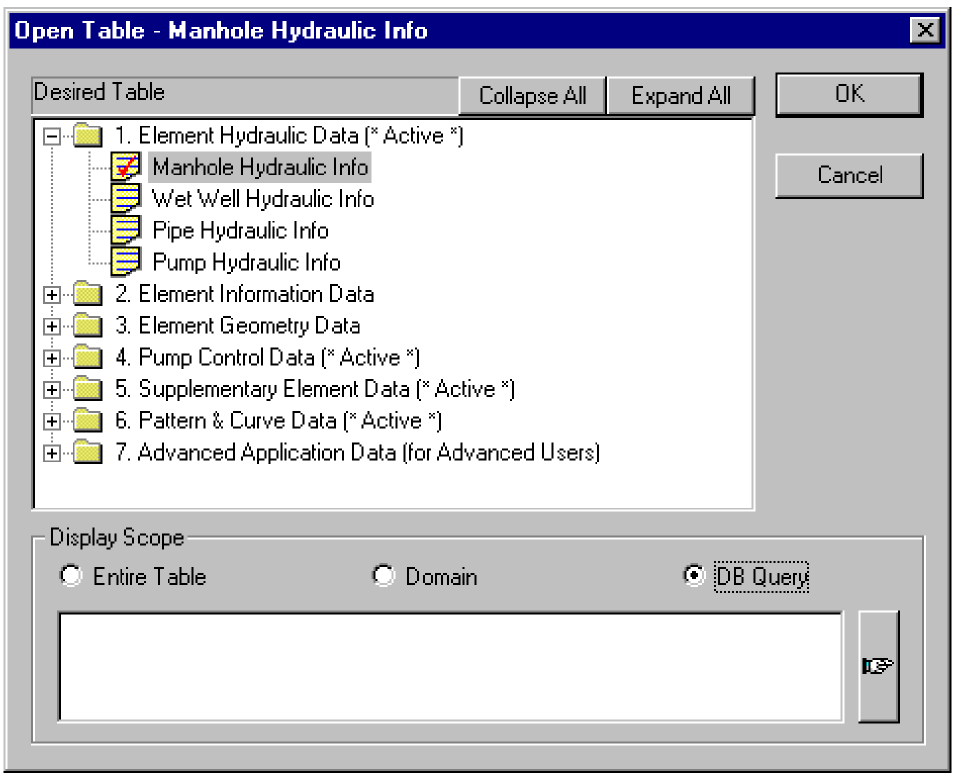
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

DB Editor

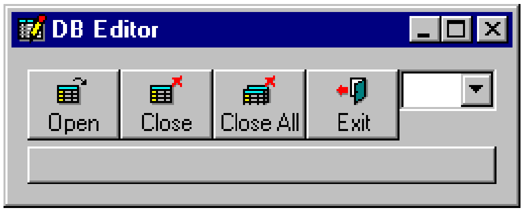
The Database Editor allows the user to open any InfoSewer database and edit user input fields (except ID). To access the DBEditor, from the **Edit** menu, select **DB Tables** to see the Open Table dialog box below.

From the Open Table dialog box, the user is able to see that InfoSewer stores database tables in seven separate folders. After selecting the desired table from under the relevant folder, the user has the option of choosing a display scope found on the bottom of the dialog box to open the desired records of the database. Once this is done, click the OK button to open the database table.



(Note: If the user has created a domain from the Domain Manager, then in order to edit only the domain table data, the user must check the Domain radial button prior to clicking OK. The user can also create a new database query from the DBEditor by clicking on the icon.)

Once a table has been opened, the user is able to control table operations via the DB Editor icons shown below.



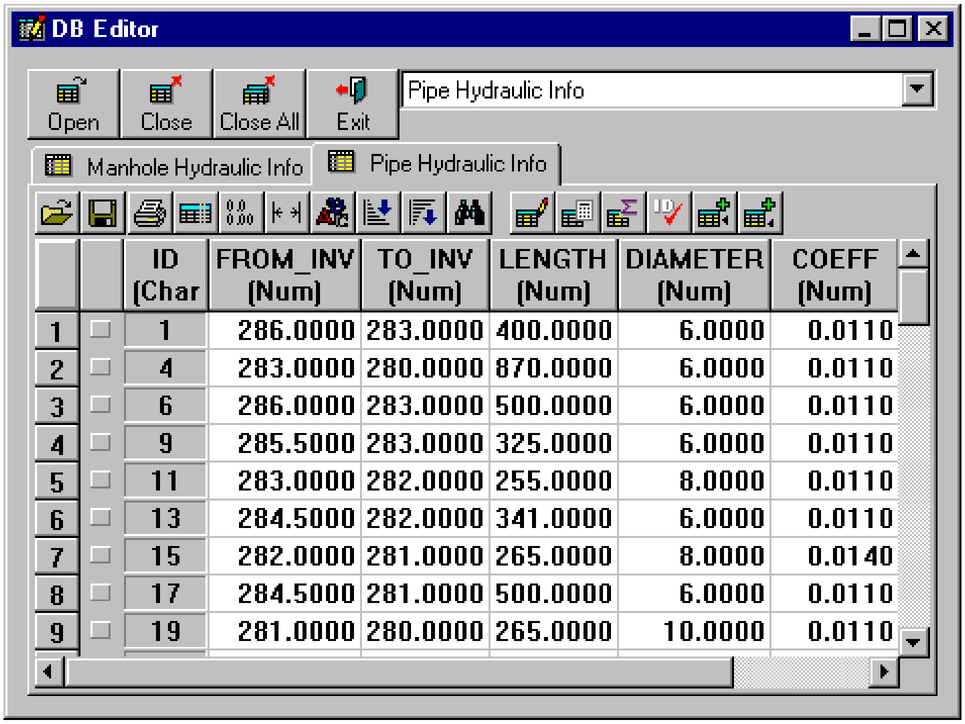
- **Open** - Open another database table as shown in the DB Table dialog box.
- **Close** - Close the currently selected database table.
- **Close All** - Close All open databases.
- **Exit** - Exit the DB Editor.

Features of the DB Editor

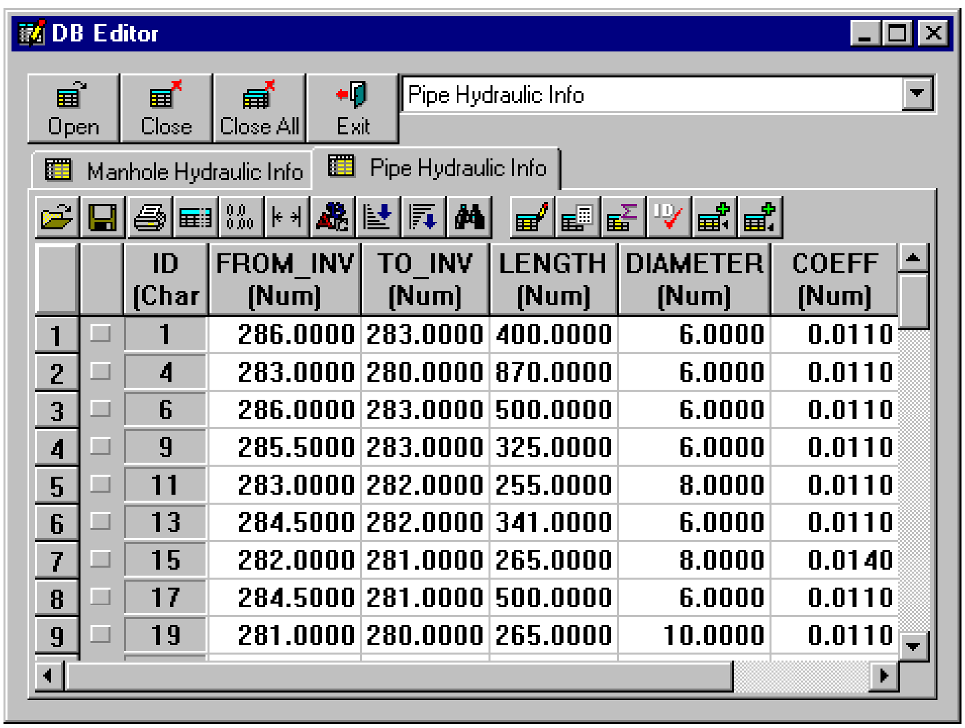
The DBEditor allows the user to:

- **View Data** - View (including custom format) modeling data tables.
- **Add/Delete Custom Fields** - Customize data tables by adding user-defined fields. Click here for more information on adding custom fields.
- **Perform Block Edits** - Allows the user to edit table data values, one record at a time or numerous records simultaneously. The Block Editor is a powerful tool when using the used to perform block edits.

Once a table has been opened, the following dialog box will appear. Click on any of the elements or icons below to learn more about that feature:



		ID (Char	FROM_INV (Num)	TO_INV (Num)	LENGTH (Num)	DIAMETER (Num)	COEFF (Num)
1		1	286.0000	283.0000	400.0000	6.0000	0.0110
2		4	283.0000	280.0000	870.0000	6.0000	0.0110
3		6	286.0000	283.0000	500.0000	6.0000	0.0110
4		9	285.5000	283.0000	325.0000	6.0000	0.0110
5		11	283.0000	282.0000	255.0000	8.0000	0.0110
6		13	284.5000	282.0000	341.0000	6.0000	0.0110
7		15	282.0000	281.0000	265.0000	8.0000	0.0140
8		17	284.5000	281.0000	500.0000	6.0000	0.0110
9		19	281.0000	280.0000	265.0000	10.0000	0.0110



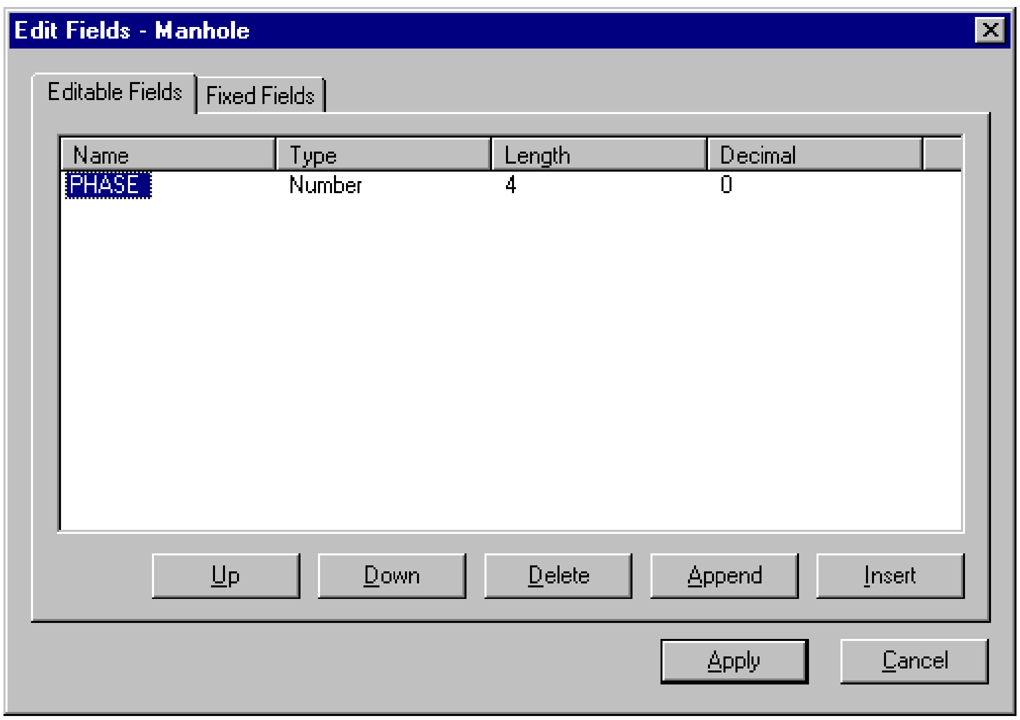
		ID (Char	FROM_INV (Num)	TO_INV (Num)	LENGTH (Num)	DIAMETER (Num)	COEFF (Num)
1		1	286.0000	283.0000	400.0000	6.0000	0.0110
2		4	283.0000	280.0000	870.0000	6.0000	0.0110
3		6	286.0000	283.0000	500.0000	6.0000	0.0110
4		9	285.5000	283.0000	325.0000	6.0000	0.0110
5		11	283.0000	282.0000	255.0000	8.0000	0.0110
6		13	284.5000	282.0000	341.0000	6.0000	0.0110
7		15	282.0000	281.0000	265.0000	8.0000	0.0140
8		17	284.5000	281.0000	500.0000	6.0000	0.0110
9		19	281.0000	280.0000	265.0000	10.0000	0.0110

Adding Custom Database Fields

You can customize InfoSewer by adding additional fields to database tables. Custom fields will be available for inclusion on output reports and for inclusion in queries and query sets.

To add, reorder, or remove custom fields, perform the following:

- Open the Database Editor and click the Open button.
- From the dialog box, select the desired table from folder 2. Element Information Data.
- From the Database Editor dialog box, click on the Edit Fields icon to add [Insert, Append], reorder [Up, Down], or remove [Delete] custom fields (see below). The user must know if the desired field to be added will be character or numeric based. Once a field has been added to the selected database, it will be available for editing via the Attribute Browser (under the Information tab). You can also choose to Block Edit the new field with a pre-defined value.



The DB Editor provides a means to view all the DB Tables. Any number of DB Tables can be open for inspection, modification and/or data edit. All the data corresponding to the InfoSewer Project may be accessed through these DB Tables.

For more information about the different types of data stored in the InfoSewer DB Tables, please refer to the section on DB Tables.

The DBEditor allows the user to:

- **View Data** - View (including custom format) modeling data tables.
- **Add/Delete/Modify data -** Click on any field to change it. The fields that cannot be edited are colored in gray.
- **Add/Delete Custom Fields** - Customize data tables by adding user-defined fields. Click here for more information on adding custom fields.
- **Perform Block Edits** - Allows the user to edit table data values, one record at a time or numerous records simultaneously. The Block Editor is a powerful tool when using the used to perform block edits.

Once a table has been opened, the following dialog box will appear.

Click on any of the elements or icons below to learn more about that feature:

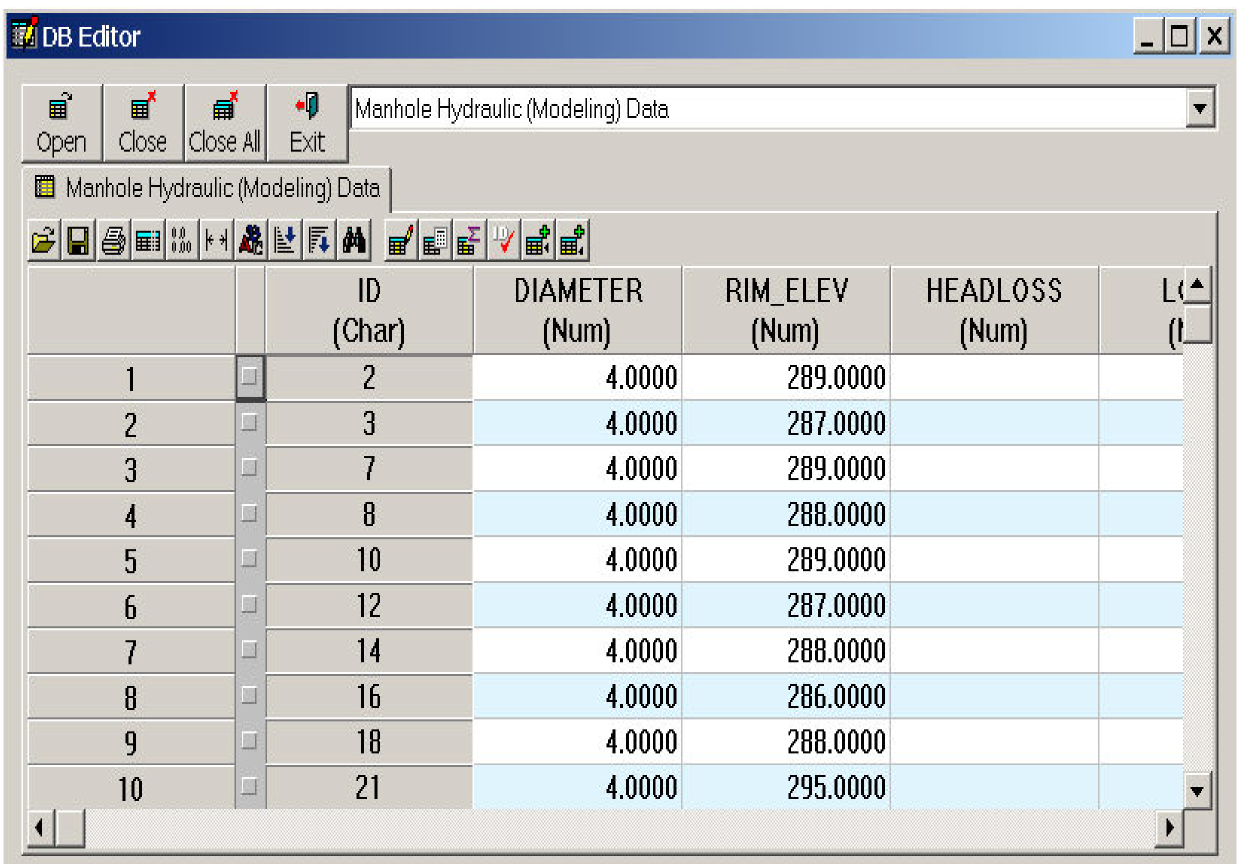


Table Records

- **Table Record Number** – A sequential counter of table records. Used for visual reference only.
- **Save Status** – Indicates whether or not a record has been modified. Any records that have been modified will be marked with a red check . To save any modifications, choose the Save button at the top of the table record display.
- **ID** – InfoSewer component identifier used for reference by other InfoSewer functions and used for modeling. The contents of this field cannot be edited from the Database Editor.
- **Table Fields** – Fields specific to the current table. The field data type (Double), (Num), (Char), etc. is displayed directly below the field name as an indication of the acceptable data format for that field. Field order for fixed fields (those required by InfoSewer) cannot be changed. Field order for custom fields can be changed.

Click here for more information on the different InfoSewer Tables.

Additional properties

- Empty numeric data will display as “0.00” (depending on user-specified decimal precision). Empty character data will display as empty data cells.
- Selected records (rows), fields (columns), or individual data values (cells) will be highlighted in black.
- Non-editable fields are displayed in gray. These fields are typically graphical/locational in nature and must be edited using other InfoSewer functions.

Other Related Topics - Advanced Application Data Tables, Pump Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Pattern & Curve Data Tables, Supplementary Data Tables, DB Editor, DB Tables, Stormwater Modeling Data

---

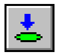
InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

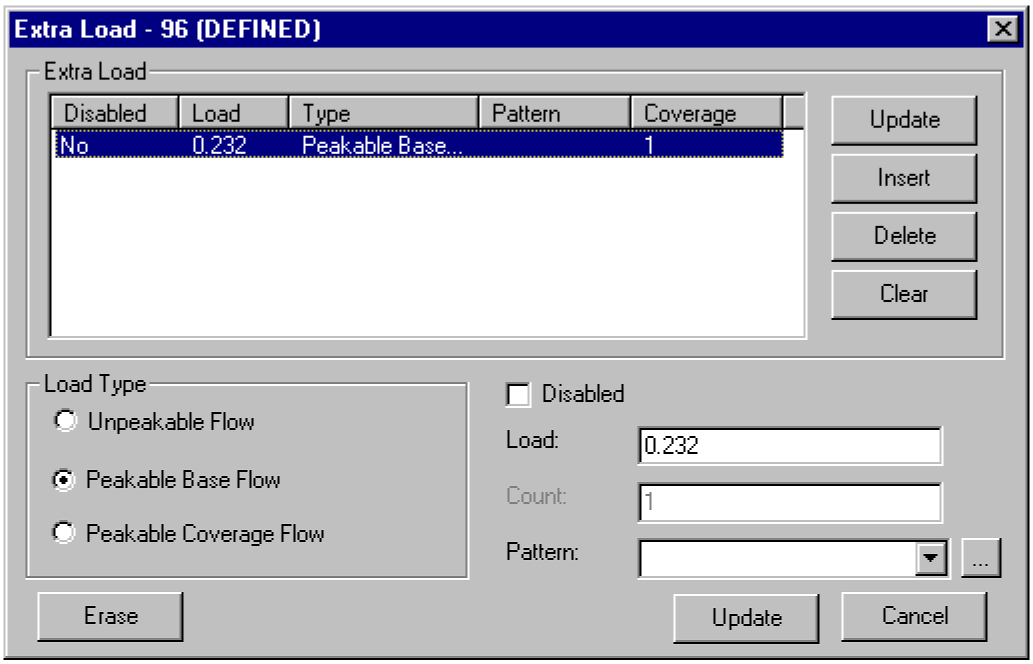
Modeling Made Easy

Extra Load 

InfoSewer allows for ten base loadings that can be applied via the Attribute Browser. Additional loads can be applied by selecting the Extra Load icon shown above.

Adding Extra Loads

To add an extra load, select the manhole where an extra load is desired and click on the extra load icon at the top of the Attribute Browser. Once this is done, the following dialog box will be displayed.



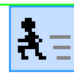
The next step is to specify the load type and enter the base loading (load or count, depending on the load type). If the user is performing a steady state simulation, click the Insert button to add the extra load to the manhole. If an EPS simulation is required, ensure that a pattern is also supplied with the load statement.

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

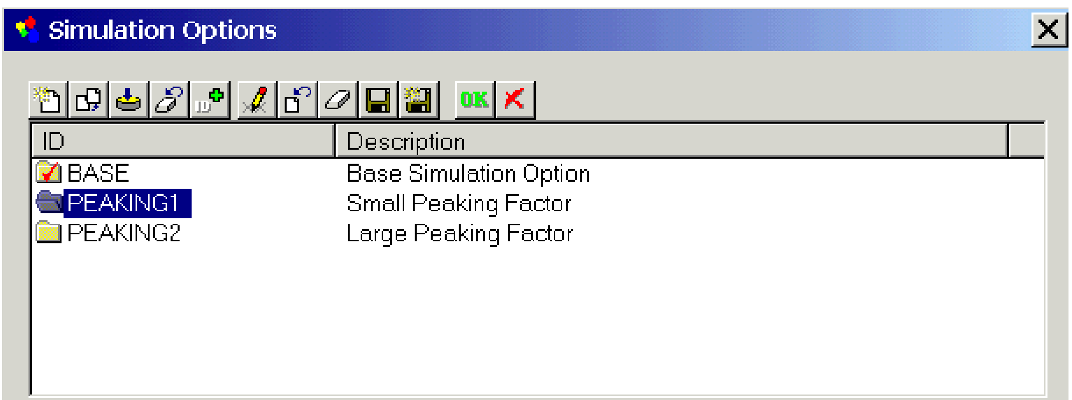
Simulation Options 

The Simulation Options dialog box is used to adjust some facet of a model prior to a model run. Simulation options are usually altered to be associated with a given scenario.

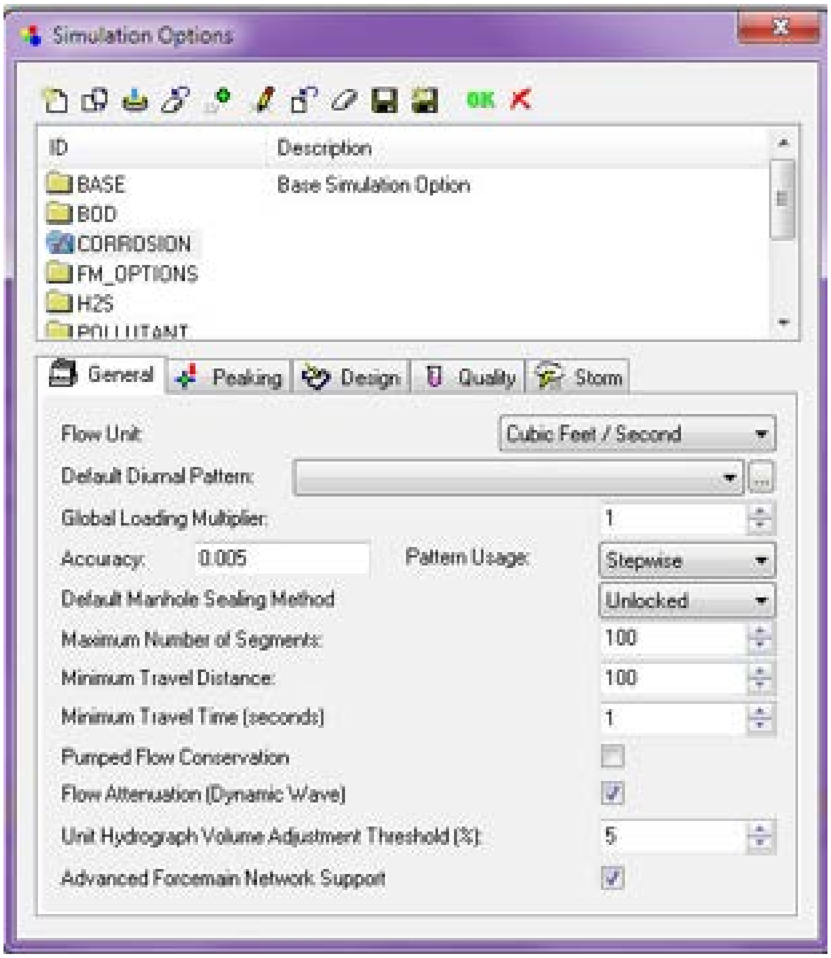
For example, suppose the user wishes to specify a peaking equation for one model run that differs from a second peaking equation. This can be accomplished through the Simulation Options where one option can be tied to one scenario while another option is specified for a different scenario.

To create a simulation option, from the Control Center click on the Operation Data tab and select Simulation Options. Click on the New button at the top of the Control Center dialog box and supply a unique name and description for the simulation option. When the OK button is chosen, the following dialog box will appear.

Click on any icon or tab below to learn more.



The General Tab



- **Flow Unit -** Units of flow measurement (cfs, gpm, mgd, imgd, afd, L/s, L/m, MLd, m3/h, and m3/d).
- **Default Diurnal Pattern -** Determines which diurnal pattern will be applied to an EPS simulation when a unique pattern has not been assigned to an individual manhole(s).
- **Global Loading Multiplier -** Used to assign a multiplier in which global loadings are to be multiplied. For example, a MAXDAY EPS model scenario may have a global loading multiplier of 2.5.
- **Accuracy -** Convergence criterion used to signal that a solution has been found to the nonlinear equations that govern network hydraulics. Trials end when the relative change in pipe flow rates between two successive iterations is less than this number. Suggested value is 0.001.
- **Diurnal Pattern Usage -** Select whether the pattern will linearly interpolate intermediate values (continuous) or evaluate patterns in a stepwise fashion. Click

here to learn more about pattern representation options.

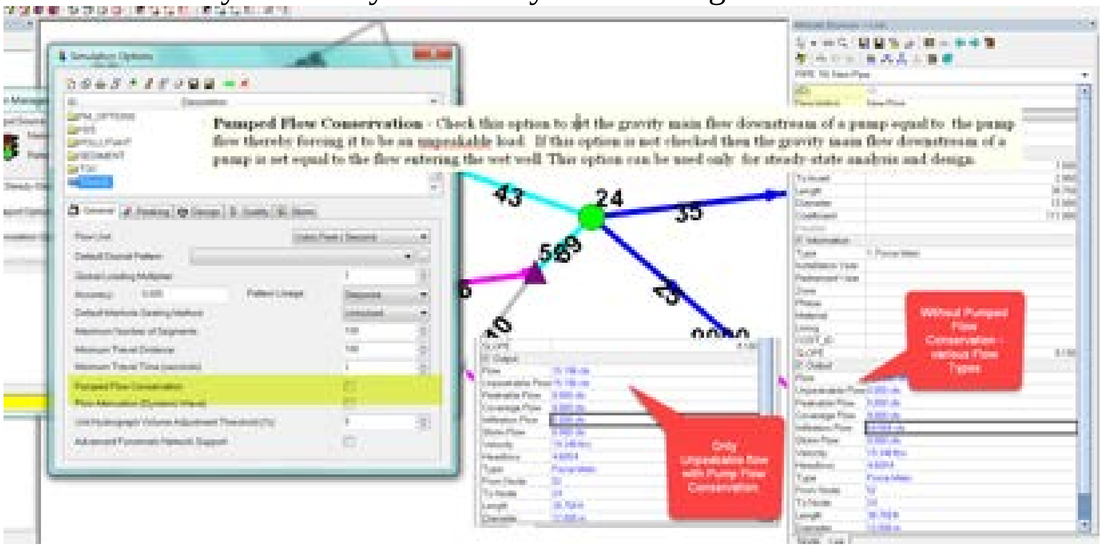
- **Default Manhole Sealing Method -** Use this to specify the default Manhole cover type for Normal Manholes. Choose *Locked* to contain flow inside the manhole structure when the water level rises above the rim elevation and choose *Unlocked* if flow is not contained inside the structure and spills over when the water level rises above the rim elevation. The maximum head possible in the second case is thus the manhole rim elevation

- **Maximum number of segments -** The maximum number of segments specifies the maximum number of segments a pipe can be divided into during a flow routing and quality analyses. The default value is 100 segments per pipe. The maximum number of segments affects the speed and accuracy of the hydraulic analysis. The smaller the value the faster the computational speed.

- **Minimum Travel Distance -** Minimum Travel Distance is the smallest length that a pipe segment could assume. Pipes shorter than this value will have only one segment. The default value is 100 ft.

- **Minimum Travel Time -** Minimum Travel Time establishes the smallest time of travel through a pipe recognized by InfoSewer Travel times smaller than this number are set equal to it (travel times through pumps are instantaneous and are not affected by this limit). The default minimum travel time is 1 second.InfoSewer determines the travel time based on pipe and flow properties, and uses this computed time step for hydraulic calculations during an EPS. The user can assign a desired minimum travel time, and this value is used when calculated travel times are less than the minimum travel time. The maximum allowable value for the minimum travel time is equal to the smaller value of report time step and pattern time step. The minimum travel time affects the speed and accuracy of the water quality analysis (i.e. time of concentration and source tracing). The larger this value is the faster the computational speed.

- **Pumped Flow Conservation -** Check this option to set the gravity main flow downstream of a pump equal to the pump flow thereby forcing it to be an unpeakable load. If this option is not checked then the gravity main flow downstream of a pump is set equal to the flow entering the wet well. This option can be used only for steady-state analysis and design.



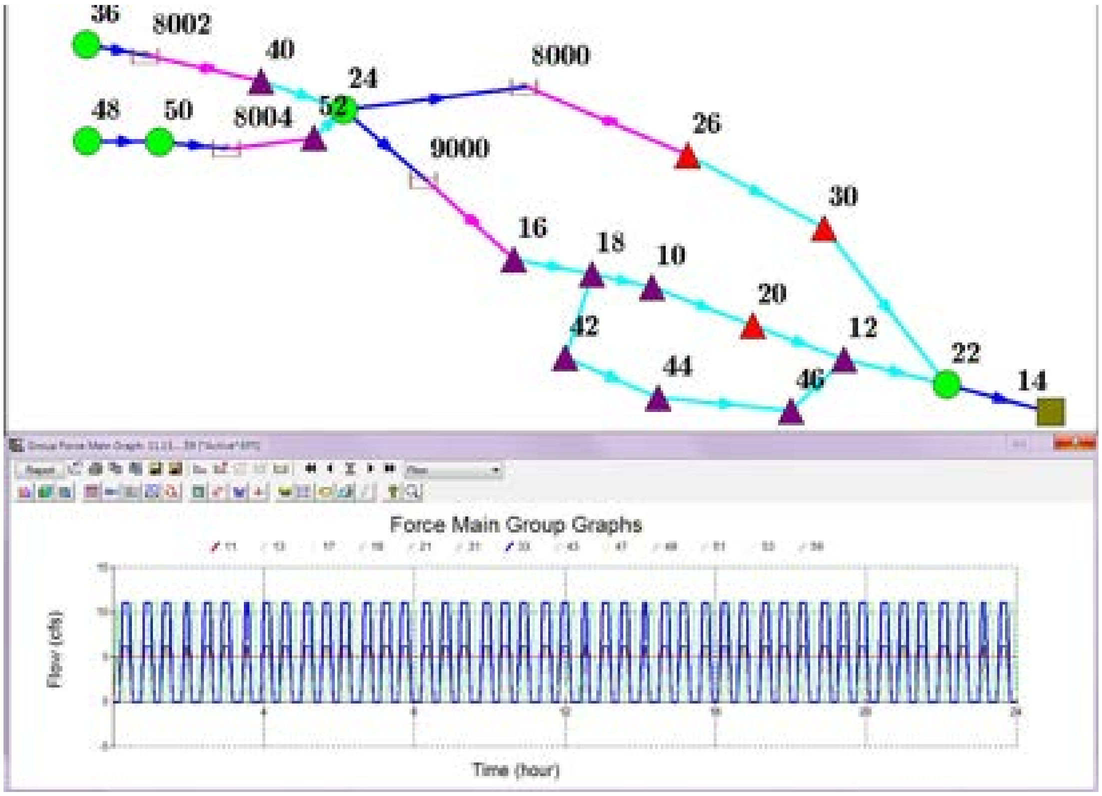
- **Flow Attenuation -** Check this option to include flow/hydrograph attenuation in the EPS simulation. If this option is not checked then flow attenuation will not be considered in the analysis. Flow attenuation in a sewer collection system is the process of reducing the peak flow rate by redistributing the same volume of flow over a longer period of time as a result of friction (resistance), internal storage and diffusion along the sewer pipes.InfoSewer uses a distributed Muskingum-Cunge flow routing method based on diffusion analogy, which is capable of accurately predicting hydrograph attenuation or peak flow damping effects (peak subsidence). This option can only be used for extended period simulation.

- **Unit Hydrograph Adjustment Threshold(%) -** The option provides the flexibility whether to adjust ordinates of synthetic unit hydrograph such as Tri-triangular, NRCS Dimensionless, NRCS Triangular and CUHP. Theoretically, volume under a UH has to be drainage area * an inch of rainfall. That may not happen while synthesizing UHs. In case the difference in volume under the synthesized UH and the theoretical volume exceeds the threshold assigned by the user, the model will adjust the UH volume to "match" the theoretical volume.

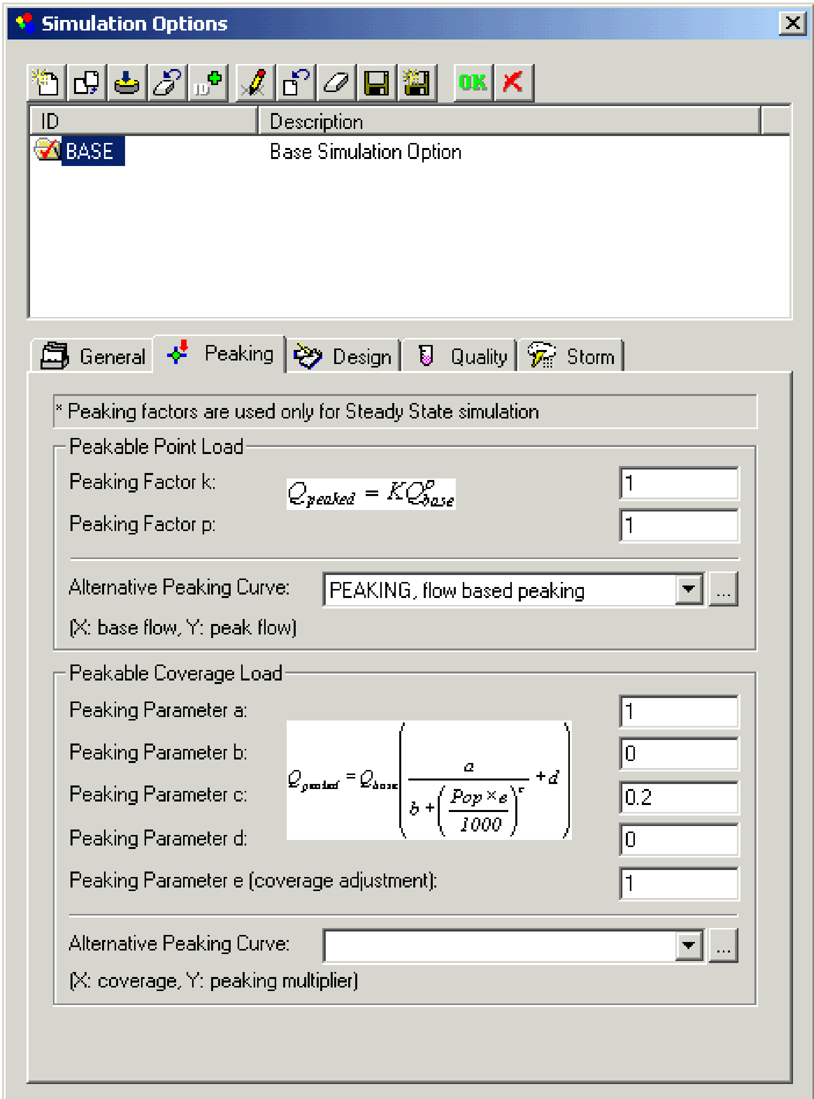
- **Advanced Forcemain Network Support -** The checkbox for Advanced Forcemain Network Support allows the simulation of two or more upstream force mains to one or more downstream force main(s) through a junction chamber. The link flows and node depths are solved iteratively to maintain the mass balance and the energy balance of the incoming and outgoing flows. The new network solver is used to calculate the flows in the forcemain pipes and the heads at the junction chambers associated with the force main(s). See Advanced Force Main Network Solution for additional information.

- If the checkbox flag is turned **on** (checked) then the new **Advanced Forcemain Network Support** will be used during the simulation.

- If the checkbox flag is turned **off** (unchecked) then the default force main solution will be used. You will get an error message if there is more than one force main connected to the same junction chamber.
  In the example shown below, the upstream force mains are links 947 and FM51, the merging junction chamber is node 52 and the downstream force main is link 53. As you can see in the graph from the Output Report Manager the flow in link 53 is the sum of the flows in links 947 and FM51. This example will only work with the new Advanced Forcemain Network Support feature.



[The Peaking Tab]()



InfoSewer provides two variations for analyzing and peaking sewer flows. The first method is to peak the base flow while the second method consists of peaking a coverage flow that is based on population. Steady state simulation flows will be peaked depending on which peaking type is specified for the manhole loading. See the Attribute Browser to learn more.

For Peakable Base Flow

- **Peaking Factor K -** The K value used in a steady state base flow peaking equation.
- **Peaking Factor p -** The p (rho) value used in a steady state base flow peaking equation.

$$Q_{peaked} = KQ_{base}^{p}$$

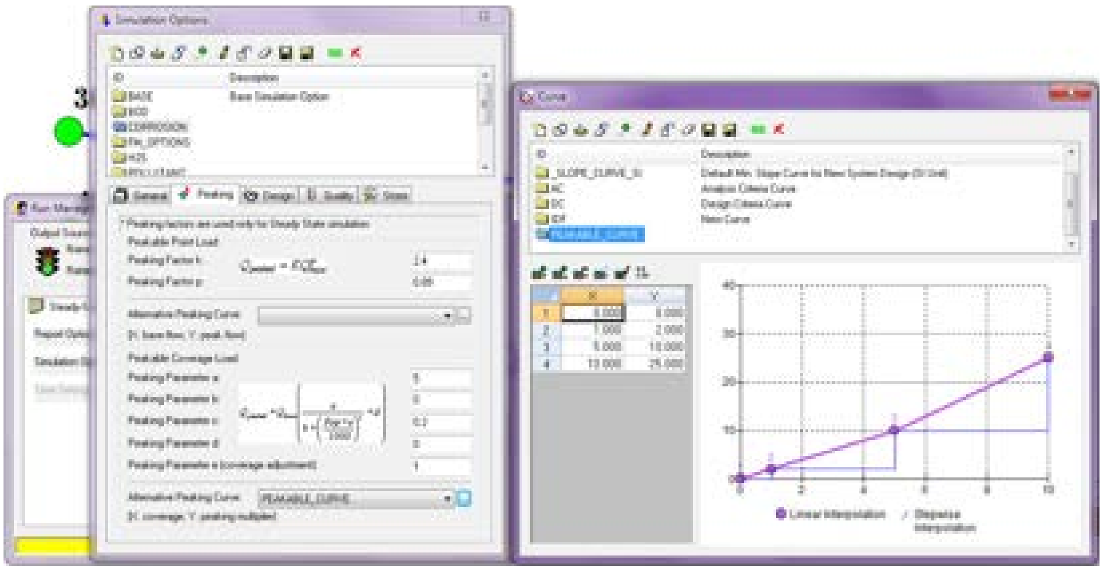
- **Alternative Peaking Curve -** The Peaking Curve method is an alternative method to compute flow data (loads) for peak conditions. Any peaking curve can be specified to estimate base flow peaks and model peak flow - base flow relationships. The peaking curve represents the peak flows (Y-axis) as a function of base flows (X-axis). Click here to learn more about setting up a curve

For Peakable Coverage Flow

- **Peaking Parameter a -** The a value used in a steady state coverage flow peaking equation.
- **Peaking Parameter b -** The b value used in a steady state coverage flow peaking equation.
- **Peaking Parameter c -** The c value used in a steady state coverage flow peaking equation.
- **Peaking Parameter d -** The d value used in a steady state coverage flow peaking equation.
- **Peaking parameter e -** The e value used in the coverage flow peaking equation indicated below.

$$Q_{peaked} = Q_{base}\left(\frac{a}{b+\left(\frac{Population \times e}{1000}\right)^{c}} + d\right)$$

- **Alternative Peaking Curve -** The Peaking Curve method is an alternative method to compute flow (loads) for peak conditions. For peakable coverage flows, the peaking curve represents the peaking multiplier (Y-axis) as a function of Population number or coverage (X-axis).



[The Design Tab]

- **New System -** designs a new sewer collection system (i.e., determines size and slope of pipes) given manhole locations, sewer loads, ground elevation at manholes, and various design criteria including capacity (i.e., depth to diameter ratio), velocity, cover depth, and slope.
- **Existing System -** conducts capacity analysis on an existing system and designs, either a replacement pipe or a duplication pipe, for conduits that fail to meet one or more design criteria including depth to diameter ratio and velocity.
- **Velocity Constraints -** Sewer pipes are designed to maintain a velocity within the minimum velocity and the maximum velocity values specified here. The minimum velocity should be assigned from the perspective of avoiding sedimentation problems whereas the maximum velocity should be set in such a way that erosion of pipe materials is avoided. It should be noted that the velocities defined here are applied to all pipes in the system for which maximum and minimum velocities are not defined individually. The velocities should be given in ft/sec for US customary units and in meters/sec for SI units. The default values are 10ft/sec (3m/sec) for maximum velocity, and 2 ft/sec (0.6 m/sec) for minimum velocity.
  - **Maximum Velocity -** The global maximum velocity criteria to be used during a steady state design simulation. Maximum velocity criteria can also be assigned to an individual pipe through the pipe constraints button under the Attribute Browser.
  - **Minimum Velocity -** The global minimum velocity criteria to be used during a steady state design simulation. Minimum velocity criteria can also be

assigned to an individual pipe through the pipe constraints button under the Attribute Browser.

- **Cover Constraints -** These constraints limit the depth, measured from the ground elevation, at which the sewer pipes are laid. These values should be defined considering local conditions such as anticipated loading from traffic and buildings, and to prevent freezing in cold regions. Like velocity constraints, the limiting cover depth values assigned here are applied to all pipes in the system for which cover constraints are not assigned. The units are feet and meters for US customary and SI systems, respectively. The default values are 20 ft (6 m) and 3ft (1m) for maximum cover depth and minimum cover depth, respectively.
    - **Maximum Cover -** The global maximum cover criteria to be used during a steady state design simulation. Maximum cover criteria can also be assigned to an individual pipe through the pipe constraints button under the Attribute Browser.
    - **Minimum Cover -** The global minimum cover criteria to be used during a steady state design simulation. Minimum cover criteria can also be assigned to an individual pipe through the pipe constraints button under the Attribute Browser.
- **Max. Crown Drop at Manholes -** Under normal design condition, crown elevation of a pipe leaving a manhole is set to the minimum crown elevation of pipe(s) entering the manhole. However, pipe velocity resulting from a pipe aligned according to crown elevation matching may exceed the maximum velocity defined for the pipe. In such cases, a drop manhole is used to lower crown elevation of the pipe, thus the pipe slope and the flow velocity. Lowering pipe elevation also leads to lowering invert elevation of the manhole. As the difference in invert elevation of the incoming pipe(s) and invert elevation of the manhole widens, flow falling from the incoming pipes to the manhole may erode the manhole. Maximum crown drop at manhole refers to the maximum allowed difference in crown elevations of the upstream pipe(s) and the downstream pipe to avoid formation of large water falls and subsequent erosion of manhole materials. However, damage cased by water falls coming from small pipes may not be significant. Therefore, it may not be necessary to limit crown drop for small pipes. The **Cutoff Diam.** refers to pipe diameter beyond (inclusive) which the maximum crown drop constraint has to be considered. Units used for max. crown drop at manhole and cutoff diameter are ft and inch in US customary, and, meter and mm in SI systems. The default values are 3ft and 24 inch for crown drop at manhole and cutoff diameter, respectively, in US customary units, and 1 m and 600 mm for crown drop at manhole and cutoff diameter, respectively, in SI units.

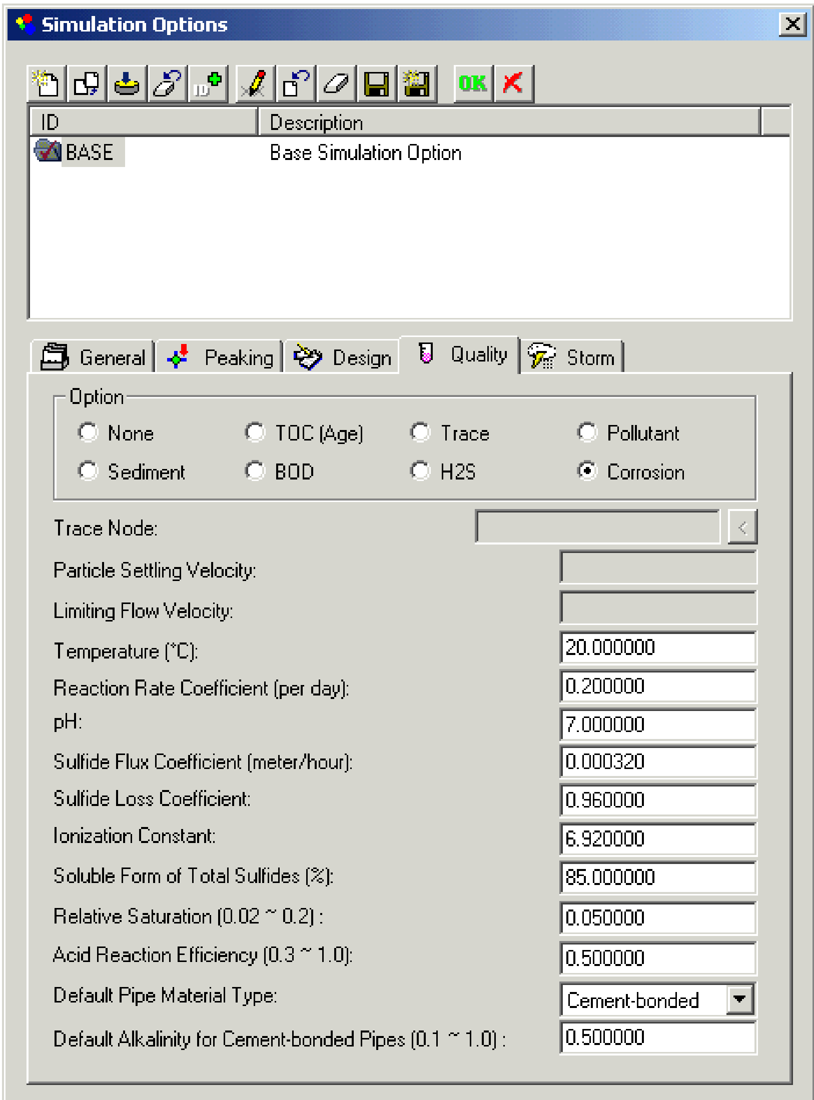
- **Manhole Invert Offset -** refers to the required offset between the minimum invert elevation of the incoming and the outgoing pipe and invert elevation of the manhole. Units are inch and mm in US customary and SI systems, respectively. The default value is zero.
- **Design Manning Coefficient -** Refers to Manning's n for the new sewer pipe. The pipe Manning coefficient that is to be applied to parallel and replacement pipes during a steady state design simulation. The default value is 0.001.
- **Default D vs. Min Slope Design Curve:** The minimum slope allowed for various conduit sizes is supplied to the model using this curve. X-axis values are pipe diameters and Y-axis values are minimum slope. The curve defined here will be applied to all pipes in the collection system for which D vs. Min Slope Design Curve is not locally defined. A default curve is available for both US customary units and SI units.
- **Default D vs. d/D Analysis Criteria Curve -** The curve used when evaluating pipes during a steady state analysis. When d/D values for specific diameters exceed these values, the pipe is deemed deficient and a parallel and replacement pipe are recommended in a steady state design simulation.
- **Default D vs. d/D Design Curve:** Defines maximum allowed carrying capacity for a pipe in terms of depth to diameter ratio for various pipe sizes. X-axis values are pipe diameters and Y-axis values are depth to diameter ratio values.
- **Include Existing Pipe in Parallel Design -** If this option is checked, then the existing pipe d/D ratio constraint must be satisfied when a new parallel pipe size is determined. This option is only used for steady-state design.
- **Design Only Coded Inverts (Invert <= -9999) -** This option is provided to allow fixing invert elevation of a newly designed pipe to predefined value, if needed. If this option is checked, pipes that are assigned invert elevation greater than -9999 will have their invert elevations fixed at those values. New invert elevation would be designed only for pipes whose existing invert elevation is less than or equal to -9999.
- **Default Pipe Replacement Cost Cu Irve:** Used to define pipe cost per unit length of a pipe (i.e., per unit feet for US Customary, and per unit meter for SI units). X-axis values are pipe diameters and Y-axis values are cost per linear length of a pipe. The cost could include pipe cost and many other cost types. Click here to learn more.
- **Include Excavation/Reinstatement Cost:** Some cost types such as excavation cost and reinstatement cost may be readily defined in terms of unit volumes, not unit length. For such cases, the design model estimates incurred costs based on average cover depth for the pipe, trench width computed based on pipe diameter,

and pipe length. This cost provision option is provided to offer the flexibility for modelers who wish to define cost in terms of unit volume, or, for those who want to specify some cost types in unit lengths using the replacement cost curve option previously described, and other cost types in terms of unit volume of a pipe using this option.

- To activate this cost provision option, click on the “Define Cost” button and complete input data on the initiated dialog editor.
- **Default Parallel Pipe Cost Curve -** The default diameter vs. cost curve that is used for deficient pipes where a parallel pipe is recommended. Click here to learn more.

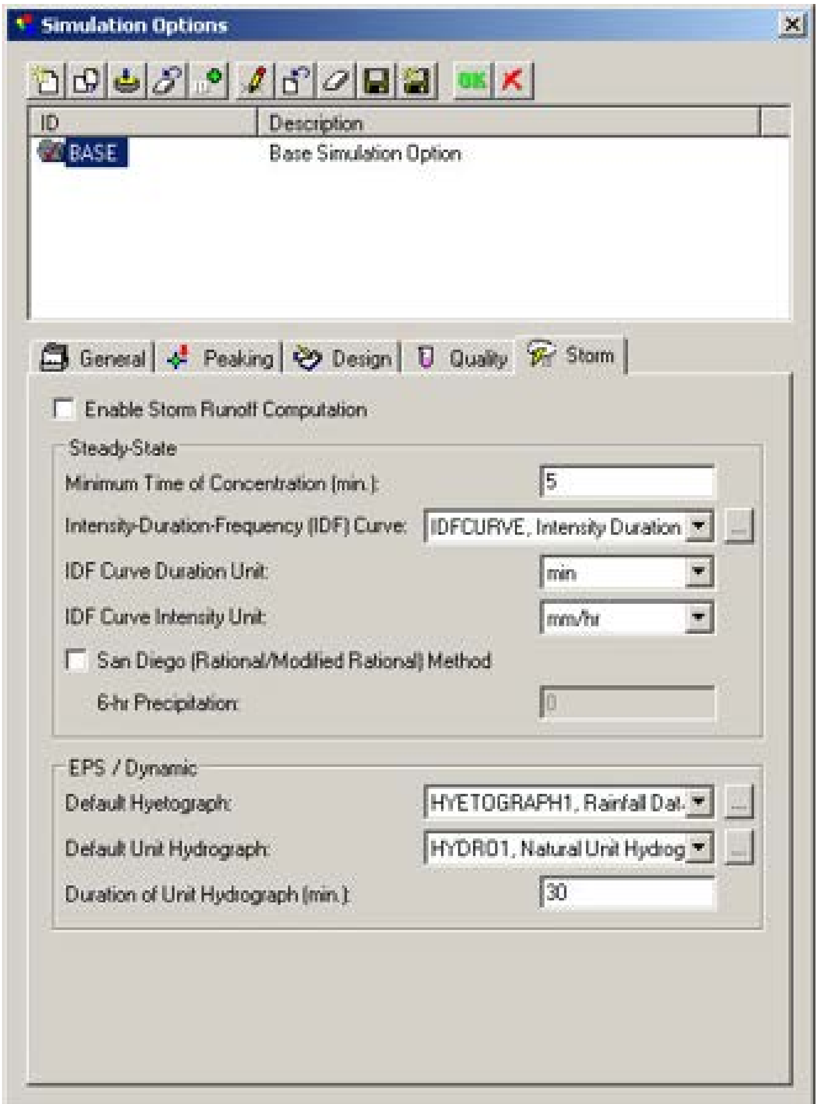
For information on the report generated by this analysis, click here.

The Quality Tab



InfoSewer provides seven options for sewage quality analysis in conjunction with a hydraulic simulation. These options are available only for EPS runs, and may be chosen on the Quality panel from the Simulation Options dialog box given above. Please click here for details on how to use the quality module of InfoSewer.

The Storm Tab



InfoSewer provides comprehensive stormwater modeling capability. This feature is only available for InfoSewer. Click here to learn more about the storm tab.

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Manholes

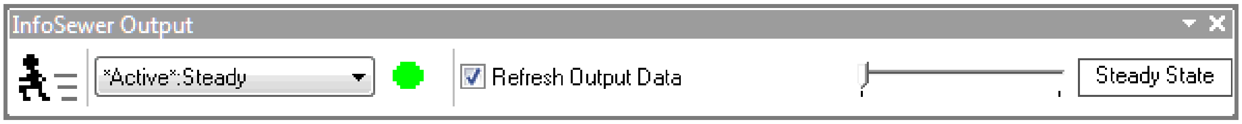
Manholes are junctions placed at the intersection of two or more links. Three types of manholes exist in a sewer collection system.

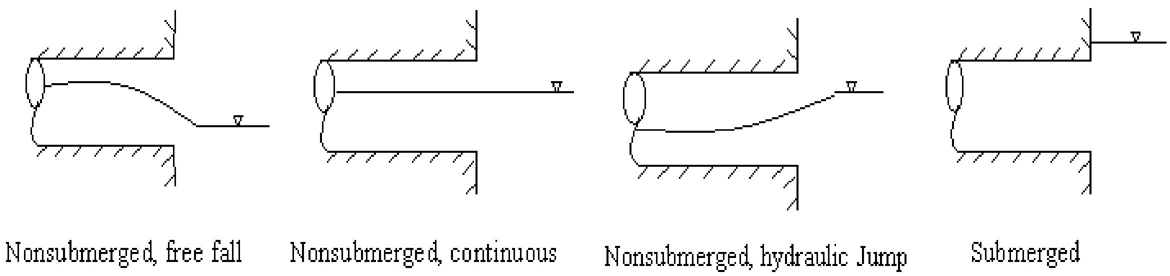
1. **Normal:** Where the physical location of a manhole exists in the collection system or a physical break is needed for a change in pipe attributes (e.g. diameter, material).
2. **Chamber:** A quasi-node placed to represent a break between a pump and a force main. No loads can be defined for a chamber.
3. **Outlet:** The facility where sewage flows exit the collection system. These nodes define the discharge end or the most downstream element of a sewer network. The exit condition can be grouped into four cases as shown below.

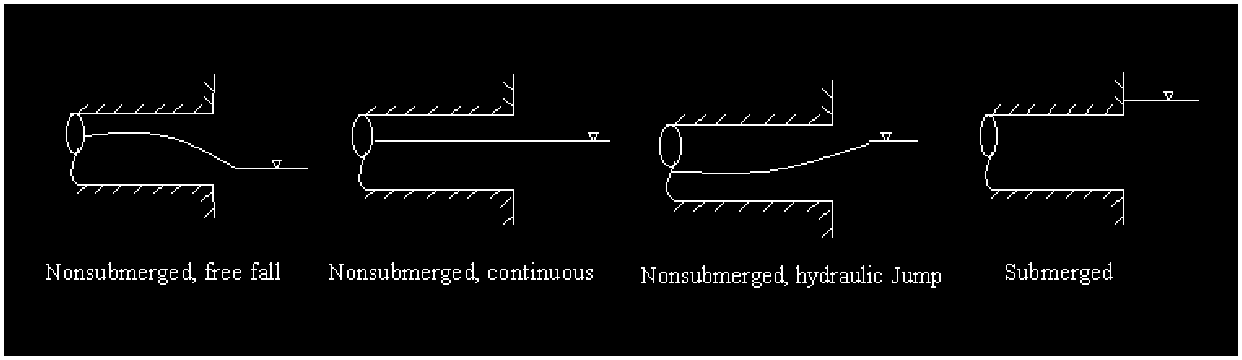
Nonsubmerged, free fall

Nonsubmerged, continuous

Nonsubmerged, hydraulic jump, Submerged







Manholes must also have their elevation above datum (e.g., sea level) specified in order for HGL profiles to be generated. Click here to view the fields required for a manhole.

How do I...

- **Create a manhole?**
- **Edit a manhole?**
- **Edit a group of manholes?**
- **Delete a manhole?**
- **Recall a deleted manhole?**

Create a Manhole

To create a manhole, click on the create node icon or from the **Create** menu, click **Add/Insert Manhole.**

Once the command is initiated, a crosshair cursor will appear. Select the location of the new node by left mousing clicking anywhere on the map display.

With the manhole placed, a dialog box will appear requesting the ID for the manhole, a description and the type of manhole being placed (Normal, Chamber, or Outlet). Once an ID, description and type are specified, press OK to end the command.

(Note: You can also use the **Digitize Network** command from the **Create** menu to create a pipe and node network at the same time. Once digitized, the user is able to add relevant modeling data in the Attribute Browser dialog box.)

Edit a Manhole

Graphic Edit - To edit the manhole graphically, use the Move Node icon to select the node. Once the node is selected, left mouse click on top of the node and hold the button and drag the mouse at the same time. A red rubber band will appear showing the user where the node will be located. Once the node is moved to its new location, release the mouse button. The node is now relocated. (Note: If the user has not enabled "Auto Length Calculation" under the **Tools** -> **Preferences** menu, all pipes connected to that node will not have their lengths automatically changed.)

Data Edit - To edit the data related to a node, first select the node using the Select icon. Once selected, the user is able to edit the database data related to the node under the Node tab of the Attribute Browser dialog box.

Edit the Data for a Group of Manholes

Group edits can be made by either creating a domain first and then selecting the Edit Domain Attributes icon, or the user can use the Edit Selection Attributes icon. Both icons are found on the Edit Network toolbar or can also be found under the **Edit** Menu.

Once the Edit Selection Attributes command is initialized, the user makes a selection and presses the <Enter> key. At this point, the Edit Attribute dialog box is displayed allowing the user to edit any field for any of the 4 data elements that may have been selected.

Delete a Manhole

To delete a manhole, from the **Edit** menu, select **Delete Node** or select the Delete Node icon. H2OMAP Sewer will prompt the user to confirm deletion. (Note: All links attached to the node being deleted will also be deleted.) The node and associated links are deleted graphically but can still be recalled from the database using the recall command. However, if Auto Database

Packing is enabled (Tools -> Preferences), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Manhole

To recall a deleted node, go to **Utilities** menu, point to **Recall**, then select **Node**. In order to perform the recall, the user must know the ID of the node that was deleted. If the ID is unknown, select the Show Deleted Nodes command from the Recall menu to see a list of ID's that have been deleted.

Other Related Topics - Data Elements, Manhole, Pipes, Pumps, Wet Well

Wet Wells

Wet wells are used by InfoSewer to model the storage of sewage water within a lift station. The capacity of a wet well is only evaluated during an EPS/Dynamic model run. The only facility that can leave a wet well is a pump.

How do I...

- Create a wet well?
- Edit a wet well?
- Delete a wet well?
- Types of Wet Wells
- Recall a deleted wet well?

Create a Wet Well

To create a wet well, click on the add wet well icon or from the **Create** menu, click **Add Wet Well.**

Once the command is initiated, a crosshair cursor will appear. Select the location of the new wet well by left mousing clicking anywhere on the map display.

Once the wet well has been located, a dialog box will become present requesting the ID number of the wet well and description for the wet well being created. Once these date are entered, press OK to end the command.

Types of Wet Wells

There are 2 different types of wet wells the user is able to input into InfoSewer.

Cylindrical -A cylindrical wet well has a constant diameter in relation to its high and low water levels. Via the Modeling option under the Attribute Browser, enter the following:

Bottom Elevation -The elevation (above sea level) of the bottom of the wet well, ft (m).
Initial Water Level - The height of water above the bottom of the wet well (ex. 12 ft.) when the simulation is to begin, ft (m).
Diameter - The diameter of the wet well, ft (m).
Maximum Level - The highest allowable water level (ex. 22 ft.), ft (m).
Minimum Level - The lowest allowable water level (ex. 3 ft.), ft (m)

Variable Area -A wet well where the volume changes with respect to height of sewage in the wet well. The only way to model a variable area wet well is to create a curve that represents the relationship between height and volume. Via the Modeling option under the Attribute Browser, enter the following:

Curve - The curve assigned to the wet well for an EPS. Click here to see an example curve.

There are 2 different types of wet wells the user is able to input into InfoSewer. Click on any wet well below to learn more.

0. *Cylindrical*

1. *Variable Area*

Edit a Wet Well

Graphic Edit - To edit the wet well graphically, use the Move Node icon to select the wet well. Once selected, left mouse click on top of the node and hold the button and drag the mouse at the same time. A red rubber band will appear showing the user where the node will be located. Once the node is moved to its new location, release the mouse button. The node is now relocated. (Note: If the user has not enabled "Auto Length Calculation" under the **Tools** -> **Preferences** menu, all pipes connected to that node will not have their lengths automatically changed.)

Data Edit - To edit the data related to a wet well, first select the node using the select tool. Once selected, the user is able to edit the database data related to the storage node under the Node tab of the Attribute Browser dialog box.

Delete a Wet Well

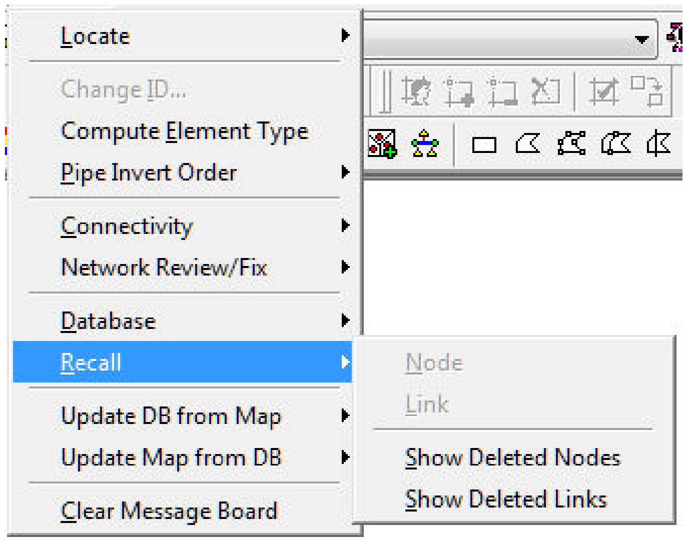
To delete a wet well, from the **Edit** menu, select **Delete Node** or select the Delete Node icon. InfoSewer will prompt the user to confirm deletion. (Note: All links attached to the node being deleted will also be deleted.) The node and associated links are deleted graphically but can still be recalled from the database using the recall command. However, if Auto Database Packing is enabled (Tools -> Preferences), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Wet Well

To recall a deleted wet well, go to **Utilities** menu, point to **Recall**, then select **Node**. In order to perform the recall, the user must know the ID of the node that was deleted. If the ID is unknown, select the Show Deleted Nodes command from the Recall menu to see a list of ID's that have been deleted.

The Recall command is used to restore/or undelete network components that have been deleted. This command will not work if the user has enabled Auto

Database Packing option (Project Preferences). Recall is found under Utilities / Recall. Recalled components will be redrawn with their graphical properties (node size, pipe connectivity and shape) at the time of deletion. All database values assigned to those components will be restored.



Other Related Topics - Data Elements, Manhole, Pipes, Pumps, Wet Well

InfoSewer™

Modeling Made Easy

Recall

The Recall command is used to restore – or undelete – network components that have been graphically deleted. This command will not work if the user has enabled Auto Database Packing from Tools -> Preferences or has recently packed the database. Recalled components will be redrawn with their graphical properties (node size, link connectivity and shape) at the time of deletion. All database values assigned to those components will be restored.

To run a Recall, from the **Utilities** menu, select **Recall.** There the user will see the following options:

Recall Options

Node - Recalls one or more previously deleted nodes. If the user does not know the ID of the deleted node, use the Show Deleted Nodes command to display a listing of deleted nodes.

Link - Recalls one or more previously deleted links. If the user does not know the ID of the deleted link, use the Show Deleted Links command to display a listing of deleted links.

Show Deleted Nodes - Used to get a listing of all nodes marked for deleting in the nodes database.

Show Deleted Links - Used to get a listing of all links marked for deleting in the links database.

InfoSewer™

Modeling Made Easy

Output Source

An output source is nothing more than an association between InfoSewer and the output (.out) file created from a simulation run. InfoSewer refers to model runs as output sources, whether they be *active*.Steady or ScenarioX.Design, every model run is affiliated to an external output file by its "output source" name.

From the icons at the top of the Run Manager, the user has full control over the associations between an output source in InfoSewer and its external output file.

For Example

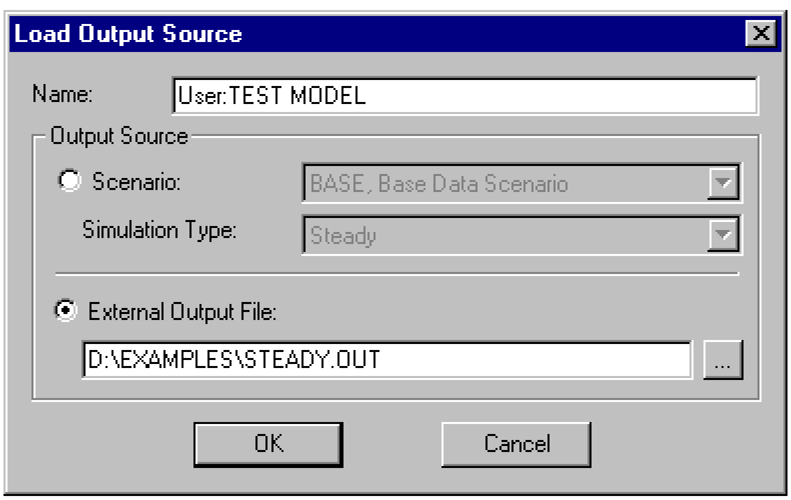
If the user has created and modeled four custom scenarios, the output for each scenario will be made available for report and graphing purposes under the Output Report Manager (under the All Output Sources box). If the user then wanted to add (load) another model run performed by someone else (perhaps multiple people working on the same project file), then by clicking on the Load Output Source icon, the user can assign an external output file to a user-defined (internal) output source. Once this is done, the newly imported output source can now be compared with other previously run "output source" models (via graphs and reports).

How do I...

- **Load (Create) an Output Source?**
- **Remove an Output Source(s)?**
- **Rename an Output Source?**
- **Save an Output File?**

Loading Output Sources

To load (create) an output source, from the **Tools** menu, select **Run Manager**. At the Run Manager dialog box, select the Load Output Source icon. From here, the user will see the following dialog box:



There are two types of output sources that the user can load (see below):

- Scenario - This option is only available if the user has previously removed an existing output source via the "Remove Output Source" icon. By selecting this option, the user can reinitialize a previously run model without having to re-run a scenario simulation.
- External Output File - This option is used to assign an existing .out file to a user specified output source "name" at the top of the dialog box. Once the link is established, the user can now compare existing simulation data with the newly imported data.

Remove Output Source(s)

To remove an output source from the current H2OMAP Sewer project, select either the Remove Output Source or the Remove All Output Sources icon. The first will remove the currently selected output source as shown in the upper left hand corner of the Run Manager while the latter will remove all output sources. Click here to view these icons.

(Note: Be advised that removing all output sources will not only remove all external output source linkages, but will also remove all internal scenario outputs.)

Rename an Output Source

To rename an output source, from the Run Manager, select the Output Source Name and then click on the Edit Name icon to rename the output source.

Saving an Output File

To save an output file, the user must first make active the scenario for which an output is desired. Once a scenario is active, go to the Run Manager and run the *active* model (steady state, design, or EPS). After the model has run, select the Save icon to save the model run to a file name other than the default.

(Note: Unless the user has disabled the "Enabled Output Save As" feature from the Tools -> Preferences dialog box, all model runs will be automatically saved to their default names under their respective scenario database folders.)

More on Output Sources

Each output source is identified by a unique name and is associated with a single output file. The most recently completed analyses are given the following names:

• ***Active*.Steady**	**Output source data (simulation results) for the most recently completed steady state analysis simulation.**
• *Active*.Design	Output source data (simulation results) for the most recently completed steady state design simulation.
• *Active*.EPS	Output source data (simulation results) for the most recently completed extended period simulation.

When assigning an output file to an output source, the data type need to match. In other words, a "steady" hydraulic run cannot be imported into an EPS simulation.

When you switch to another scenario, output sources for standard simulations are immediately available as described above.

As you run a given simulation, the output source name for that simulation is *Active*.Steady. The previous *Active* simulation is converted to a user-defined source and renamed based on the active scenario at the time that simulation was performed.

For example, if the previous standard simulation was based on your scenario named “CUSTOM1”, then the output source from the previous run is renamed CUSTOM1:Steady.

Once a scenario is run, the output source will always be tied to the default output file unless the user manually re-assigns that relationship.

InfoSewer™
Modeling Made Easy

Manholes

Manholes are points in the network where links join together and where loads enter the network. They are also placed at locations where pipe characteristics change (e.g., diameter and slope). The basic input data required for manholes is:

- The rim elevation , i.e. the top elevation of the manhole structure
- The diameter of the manhole structure
- The baseline load at the manhole

The manhole diameter and rim elevation parameters are used in the determination of manhole surcharge and flooding. The loads on the system, i.e., wastewater flows collected, accumulated and conveyed by the collection system are assumed to be positive. Manholes can also have their load vary with time and different types of loading (e.g., low density residential, medium density residential, high density residential, industrial, commercial, etc.) can be assigned to them. Flows from Infiltration are also modeled as manhole loads. Up to ten different loading types may be defined for each manhole (See LOAD and PATTERN sections). A manhole can also have no loading defined.

The results computed for manholes for a simulation are:

- The base load
- The total load
- The overload storage
- The hydraulic grade

Two other types of manhole can also be modeled:

- *Junction chambers*
- *Outlets*

Flow splits (bifurcations) can also be modeled and represent points in the conveyance system where multiple pipes separate flows to different parts of the system. A flow split occurs whenever two or more pipes exit the same manhole. InfoSewer provides four practical methods of allocating flows to the downstream (outgoing) pipes:

- Fixed flow split percentage method - The user specifies the percentage of the total flow in the manhole which is assigned to each of the downstream pipes. Note that InfoSewer checks that the sum of the fractions leaving a manhole equals 100 percent. In case, the sum is less or greater than 100%, InfoSewer will automatically adjust the flow allocations for the outgoing pipes proportionally to satisfy flow balance.
- Variable flow split percentage method - The user defines a curve representing the percentage of the total flow in the manhole which is assigned to a downstream pipe as a function of the total incoming flow (See CURVE section).
- Inflow-outflow flow split method - The user supplies an inflow-outflow curve for the downstream pipes splitting from a manhole. The curve defines the amount of the incoming flow to be diverted to a downstream pipe. This flow split method is designed to accommodate flow conditions where hydraulic structures such as weirs and dams are used to regulate downstream flows.
- Automatic flow split method - InfoSewer automatically computes the fraction of the total flow in the manhole assigned to each of the downstream (outgoing) pipes based on their invert levels and diameters.

Note: H2OMAP Sewer assumes that a pipe is closed if its flow split percentage is zero.

HEADLOSS AT MANHOLES

Manhole structures can induce backwater effects to their connecting sewer pipes. However, the precise hydraulic description of the flow in manhole structures is complicated because of the complex degree of mixing, separation, turbulence, and energy losses. These losses are commonly estimated as a function of headloss coefficients and are used to calculate hydraulic grade lines for upstream pipes during backwater analysis.

The headloss at a sewer manhole is determined based on the exit pipe's velocity from the following equation:

$$H_s = K \frac{V_{exit}^2}{2g} \quad (1)$$

where

- Hs = manhole headloss, ft (m)
- Vexit = exit pipe velocity, ft/s (m/s)
- g = gravitational acceleration, 32.184 ft/s2 (9.806 m/s2)
- K = headloss coefficient (unitless).

The headloss coefficient, used for estimating headloss through a manhole, depends on the type of manhole and typically ranges from 0.5 to 1.0.

InfoSewer™

Modeling Made Easy

Element Attribute Browser

The Attribute Browser dialog box allows the user to view and edit the database information related to a selected link or node facility. Shown below, the dialog box contains all information related to links and nodes when selected by the user.

Unless "grayed" out, any of the records under the section headers (Geometry, Modeling, Information) are editable at any time. While using the Attribute Browser, click the red check box next to the title to expand and contract the section headers. The Output data section is available only after a successful model run. Additionally other features allow you to locate (search) elements, create default values and assign specific values through the Tools menu.

To learn more about the multitude of icons related to the Attribute Browser, click here or select any of the icons below.

The Link Tab

The link tab allows the user to see the database information related to a graphically selected link.

Click on any of the drop down links below to learn more about the database information contained in the Attribute Browser.

Pipe Data

Pipe ID

- **ID** - The unique H2OMAP Sewer identification of the pipe.
- **Description** - A user defined description of the pipe.

Geometry

- **Start Node** - The upstream node for the pipe.
- **End Node** - The downstream node for the pipe.

Modeling

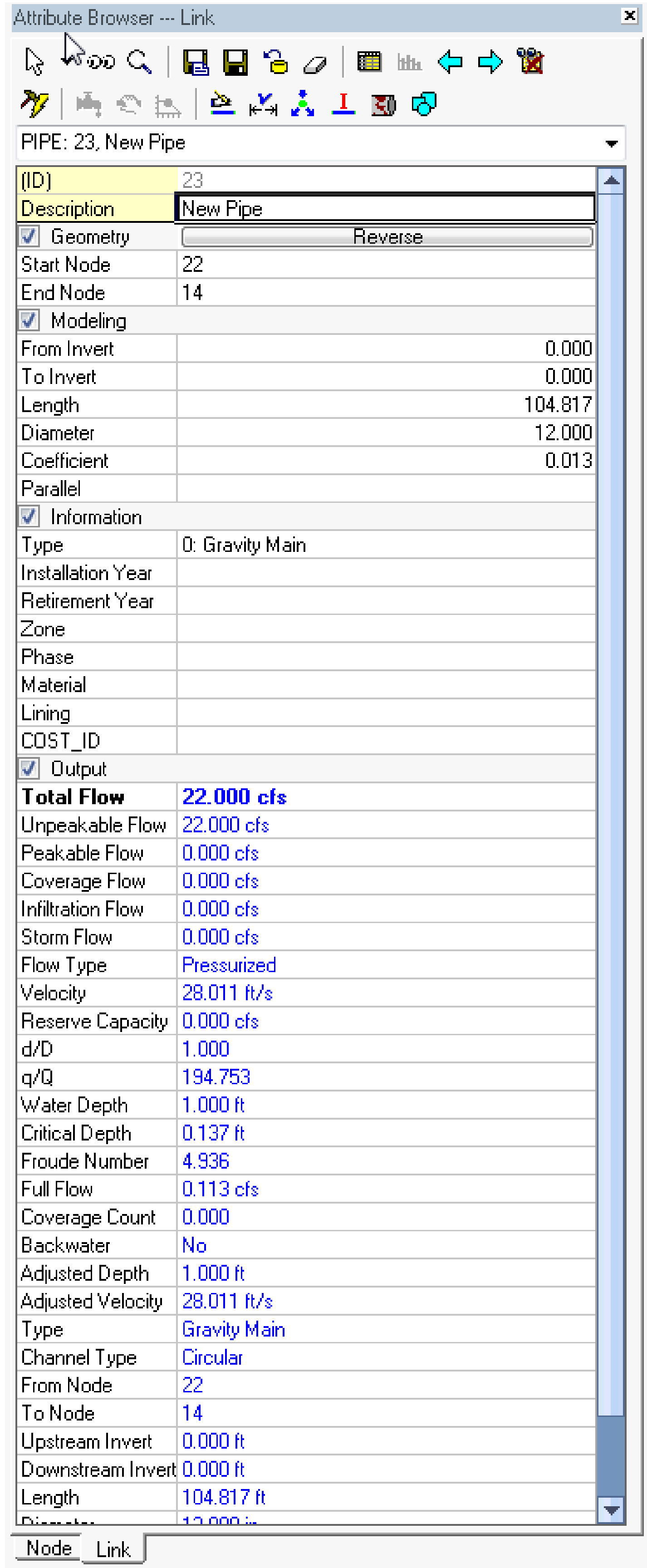
- **From Invert** - The upstream invert elevation of the pipe (meters or feet)
- **To Invert** - The downstream invert elevation of the pipe. (meters or feet)
- **Length** - The length of the pipe. Created automatically if Tools -> Preferences -> Operation -> Auto Length Calculation is

checked on. (meters or feet)

- **Diameter** - The diameter of the channel (for circular pipes) as specified by the user. For non-circular conduits, required input parameters may be one or more of channel depth, channel width, and side slopes. (inches or millimeters)

- **Coefficient** - The Manning (gravity flow) or Hazen-Williams (force main) roughness coefficient as assigned by the user.

- **Parallel** - The number of pipes in parallel with the existing pipe. Each parallel pipe is given the exact same parameters as the existing pipe.



[Output](#)

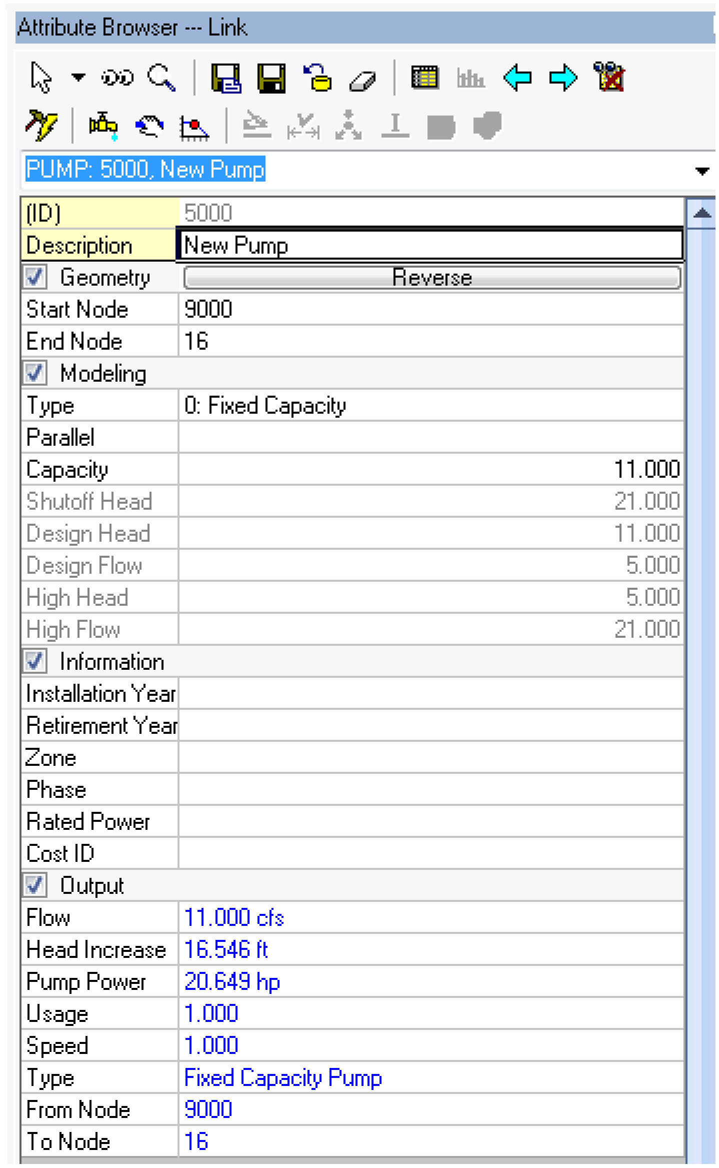
- **Output** - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a pipe output report.

Pump Data

Pump ID

- **ID** - The unique H2OMAP Sewer identification of the pump.

Description - A user defined description of the pump.



Geometry

- **Start Node** - The upstream node for the pump.
- **End Node** - The downstream node for the pump.

Modeling

- **Type** - The type of pump to be used by H2OMAP Sewer. To change the type, merely click in the drop down box and select from the list. Click here to learn more about each type and the fields required.

- **Parallel** - The number of pumps in parallel to the existing pump. Parallel pumps are given the exact specifications of the existing pump.

Click here to learn more about the types of pumps available in H2OMAP Sewer.

Information

- **Installation Year** - The year the pump was installed.
- **Retirement Year** - The year the pump is expected to be retired from service.
- **Zone** - The service area of the selected pump.
- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to modeling a simulation.
- **Rated Power** - The rated horsepower of

the pump in the field.

- **Cost ID** - A database field used to assign differing cost identifiers to differing facilities.

- **(Other User Created Fields)** - Any fields added by the user will appear at the end of the Information tab. Click here to learn how to add a field.

Output

- **Output** - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a pump output report.

The Node Tab

The node tab allows the user to see the database information related to a graphically selected node.

Manhole Data (Normal, Chamber, Outlet)

Node ID

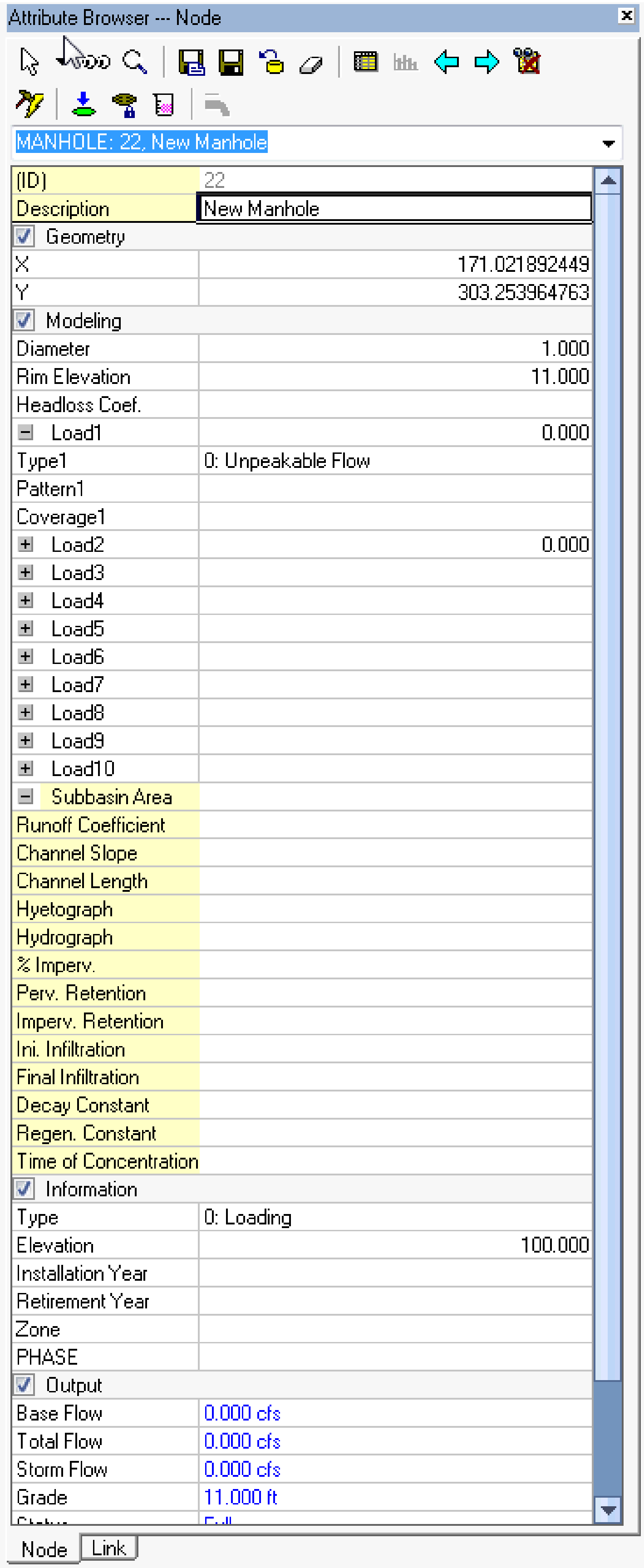
- **ID** - The unique H2OMAP Sewer identification of the manhole.
- **Description** - A user defined description of the node.
- *Geometry*
- **X** - The x-coordinate of the node.
- **Y** - The y-coordinate of the node.

Modeling

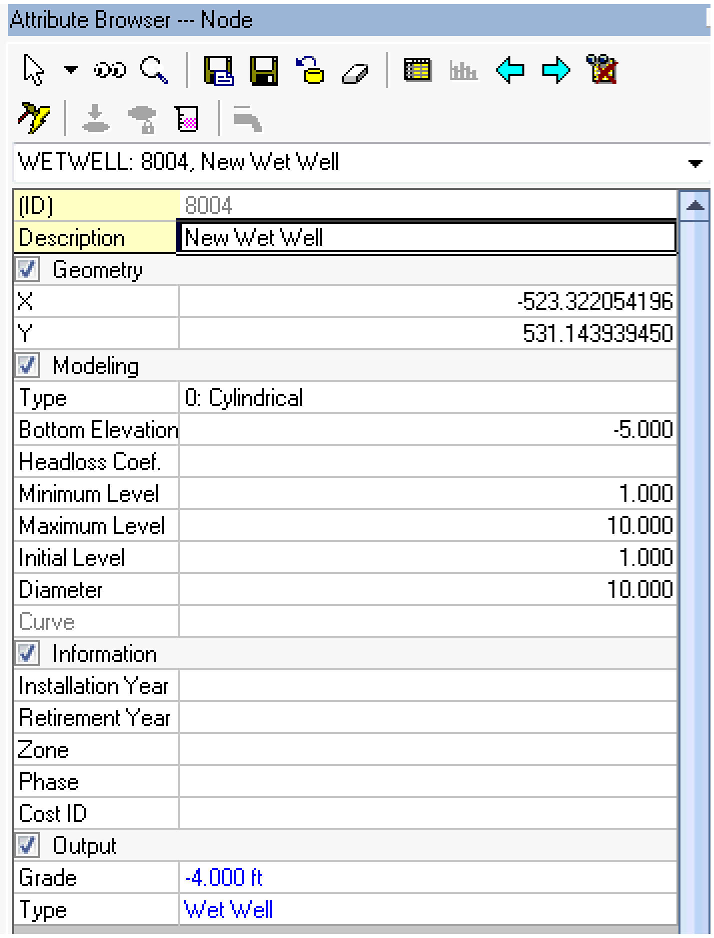
- **Diameter** - The internal diameter of the manhole. (feet or meters)
- **Rim Elevation** - The elevation of the rim of the manhole as specified by the user (usually a value denoted above sea level) (meters or feet)
- **Headloss Coef** – Used to calculate headloss in the manhole itself using

the exit pipe velocity (see link for more details on Headloss Coefficients and manhole headloss)

- **Load1** - The base sewage loading demand at the manhole. Up to ten base loadings can be entered for a single manhole. The extra load option allows additional loadings to be added as needed.
- **Type1** - Specify whether the load is an unpeakable flow, an peakable base flow or a peakable coverage flow. Click here to learn more about each type.
- **Pattern1** - The user defined diurnal pattern for the manhole loading. Required for an EPS analysis.
- **Coverage1** - The base population to be used during a peakable coverage flow determination.



- **Subbasin Area** - Area of the subwatershed that drains to the loading manhole. This variable is required for stormwater modeling.
- Runoff Coefficient - Ratio of storm runoff to rainfall volume. This variable is required during steady state simulation of stormwater.
- **Channel Slope** - Average slope of the longest flow channel for the subwatershed that drains to the loading manhole. This variable is required during steady state simulation of stormwater.
- **Channel Length** - Length of the longest flow path for the subbasin that drains to the manhole. This variable is required during steady state



simulation of stormwater.

- **Hyetograph** - A plot of rainfall intensity versus time. This variable is required during dynamic simulation of stormwater.
- **Hydrograph** - Refers to a unit hydrograph pattern, and is required is required during dynamic simulation of stormwater.
- **% Impervious** : Percentage of the subwatershed that is covered with impervious land uses. This parameter is used to estimate depression storage losses.
- **Pervious Retention** : Refers to depression storage of the pervious portion of the subwatershed. The value should be given in feet for US Customary units,

and in mm for SI units.

- **Impervious Retention** : Refers to depression storage of the impervious portion of the subwatershed. The value should be given in feet for US Customary units, and in mm for SI units.
- **Initial Infiltration** : maximum or initial infiltration rate (at t = 0). The parameter is accepted in inches/hour for US Customary, and in mm/hour for SI units. The default value is value is 3 inches/hour (76.2 mm/hour).
- **Final Infiltration** : final infiltration rate. The value should be provided in inches/hour for US Customary, and in mm/hour for SI units. The default value is 0.5 inches/hour.

- **Decay Constant** : decay coefficient for Horton's infiltration equation, in per second. The default value is 0.001/second.
- **Regeneration Constant** : Decay coefficient for the exponential recovery curve, in per second. It is generally considered to be less than decay constant, implying a longer drying curve than wetting curve.

- **Time of Concentration:** Time of concentration for the subcatchment that is just starting to contribute to the node.

Information

- Type - The type of manhole. Type 0 is a normal manhole, type 1 is a chamber manhole while type

2 is an outlet. Click here to learn more.

- **Elevation** - The ground elevation of the manhole. This value may differ than the rim elevation.
- **Installation Year** - The year the manhole was installed.
- **Retirement Year** - The year the manhole is expected to be retired from service.
- **Zone** - The service are of the selected manhole.
- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to modeling a simulation.
- **(Other User Created Fields)** - Any fields added by the user will appear at the end of the Information tab.

Click here to learn how to add a field.

Output

Output - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a manhole node output report.

Wet Well Data

Wet Well ID

- **ID** - The unique H2OMAP Sewer identification of the wet well.
- **Description** - A user defined description of the wet well.

Geometry

- **X** - The x-coordinate of the wet well.
- **Y** - The y-coordinate of the wet well.

Modeling

Type - The type of wet well to be used by H2OMAP Sewer. To change the type, merely click in the drop down box and select from the list. Click here to learn more about each type and the fields required.

Information

- **Installation Year** - The year the wet well was installed.
- **Retirement Year** - The year the wet well is expected to be retired from service.
- **Zone** - The service area zone of the selected wet well.
- **Phase** - Assigning a phase enables the user to activate and deactivate facilities through the facility manager prior to modeling a simulation.
- **Cost ID** - A database field used to assign differing cost identifiers to differing facilities.
- **(Other User Created Fields)** - Any fields added by the user will appear

at the end of the Information tab. Click here to learn how to add a field.

Output

Output - The output fields will display the output results for the latest model run (*active* output source). Click here to learn more about the fields provided in a wet well output report.

Other Related Topics - Tools menu

InfoSewer™

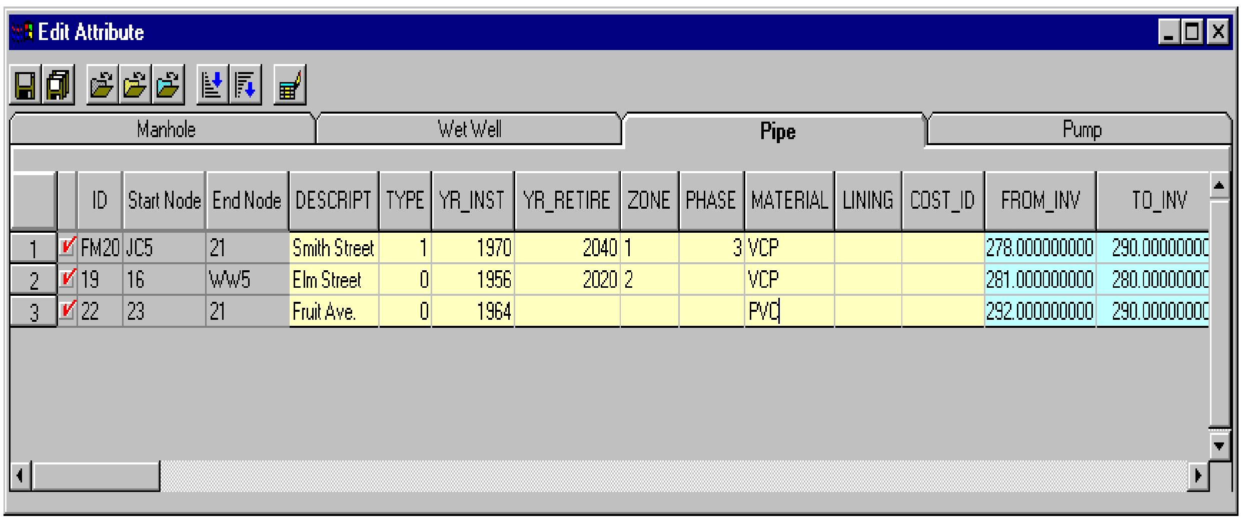
Modeling Made Easy

Edit Attribute

The Edit Attribute dialog box allows the user to edit any field from the selected (graphical or domain) InfoSewer data elements. This feature is helpful by allowing quick edits of database data to occur without having to use to DBEditor command.

To run the Edit Attribute command, from the **Edit** menu, select either the **Edit Selection Attributes** or **Edit Domain Attributes**. If you have created a domain, the dialog box below will immediately appear. If you wish to graphically select data elements, a "?" will appear next to the mouse cursor. Drag a selection window across the screen to select the data elements for editing in the Edit Attribute table.

Click on the icons below to learn more.



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Curves

Curves are objects that contain data pairs representing a relationship between two quantities. A curve is used by InfoSewer in comparing one value against another (ex. Volume vs. Depth). Curves are assigned to data elements through the Attribute Browser.

How do I...

- Create a Curve?
- Edit a Curve?
- Delete a Curve?

An InfoSewer model can utilize the following types of curves:

- Volume vs. Depth (Wet Wells - Variable-Area)
- Flow Split %
- Criteria Curves (Design and Analysis)
- Cost Curve (Replacement and Duplication)
- Peaking Curves (Flow based and Coverage Based)

Each curve is identified by a unique identifier within the project. It is advised that when defining curves, a detailed description is also provided to assist in curve selection.

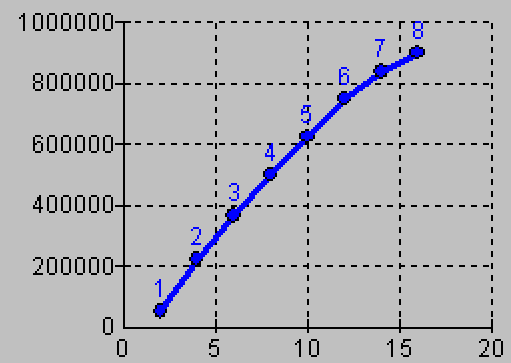
Types of Curves

InfoSewer utilizes the following types of curves:

Volume vs. Depth

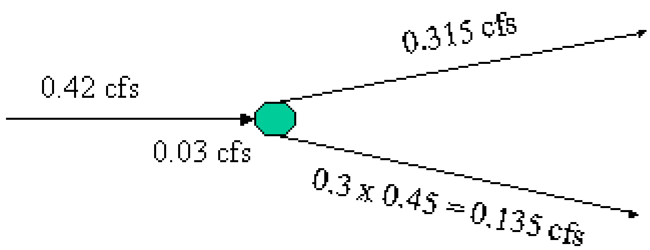
The curve for variable-area wet wells is defined as the volume of the well versus its corresponding depth. It is very important to note that the volume must be defined in cubic feet (ft3) or cubic meters (m3) and not in any other unit (such as gallons or MG). Depth must also be defined in the same units as the volume (ft or m). The volume of the wet well is provided on the (Y-axis) with the corresponding depth on the (X-axis). A sample table and curve are provided below:

	X	Y
1	2	50000
2	4	220000
3	6	365000
4	8	500000
5	10	625000
6	12	750000
7	14	840000
8	16	900000

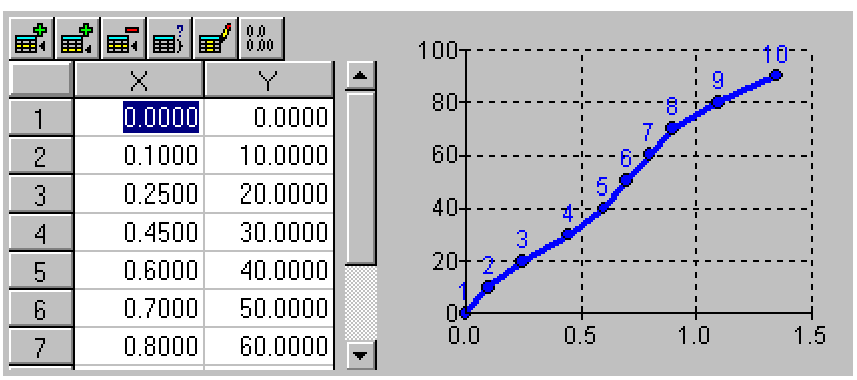


[Flow Split %]

In some instances, the user may wish to disable the automatic flow split computation provided by InfoSewer and create a curve that represents the amount of flow being diverted at a split manhole. The flow split curve is defined as the percentage or actual amount of downstream flows being routed to the selected downstream pipe (Y-axis) versus the total flow of all upstream sources (X-axis). For each increment of total flow (upstream flow + upstream manhole loading) a certain percentage/actual amount (depending on the flow split method) of flow will enter the selected flow split pipe. In other words, the y-axis could be a percentage (variable flow percentage option) or actual amount (inflow-outflow curve option) while the x-axis is defined by actual flow units. In the example below, when the total flow from the upstream pipe and the upstream manhole is 0.45 cfs, 30% of that flow will be routed to the selected pipe.



A sample flow split curve that illustrates the variable flow percentage option is provided below. [Click here] to learn more about flow splits.

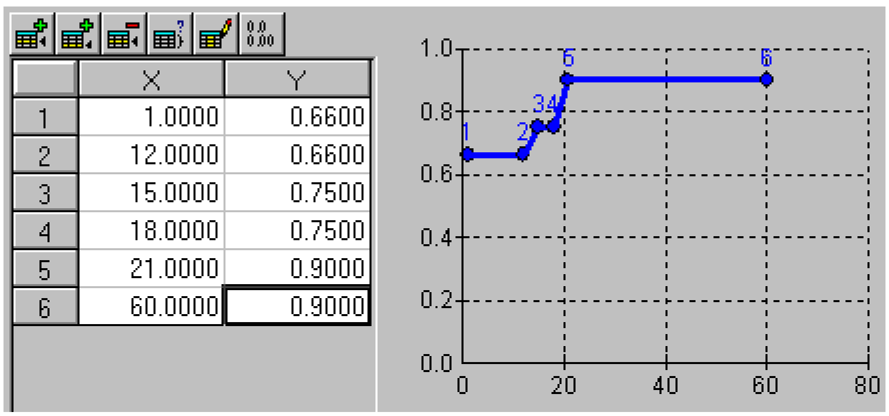


	X	Y
1	0.0000	0.0000
2	0.1000	10.0000
3	0.2500	20.0000
4	0.4500	30.0000
5	0.6000	40.0000
6	0.7000	50.0000
7	0.8000	60.0000

Criteria Curves

A criteria curve represents the relationship between the d/D ratio for a pipe and its diameter. Two types of criteria are specified in sewer modeling. The first criteria is evaluated during the analysis of a steady state simulation while the second criteria is utilized during a design analysis (replacement or parallel pipe).

During a steady state simulation (design and analysis), the analysis and design criteria curves are utilized for determining the d/D ratio, flow and excess capacity for each pipe in the system. When the analysis criteria is exceeded, a replacement and parallel pipe are recommended and provided in a steady state design analysis. Click here to learn more about steady state modeling.

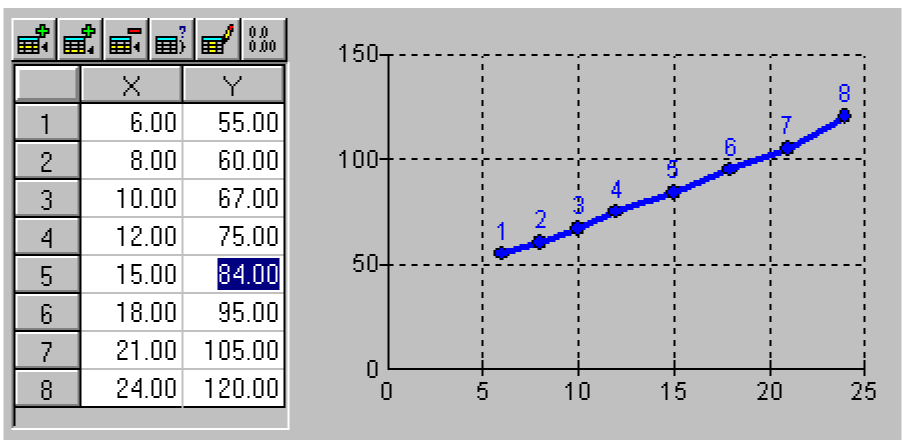


Cost Curves

A cost curve is used by InfoSewerduring a steady state design simulation. When velocity or d/D analysis criteria are exceeded, replacement and parallel pipes are derived. With this derivation, a monetary value is assigned to the replacement pipe via the cost curve. These costs are then displayed in the *Active*.Design Pipe Design Report.

To create a cost curve, simply assign a monetary value each pipe diameter in the system. Two cost curves can be established in InfoSewer. The first is for replacement pipes while the second curve is for duplicate (parallel) pipes. Generally speaking, monetary values for replacement costs should exceed parallel costs.

Cost curves are assigned under the Design tab of the Simulation Options.



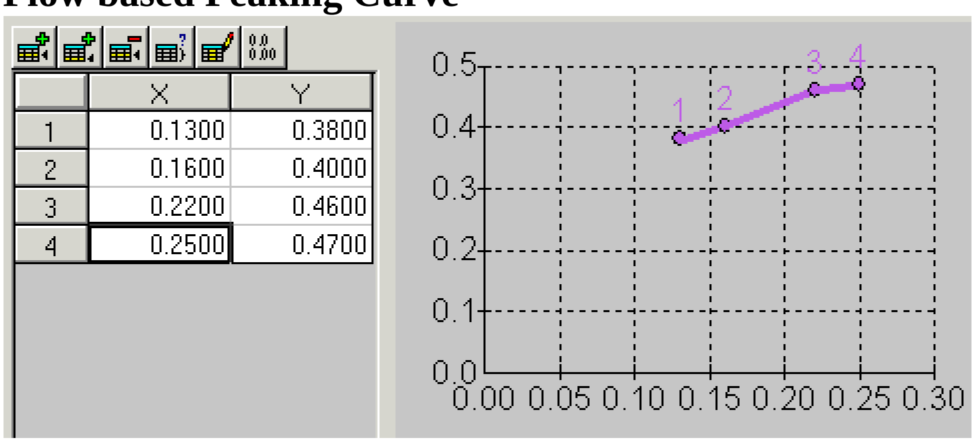
[Peaking Curve]

The Peaking Curve method is an alternative method to compute peak flow (loads) for steady state analysis and design simulations. Two peaking curve types are available depending on the load (i.e., flow) type. If the flow type is peakable coverage flow (i.e., derived based on number of population served), the modeler may use the population based peaking curve. However, if flow type is peakable base flow, the model may use flow based peaking curve. Please, note that curve based peaking methods are provided as an alternative to Federov's equation and Harman and Babbitt's equation. The model may use either the curve based peaking approach or the equation based approach. To learn more about the parameters of Federov's equation and Harman and Babbitt's equation, please click here.

The flow based peaking curve represents peak flows (Y-axis) as a function of base flows (X-axis), where as the population based peaking curve represents peaking multiplier (Y-axis) as a function of number of population (X-axis). Based on this curve and the cumulative population contributing to a desired pipe , the model derives the peaking multiplier, and multiplies it by the base flow to generate the peak flow.

To create a flow based peaking curve, assign a peak flows under the y-axis and the base flows on the x-axis. To create a population based peaking curve, assign a peaking multiplier under the y-axis and the number of population in the x-axis. Peaking curves are assigned under the Steady State Analysis tab of the Run Manager.

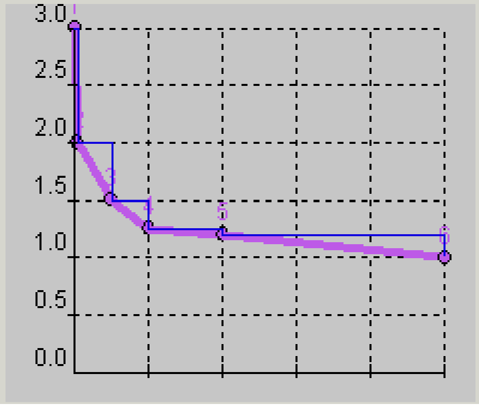
Flow based Peaking Curve



	X	Y
1	0.1300	0.3800
2	0.1600	0.4000
3	0.2200	0.4600
4	0.2500	0.4700

Population based Peaking Curve

	X	Y
1	0.0000	3.0000
2	100.0000	2.0000
3	1000.0000	1.5000
4	2000.0000	1.2500
5	4000.0000	1.2000
6	10000.0000	1.0000



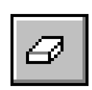
[Create a Curve]

To create a curve, from the Control Center -> Operation Data dialog box, select and highlight Curve and click on the New icon. At this point the user is asked to provide a name and description for the new curve. When created, the curve dialog box will appear and the user is able to enter the desired X,Y coordinates of the new curve.

[Edit a Curve?]

From the curve dialog box, the user can change any X,Y value as necessary to reflect the relationship between the two values (ex. more flow at the same head). The user can also edit the graph directly by selecting either the X or Y field to the left and then clicking on the graph. With the X or Y field highlighted, the user can adjust the graph accordingly.

[Delete a Curve?]

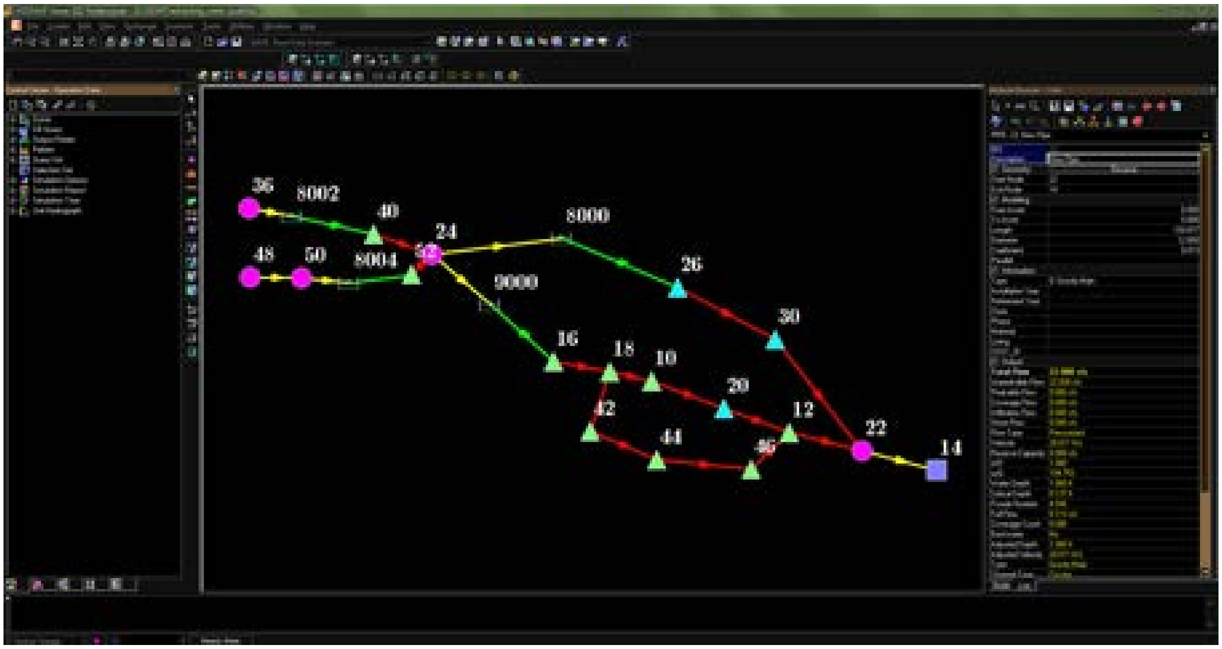
To delete a curve, the user can either select the Delete icon  or right mouse click on the curve in question and select *Delete*.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Fancy Map



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Pumps

A pump is a link that imparts energy to a fluid thereby raising its hydraulic head. The relationship describing the head gained to a fluid as a function of its flow rate through the pump is defined as the pump characteristic curve. H2OMAP Sewer represents pumps as links of negligible length which are directly connected to wet-wells. The principal input parameters for a pump are:

- The wet-well ID representing the inlet side of the pump
- The manhole junction chamber ID representing the discharge side of the pump
- The additional number of parallel (identical) pumps with the same characteristics (optional)

H2OMAP Sewer uses different types of pump curves depending on the number of head-flow data points supplied. Three different types of pumps can be modeled as follows:

- Fixed Capacity representing a fixed pump flow. The pump flow is set to the fixed capacity and is independent of the flow entering the wet-well. It is assumed that the flow will transfer at the fixed rate, independent of the head requirements.
- Single-Point Curve - A single-point pump curve is defined by a single head-flow data point describing the desired pump operating point (design point). H2OMAP Sewer assumes that the cutoff head (at zero flow) is 133% of the design head and the maximum flow (at zero head) is twice the design flow. It then treats the curve as a three-point curve.
- Three-Point Curve - A three-point pump curve is defined by three points of operating data. H2OMAP Sewer will fit a continuous function of the form:

$h_G = h_0 - \alpha Q^{\beta}$ (9)

where

hG = head gain imparted by the pump, ft (m)

h0 = cutoff head, ft (m)

Q = flow through the pump (in flow units)

a = a resistance coefficient

b = a flow exponent

By supplying H2OMAP Sewer with the cutoff head ho and two other points [(h1, q1), (h2, q2)], the program is able to estimate values for a and b from:

$$\alpha = \frac{h_0 - h_1}{q_1^{\beta}} \quad (10)$$

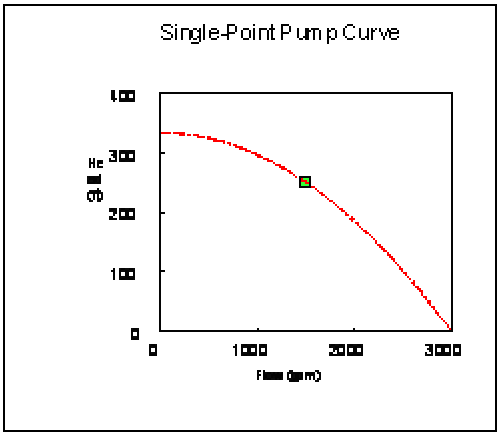
$$\beta = \frac{\log\left(\dfrac{h_0 - h_2}{h_0 - h_1}\right)}{\log\left(\dfrac{q_2}{q_1}\right)} \quad (11)$$

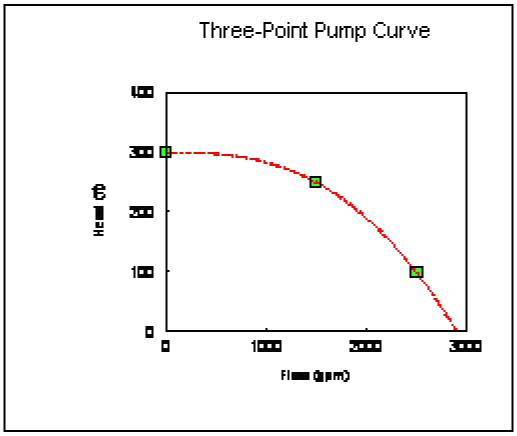
The computed results for pumps are the flow pumped and the head gained.

For variable speed pumps, the pump curve shifts as the speed changes. The relationships between flow (Q) and head (H) at speeds n1 and n2 are

$$\frac{Q_1}{Q_2} = \frac{n_1}{n_2} \quad (12)$$

$$\frac{H_1}{H_2} = \left(\frac{n_1}{n_2}\right)^2 \quad (13)$$





By definition, the original pump curve supplied to the program has a relative speed setting of 1. If the pump speed doubles, then the relative setting would be 2; if run at half speed, the relative setting is 0.5 and so on. The figure below illustrates how changing a pump's speed setting affects its characteristic curve.

If n denotes the pump speed ratio (n1/n2), then the pump characteristic curve (Eq. 10) becomes:

$$H_G = H_0 n^2 - \frac{\alpha}{n^{\beta-2}} Q^\beta \quad (14)$$

InfoSewer™
Modeling Made Easy

Data Sets

A data set is one of three components that comprise a scenario. Data sets provide the capability to take a one-time “snapshot” of data (database information) in the active model and store that information separate from the network itself.

In essence, InfoSewer creates separate database tables for each data set created, allowing the user to manipulate the database characteristics of the data set, separate from the "Base" data set. Once new data is stored in a data set, it may be reloaded back into the active model (via a scenario) at any time.

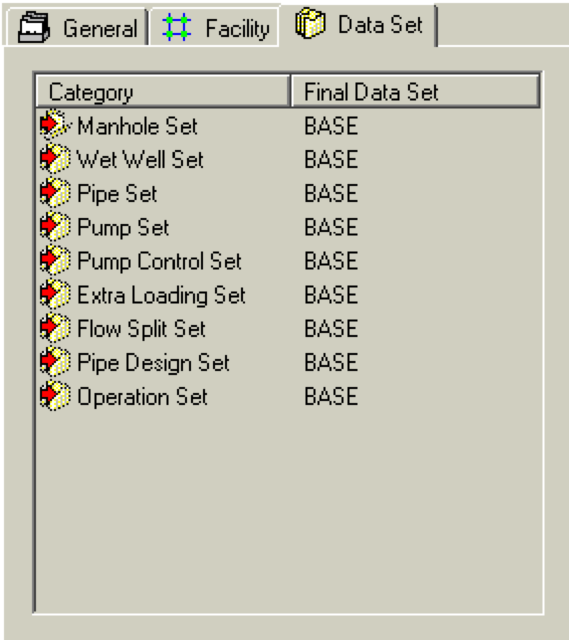
To learn more about data sets, click on the following subjects.

- **Data Sets Defined**
- **Step to Create a Data Set (Tutorial)**
- **How do I Activate a Data Set?**
- **What Happens During a Data Set Activation?**

Data Sets Defined

Category - Represents the specific subset of a data set.

Final Data Set - Represents the data set created and selected by the user or inherited from the parent scenario. Click here to learn more about inheritance.



- **Manhole Set** – Modeling data associated with manholes. Manhole sets are used to save modeling data on manhole loadings and pattern identifiers and retrieve those data as part of a scenario. The attributes stored in a manhole set are the baseline loads, the type of load, associative patterns, and the coverage. The Auto-Manhole Reset preference governs how loadings are assigned when activating manhole sets.

- **Wet Well Set** – Modeling data associated with wet wells. Wet wells are used to save modeling data on wet wells and retrieve those data as part of a scenario. Wet well sets contain modeling information including type of wet well (constant or variable-area) and related wet well node characteristic data such as bottom elevation, initial water level, diameter, minimum and maximum levels, and any curve defining a variable area wet well.

- **Pipe Set** – Modeling data associated with pipes. Pipe sets are used to save modeling data on pipes and retrieve those data as part of a scenario. Pipe sets contain modeling information including pipe diameters, lengths, Manning's roughness coefficients, upstream and downstream inverts and presence or absence of parallel pipes.

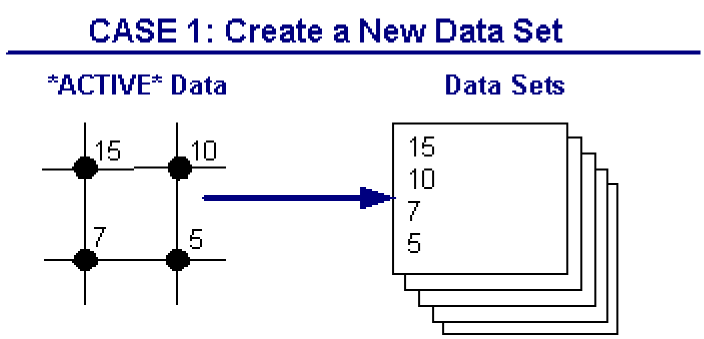
- **Pump Set** – Modeling data associated with pumps. Pump sets are used to save modeling data on pumps and retrieve those data as part of a scenario. Pump sets contain modeling information including curve type and associative parameters.

- **Control Set** – Simple controls used for pumps located at wet wells. Control sets contain logical controls for pumps (on/off settings) as part of a logic set that are stored and recalled as part of a scenario.
- **Extra Loading Set** – An extension of a manhole set, the XLoad set is used to assign additional loadings to a manhole node and include the same data parameters as a manhole set.
- **Flow Split Set** – Flow split percentages or patterns assigned by the user. Flow split sets are only assigned if the automatic calculation is overridden by the user.
- **Pipe Design Set** – Design and analysis criteria curves as well as replacement and duplicate curves assigned by the user to various facilities.
- **Pipe Infiltration set** – Depending on the infiltration modeling option selected, infiltration rate and pattern needs to be created.
- **Operation Set** – All patterns and curves in the H2OMAP Sewer project at the time the operation set is created.

Steps to Create a Data Set

Data sets can be created in one of two fashions:

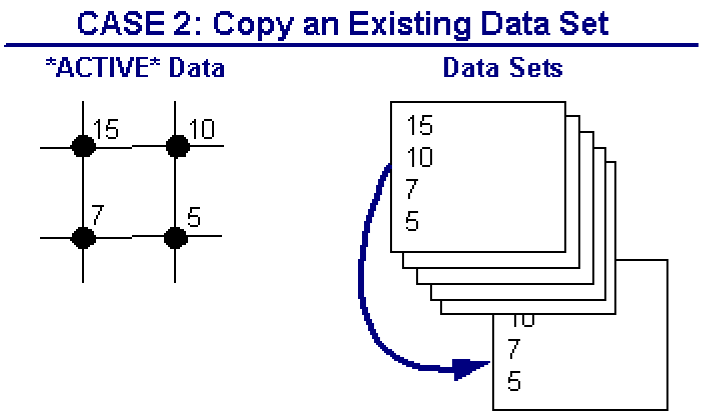
- Create a new data set from the current *ACTIVE* or currently loaded data set into a new data set.



(Note: For all new data sets (with the exception of manholes sets), creating a new set will copy the original database table of the

**ACTIVE* set to the new data set database table. When a new Manhole Set is created (not cloned), the database field for system loadings will be empty.)*

- Clone data from a previously-defined data set into a new data set.

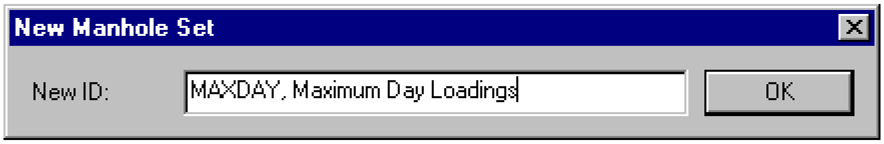


Example Data Set Creation (Tutorial)

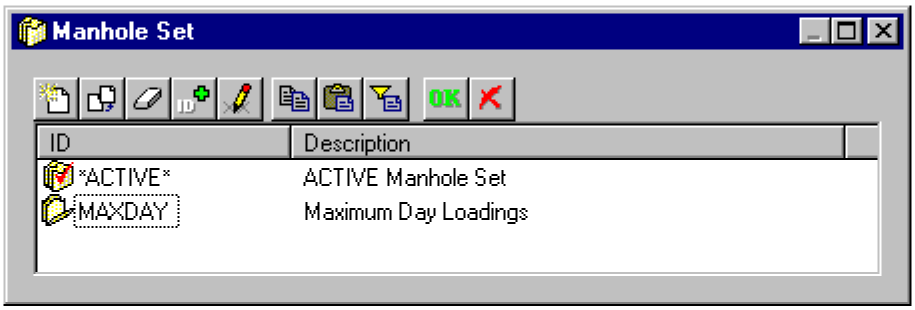
In this example, steps for the creation of a cloned manhole set are provided. For a more detailed tutorial, please see Chapter 3 of the hard-bound User's Guide.

- From the **Scenario** menu, select **Manhole Set**. Once selected, the following dialog box will appear:

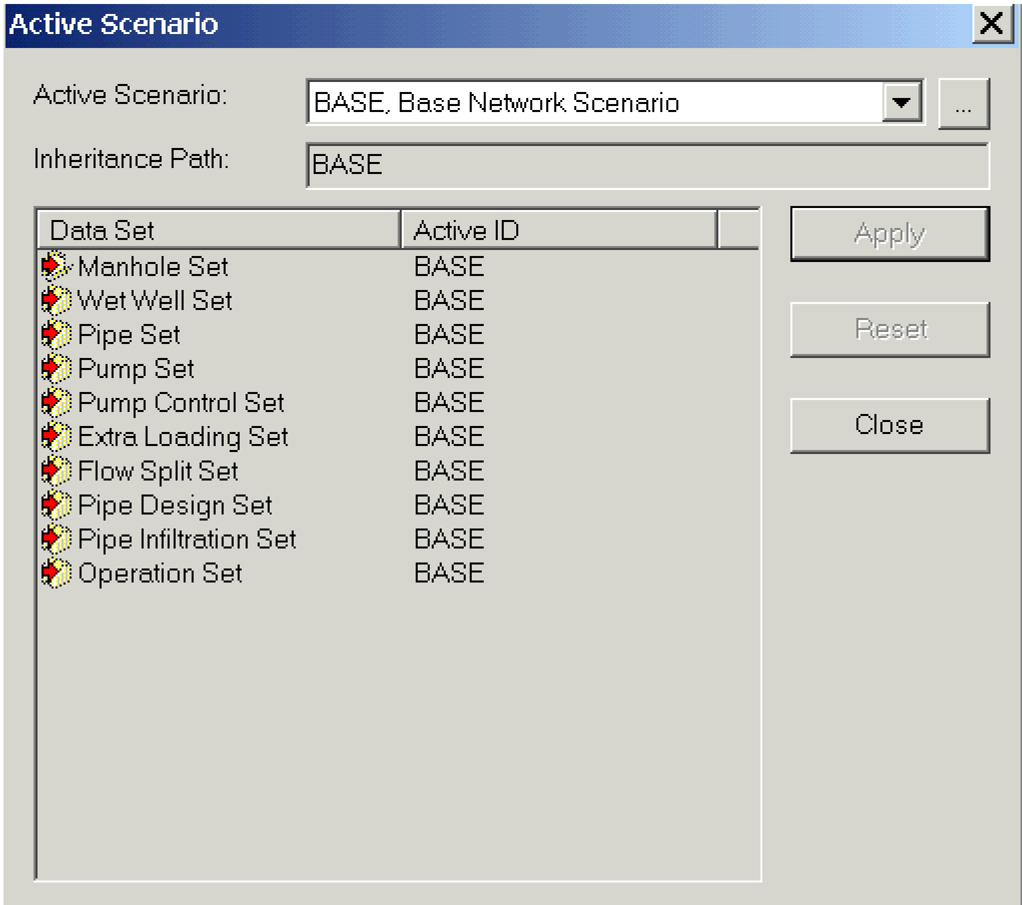
- The *ACTIVE* manhole set represents the Base loading set. Select the Clone icon to replicate the existing *ACTIVE* manhole set. (By selecting clone, the user is requesting that all information in the *Active* database be copied to the new manhole set database - including system loadings.) Once selected, the following dialog box will appear:



- Enter in the unique ID for the manhole set (in this case, MAXDAY) and then provide a description for future reference. Once entered, select OK and the following dialog box will appear:



- Notice that there is no red check mark next to the new MAXDAY manhole set. The reason is because the red check mark indicates whether a data set is currently active or not. To edit the data related to the MAXDAY manhole set it must first become associated with a scenario and made *active*. (Once a data set becomes active, any edits made to a data set will be saved and stored ONLY in the *active* data set.) The next step is to click OK to exit the manhole set dialog box and associate your new data set with a new (or existing) scenario. Go to the **Scenario** menu and select **Active Scenario**. Once selected, the following dialog box will appear:



- The currently *active* scenario will appear in the Active Scenario drop down box (in the example above, FUTURE1). To create a new scenario, select the [...] icon next to the Active Scenario drop down box to see the Scenario Manager dialog box. Highlight the BASE scenario and click on the "New Child" icon. Specify an ID and description for the new scenario and click the OK button. The user is now able to edit any data set for the new scenario by double-clicking on the Category Name (in our case, the Manhole Set) and selecting the MAXDAY manhole set. Once the MAXDAY manhole set is associated with your new scenario, select the Activate icon to activate this scenario.
- At this point, any modifications made to the manhole loadings will be saved to the MAXDAY manhole set (not the original BASE set). So, in essence, by creating this new scenario that now contains the new MAXDAY manhole set, we have created a new manhole database table that is only related to this manhole set (and to the new scenario). At

any time the user can go back to the Scenario Manager and specify a different manhole set (or any other data set). To make it active and edit the database related to the active data set, use the Active Scenario command. Any changes made will be saved and are ready to be "recalled" by the user once any specific scenario is made active.

Activating Another Data Set

There are two methods for loading new data or option sets into the current InfoSewer project:

- Associate a data set with a custom scenario and then activate that scenario – The primary method for activating data sets is by associating them with one or more custom scenarios and then activating one of your scenarios. Definition of custom scenarios and related data sets is accomplished using the Scenario Manager command.
- Load a data set at any time – To change a data set for the currently active scenario, from the **Scenario** menu, choose the **Active Scenario** command. You may load one or more data sets into the active scenario. By doing so, the current modeling information in the open H2OMAP Sewer project will be copied to the active data sets before switching to the new data sets you selected. The contents of the newly-selected data sets are then copied into the open H2OMAP Sewer project and are immediately available.

(Note: The user cannot switch data sets for the BASE (default) scenario. The user may only switch data sets for custom-developed scenarios.)

The second option is most commonly used when the user has not developed multiple scenarios - yet still wishes to interchange data sets for modeling purposes (trial and error before scenarios are created).

What Happens During a Data Set Activation

InfoSewer activates different data sets in different fashions. The following describes how each data set type is activated and data from those data sets loaded into the *ACTIVE* scenario:

Pipe, Pump, and Wet Well Sets

When these data sets are activated and where a match exists between a record in the data set and a network component in the activated facility set (the network you see on the map display), the data from the data set overwrite any data currently loaded for those network components. Where there is no match between records in the data set and activated network components, those components retain their current data values.

Control, XLoad, Split, Design and Operation Sets

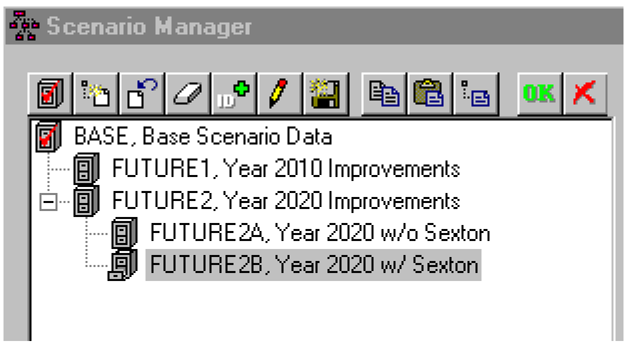
When these data sets are activated, H2OMAP Sewer first clears all data (related to these data sets) from network components in the activated facility set (the network you see on the map display). Then, where a match exists between a record in the data set and a network component in the activated facility set, the data from the data set are assigned to those network components. Where there is no match between records in the data set and activated network components, those components remain with zero or null data values.

By default, manhole set activation follows the same rules as Pipe, Pump, and Wet Well Sets above, where data in the *ACTIVE* data sets are retained and only overwritten if there is a match between records in the data set and activate network component facility set. However, you may specify that manhole set activation follow a similar procedure as Control, XLoad, Split, Design and Operation Sets, where data in the *ACTIVE* data sets are first cleared before loading data in the data sets.

The setting of the Auto-Loading Set preference controls the loading of the manhole set procedure. When ON (checked), loading sets are loaded similar to Pipe, Pump, and Wet Well Sets. When OFF, manhole sets are loaded in a fashion similar to all other data sets. To use this function - from the **Tools** menu, select **Preferences**. Under the Map Operations tab, select Auto Manhole Set to control manhole set activation.

Parent - Child Relationships

The relationship between a parent scenario and its child is defined as inheritance. InfoSewer shows this relationship in the form of a directory tree. All scenarios are children of the *BASE* scenario until some element of the child is made unique from the parent (such uniqueness comes in the form of data sets, facility sets, or option sets). In the example below, FUTURE2 is a child of BASE and FUTURE2B is a child of FUTURE2. When a change is made to a parent, unless some facet of a child is unique, it will inherit the change through inheritance.



What is Inheritance?

Inheritance refers to the relationship between a parent scenario and one or more of its children. Rather than each scenario existing independently of other scenarios, a scenario may inherit one or more of its properties from a parent scenario. With this capability, there is no need to enter redundant information to numerous scenarios that share the same data. Instead, you simply develop a master scenario (referred to as the parent) and develop one or more scenarios whose properties are dependent on the parent scenario.

For example, supposing that a child has a manhole set that is exactly the same as the parent. When a change is made to the parent manhole set, the change will also be reflected in the child. To make the child independent with respect to system loadings, create a new manhole set specifically for the child and assign the new manhole set to the child scenario. In this case, it will ensure that the child is independent of the parent with respect to its manhole set.

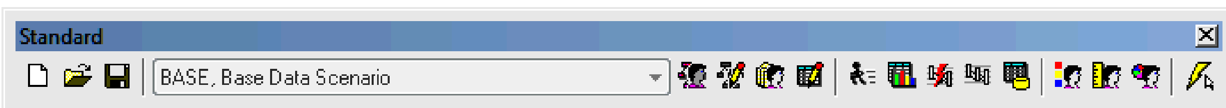
Explained further, once a child is created from a parent, the relationship is dynamic, meaning that when some piece of data is changed in the parent (like a pipe set), the child reflects that change as well. However, where a property is explicitly changed in a child scenario, the inheritance relationship for that property is broken and the change is reflected only in the child.

Toolbars & Icons

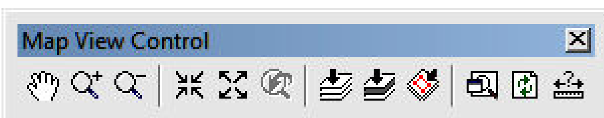
H2OMAP Sewer provides the following toolbars and icons. Click on any toolbar to learn more about the associative icons.

Edit Network

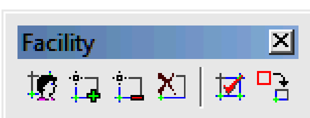
Standard



Map View Control



Facility

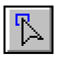


Domain

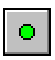
Edit Network Icons

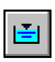
 **Select** - Used to click on any link or node in H2OMAP Sewer. Once the user does this, the information related to that entity appears in the Attribute Browser dialog box. From this box, the user can then edit any attribute related to the data element.

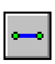
 **Move Node** - This icon is used to move a node from one location to another. Once initiated, click on the desired node and hold the mouse button down (creating a red rubber band). When the node is moved to its new location, release the mouse button to end the move. The move command will remain initiated until the Select icon is chosen. It is important to note that if the Tools -> Preferences -> Auto Length Calculation is not checked (on), any links attached to the node will not have their lengths automatically updated.

 **Edit Link Vertex** - Used to change the location of intermediate vertices along a link. The length of the link is automatically updated as long as the auto-length option is enabled.

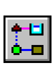
 **Redraw Link** - The redraw link command is used to re-digitize a link. Once a link is selected, the user can relocate a link to its new configuration.

 **Add/Insert Manhole** - Use this icon to create a manhole node. The user is given a dialog box to enter a unique identifier for the manhole. The information related to that entity now appears in the Attribute Browser dialog box. From this box, the user can then edit any attribute related to the data element.

 **Add Wet Well** - Use this icon to create a wet well. The user is given a dialog box to enter a unique identifier for the wet well. The information related to that entity now appears in the Attribute Browser dialog box. From this box, the user can then edit any attribute related to the data element.

 **Add Pipe** - Use this icon to create a pipe. The user is given a dialog box to enter a unique identifier for the pipe. The information related to that entity now appears in the Attribute Browser dialog box. From this box, the user can then edit any attribute related to the data element.

 **Add Pump** - Use this icon to create a pump. The user is given a dialog box to enter a unique identifier for the pump. The information related to that entity now appears in the Attribute Browser dialog box. From this box, the user can then edit any attribute related to the data element.

 **Digitize Network** - This icon is used to create an entire network without using the *Add Pipe* or *Add Manhole* icons. When selected, the mouse pointer changes to a crosshair. Click anywhere on the screen to place a manhole

node, then digitize a pipe as desired. When a pipe has been placed, double-click the mouse over one location to place the "to-node" and to exit the command. Nodes and pipes will be numbered via the Tools -> Preferences -> Suggest ID tab.

 **Edit Text** - Used to place and edit text inside an H2OMAP Sewer project. When this icon is selected, the Text Property dialog box is made available. The user is able to change the font and color of the text with their respective buttons. The "Text" box at the bottom of the dialog box is where the user creates and edits text. Click here to learn more.

 **Edit Selection Attributes** - By clicking on this icon, the mouse cursor changes by adding a question mark. To edit a group of selected attributes, single mouse click and then drag the mouse box over the attributes while holding down the mouse button. When all entities are selected, let go and then right mouse click to initialize the Enter Command. The Edit Attribute dialog box will now appear, allowing the user to edit the associated database fields related to the selection.

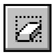
 **Edit Domain Attributes** - Same as the Edit Selected Attributes command, but will only work after a domain has been created. To create a domain, use the Domain Manager icon under the Domain toolbar.

 **Group Editing on Selection** - This command is used to assign an initial value to the elements being edited. From this command, the user selects a group of entities that required some initial value. Once initialized, the Group Editing dialog box will appear to assign these initial values.

 **Group Editing on Domain** - Same as the Group Editing on Selection command, but will only work after a domain has been created. To create a domain, use the domain manager icon under the Domain toolbar.

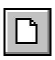
 **Delete a Node** - This command will delete a singular node and the links connected it. When initialized, this command will remove not only the graphical entities, but the database records associated with those entities. To recall a deleted node, go to the **Utilities** menu, point to **Recall**, the select **Node**.

 **Delete a Link** - This command will delete a link. When initialized, this command will remove not only the graphical entity, but also the database records associated with the entity. To recall a deleted link, go to the **Utilities** menu, point to **Recall**, the select **Link**.

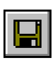
 **Delete a Selection** - This command will delete a selection created by the user. When initialized, this command will remove not only the graphical entities, but the database records associated with those entities. Use the Recall command to restore deleted elements.

 **Delete a Domain** - Same as the Delete Selection command, but only works on a domain that has been created.

Standard Icons

 **New Project** - Use this command to create a new H2OMAP Sewer project.

 **Open Project** - Use this command to open an existing H2OMAP Sewer project.

 **Save Project** - Use this command to save the current session of H2OMAP Sewer.

Change Scenario - The change scenario drop down is used to select a previously created scenario and make it the active scenario. Once a scenario is activated, all graphical and database records related to that scenario are active and able to be modified. The Change Scenario drop down menu is usually used to make a scenario active prior to a model run.

 **Scenario Manager** - The scenario manager icon is used to create, edit, and delete scenarios for differing modeling conditions. From the Scenario Manager dialog box, the user can create new scenarios from parent scenarios, modify sets, and activate a new scenario prior to a model run.

 **Edit Active Scenario** - The edit active scenario command allows the user to view differing scenarios and enable any scenario as the active

scenario.

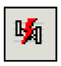
 **Dataset Manager** - This command initializes the dataset manager dialog box which allows the user to view all 9 data sets used in H2OMAP Sewer. From this dialog box, the user is able to create new data sets for use in future scenarios.

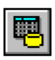
 **DBEditor** - The Database Editor is used to view and edit H2OMAP Sewer databases. From this editor, the user is able to change any piece of data related to any facility in H2OMAP Sewer.

 **Run Manager** - This command initializes the Run Manager dialog box. The Run Manager is where the user specifies options regarding a hydraulic (or other) simulation.

 **Output Report Manager** - This command initializes the Output Report Manager dialog box. It is here that the user can generate graphs, reports, and perform comparisons between multiple model runs.

 **Input Profile** - This command initializes the Pipe Profile dialog box which allows the user to generate an input data profile for a pipe or a group of pipes.

 **Output Profile** - This command initializes the Pipe Profile dialog box which allows the user to generate a profile of a pipe or a group of pipes which includes the output data of a solved simulation.

 **Query Report Manager** - This icon is used to initialize the Query Report dialog box where the user can use a query to create a customized report. In essence, the query report manager is the same as a customized report, just a shorter process and not as robust (no time features, etc.).

 **Project Preferences** - This icon will initialize the Project Preferences dialog where the user is able to change a multitude of graphical display and modeling options regarding H2OMAP Sewer.

 **Output Unit Manager** - This icon will initialize the Output Unit Manager which enables the user to change the units used in H2OMAP Sewer. (For example, using flow units of cfs instead of gpm.)

 **Add-On Manager** - This command allows the user to enable add-on features to H2OMAP Sewer like the Sewer Load Allocator program.

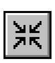
 **Execute Hot Link** - This icon will initialize the hot link action and associate file that has been attached to the selected element. Click here to learn more about hot links.

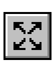
Map View Control

 **Pan** - This command allows the user to pan around the screen to view features not presently on the map display. This command can also be initialized during a link creation (pump or pipe) to connect to a node not on the screen.

 **Zoom In Window** - When selected, the user is required to locate two corners of a zoom window on the map display. When initialized, this window will become the display extents of a "zoom in".

 **Zoom Out Window** - When selected, the user is required to locate two corners of a zoom window on the map display. When initialized, this window will become the inverse of a zoom in window, via zooming out.

 **Zoom In** - This command will zoom in the map display via a set magnification factor.

 **Zoom Out** - This command will zoom out the map display via a set magnification factor.

 **Zoom Previous** - When initialized, this command will zoom the user to the previous zoom extents.

 **Full Extents** - Zooms the user to the maximum extents of all graphical elements.

 **Zoom to Active Layer** - Zooms the user to whatever layer is currently active. The active layer is selected from the Control Center -> Map Legend tab by highlighting the layer in the Map Legend.

Named Views - Allows the user to create and save named views for selected viewing. Once a view has been saved, merely use the named views command to restore the saved view for viewing a particular part of the hydraulic model. This command is especially useful in large system where panning and zooming to particular parts of the system can be time consuming.

Refresh Map - This command refreshes the H2OMAP display.

Measure - Allows the user to take a physical measurement between two or more points. When this command is active, the mouse cursor will change to a cross-hairs and ruler. Once this is done, click on the H2OMAP display the first point from which to measure. A rubber band will now follow the mouse cursor. At the bottom left of the H2OMAP project window, two fields will be displayed. The first is "segment length" which represents the length of the current segment while the second value "length" represents the total length of all segments. Click for the second or any intermediate points to see these two fields updated. It is important to remember that the length is only relevant when your water system has been digitized to some known coordinate system or user defined scale.

Facility

Facility Manager - This icon initializes the Facility Manger dialog box which enables user to create active facilities for use in modeling. The Facility Manager is also used in conjunction with the Scenario Manager to create facility sets.

Activate Elements - Use this icon to make elements (through a MAP selection) active. Simply click on the MAP display and drag a window over the elements you wish to activate. *(Note: Elements must obviously be inactive before you can make them active.)*

Deactivate Elements - Use this icon to make elements (through a MAP selection) inactive. Simply click on the MAP display and drag a window over the elements you wish to deactivate. *(Note: Elements must obviously be active before you can make them inactive.)*

Reset Active Network - Using this icon will reactivate the entire network, making all H2OMAP Sewer facilities active.

Activate All - Clicking this icon will activate all data elements in the current H2OMAP Sewer project, regardless of their current *active* status.

Activate Domain - This feature will activate a set of elements that have been chosen via the Domain Manager. The user must first create a domain in order to use this feature.

Domain

Domain Manager - Initializes the Domain Manager dialog box. From here the user has complete control over domain creation.

Enlarge Domain - Using this feature allows the user to graphically expand a domain from a window selection. *(Note: To select single entities, hold the <Shift> key and select individual elements.)*

Reduce Domain - Using this feature allows the user to graphically reduce a domain from a window selection. *(Note: To select single entities, hold the <Shift> key and select individual elements.)*

Clear Domain - This icon will clear any domain created by the user, returning all elements to their original color.

InfoSewer™

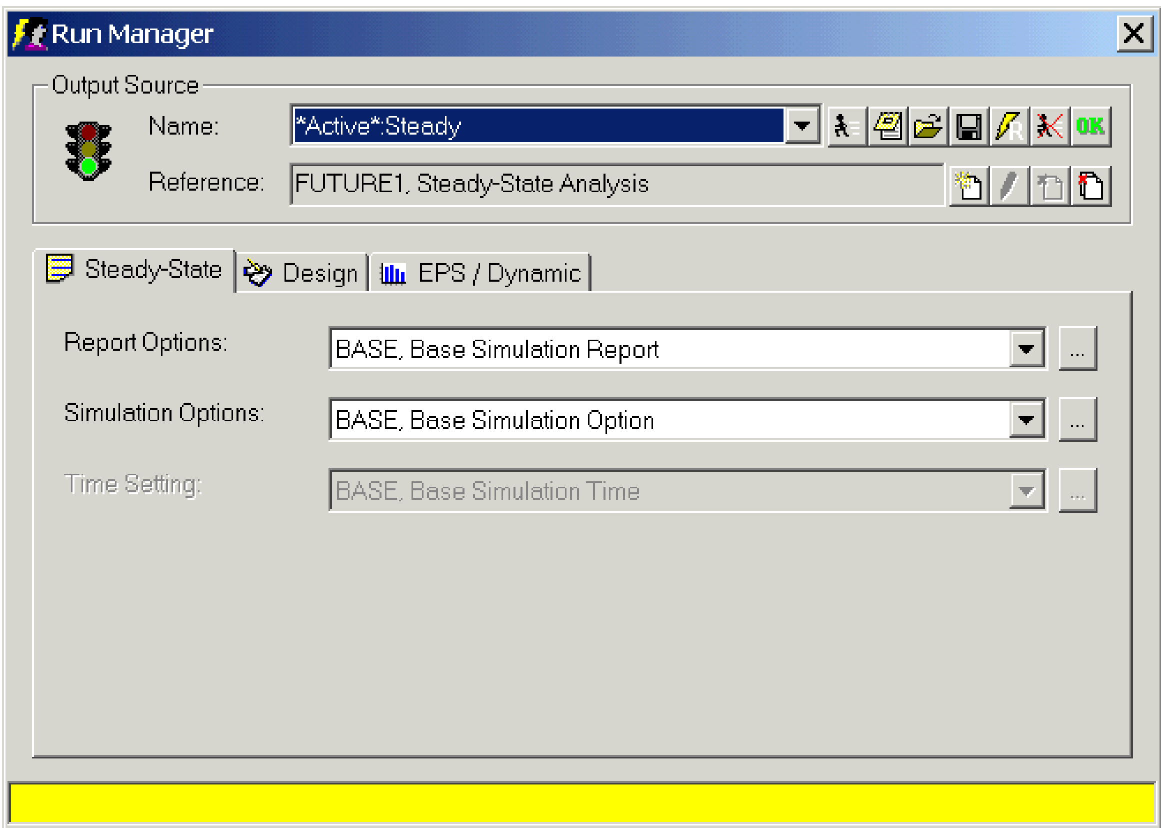
Modeling Made Easy

The Run Manager

The Run Manager is used to perform simulations and to manage simulation output sources (results). Three individual simulation types are available from the Run Manager: Steady-State , Design, and Extended Period Simulation/Dynamic simulation (including quality analyses). You may also use the Batch Run Manager to run multiple standard simulations with one operation.

To activate the Run Manager, from the **Tools** menu, select **Run Manager** .

Click on any portion of the Run Manager dialog box for more information.



Before the Run icon is selected, the user can elect which Report Options and Simulation Options are desired for the active scenario (an EPS/Dynamic simulation model requires a Time Setting as well). Each scenario can have options assigned to it through the General tab under the Scenario Manager. The Run Manager also allows the user to select and replace any option file as desired. This enables the user to do "on the fly" what-if analyses without having to redefine a scenario for different user options.

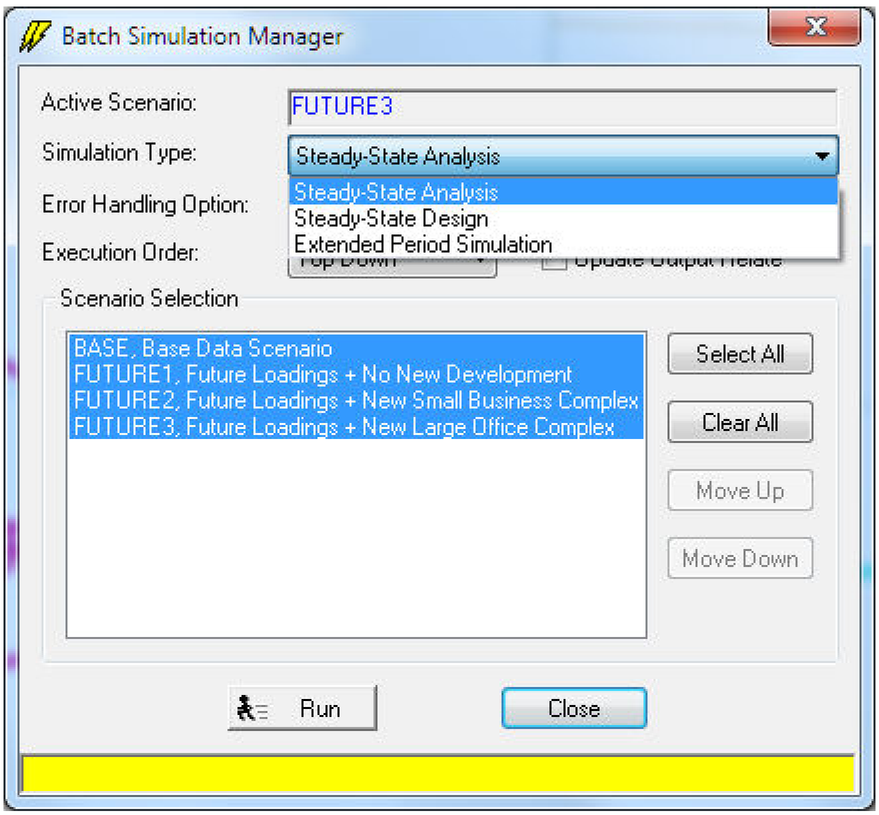
After a model is run, the most recent simulation run results are referred to as the *active* output source. The icons on the Run Manager allow the user to run simulations and create new *active* output sources, save output sources for later use, or load output sources for visualization, analysis, and comparison with other output sources. Use the Output Report Manager to see simulation results in either report or graph formats.

When you run a simulation, H2OMAP Sewer uses modeling data associated with the currently active scenario. If you have not developed a custom scenario, the *active* scenario is the “BASE” condition. To run a model based on a different scenario, activate the desired scenario using the Active Scenario command and then use the Run Manager to perform the simulation based on the data sets associated with the selected scenario.

Batch Run Manager in InfoSewer

The Batch Run Manager is used to run models for numerous user-selected scenarios in a single operation. This command is especially useful where several simulations are simultaneously required for a large model. With the Batch Run Manager, the user can select the desired scenarios and run each model in a "batch" process.

To activate the Batch Run Manager, from the **Tools** menu, select **Batch Simulation**.



- **Active Scenario** - Lists the active (currently loaded scenario) before starting the batch run process and the scenario currently being processed while the batch run is in progress.
- **Simulation Type** - Indicates whether a steady state analysis, steady state design or extended period simulation will modeled.
- **Error Handling Option** - Indicates how InfoSewer will proceed should an error occur during simulation of one of the scenarios

during a batch (multi-scenario) simulation run.

- *Continue Regardless of Errors* – When an error or warning occurs during a simulation, InfoSewer will open a window to indicate the error encountered and will continue to the next scenario to be processed.
- *Stop When Errors Occur* – When an error or warning occurs during a simulation, InfoSewer will open a window to indicate the error encountered and will immediately stop the batch run process.
- **Execution Order** - Indicates the order the selected scenarios will be processed. The last scenario processed remains the active scenario (the scenario whose modeling data will be loaded and available in the open InfoSewer project and whose simulation results are immediately available).
- *Top Down* – InfoSewer processes the first scenario at the top of the list then moves down the list processing each scenario in turn. The last scenario at the bottom of the list remains the active scenario.
- *Bottom Up* – InfoSewer processes the last scenario at the bottom of the list then moves up the list processing each scenario in turn. The first scenario at the top of the list remains the active scenario.
- **Scenario Selection** - Lists all of the scenarios currently available in the open InfoSewer project including the default scenario BASE and all custom scenarios defined by the user. **Only selected (highlighted) scenarios will be included in the batch run.** To include scenarios in the batch run, click once on the scenario name with the mouse. That scenario becomes highlighted. To choose additional scenarios to add to the list, hold the shift key down while selecting the additional scenarios.
- **Run** - Run the Batch Simulation. The Run button is disabled until you choose at least two scenarios for inclusion in the batch run.

After a batch simulation is run, the last scenario simulated will be available as the *active* output source. Other scenarios simulated will be available and loaded as user-defined output sources identified by the scenario names, for example Future1.Steady and Future2.EPS. To review results associated with those scenarios, from the Output Report Manager, choose the desired output source from the All Output Sources box and select either graph or report.

Pipe Profile

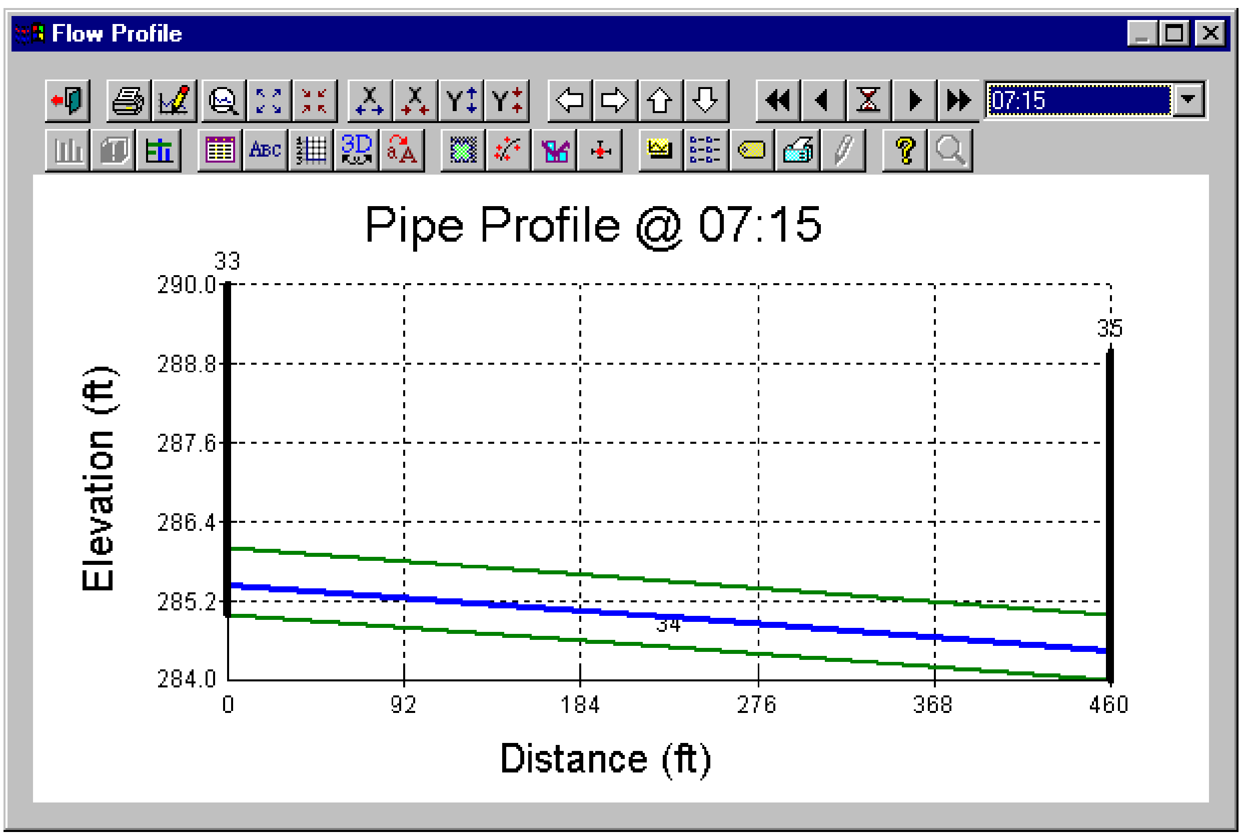
The Pipe Profile feature of H2OMAP Sewer allows the user to create profile graphs of a single pipe or group of pipes. Once a pipe has been profiled, the user has many options to modify the appearance of the pipe profile graph. Click on any of the icons to learn more.

Pipe Profile for Output - Profiles the Output data for a series of links.

Pipe Profile for Input - Profiles the Input data for a series of links.

Creating a Pipe Profile

To create a pipe profile, from the **Tools** menu, select **Pipe Profile**. The mouse cursor will now change with a question mark. Select a single pipe or group of pipes by holding the <Shift> during the selection process. Once complete, right mouse click and select Enter to display the Pipe Profile for the selected pipe(s).



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

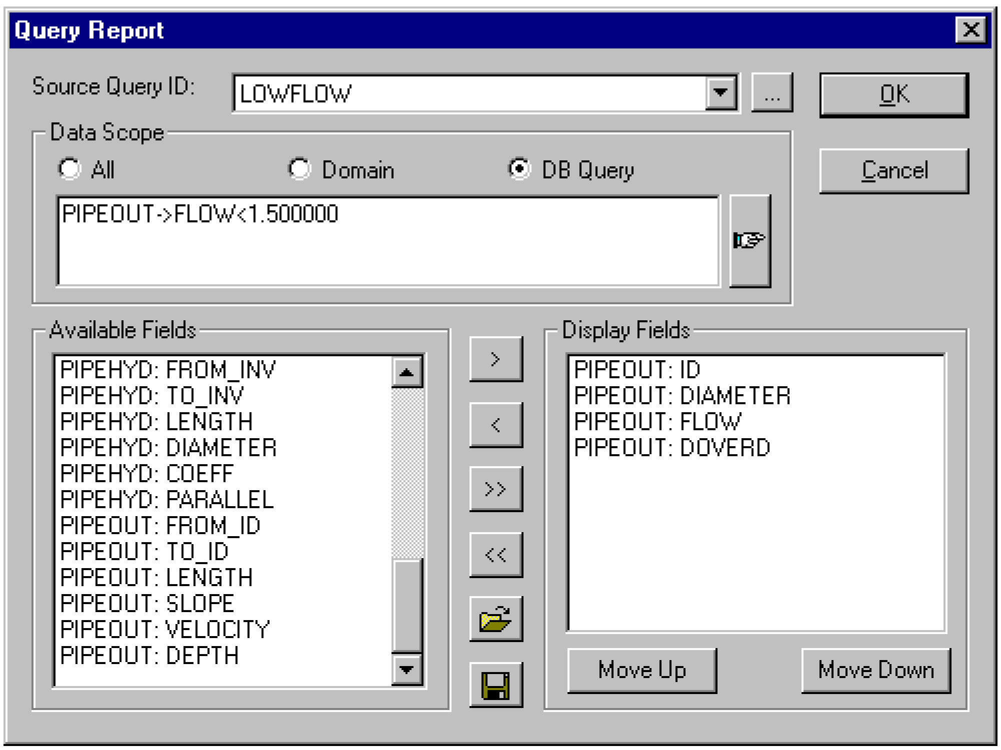
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

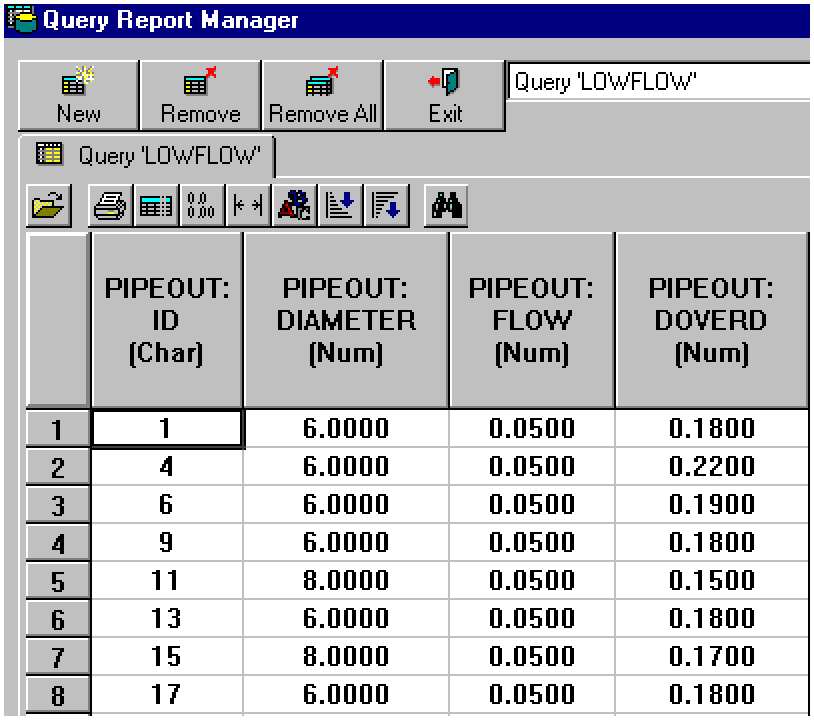
Query Report

Like a customized report, a query report is used to selectively choose which input and output data will appear in a report. However, unlike a customized report, a query report is required to relate data from an Output Relate with output data in a report format. In other words, the only way to see output data in a report format is to create a query report that contains a desired output relate. To follow a step-by-step procedure in creating a query report, click here.

To create a query report, from the **Tools** menu, select **Query Report**. Once this is done, the following dialog box will appear. Click on any portion to learn more.



Once the OK button is clicked, a query report is created as shown below.



	PIPEOUT: ID (Char)	PIPEOUT: DIAMETER (Num)	PIPEOUT: FLOW (Num)	PIPEOUT: DOVERD (Num)
1	1	6.0000	0.0500	0.1800
2	4	6.0000	0.0500	0.2200
3	6	6.0000	0.0500	0.1900
4	9	6.0000	0.0500	0.1800
5	11	8.0000	0.0500	0.1500
6	13	6.0000	0.0500	0.1800
7	15	8.0000	0.0500	0.1700
8	17	6.0000	0.0500	0.1800

Click here to learn more about the fields presented in a query report.

How to Query Model Results

This quick tutorial will guide you through the query report building process and show you how to create and link an output relate to a query report.

Run a Hydraulic Simulation

- Using the Run Manager, run either a steady-state or EPS simulation.

Establish Output Relate

- Open the Output Relate Manager under the Operation Data tab under the Control Center by creating a new output relate called "PIPEOUT"
- Once the dialog box is open, check the Automatic Update box, select the *Active* Target Scenario, Standard Hydraulic as the Simulation, EPS Gravity Main Report as the Desired Output Report, and 0 for the Desired Report Page (same as timestep 00:00 hrs).
- Once step 3 is complete, click on the "Update Current Relate Now" button. This will ensure that the output results for the latest model can now be queried by the Query Report Manager.

- Choose the “Save” button to save the relate and then choose the “OK” button to close the Output Relate Manager.

Generate Query Statement

- Open the DB Query Manager under the Control Center by creating a new query called "LOWFLOW, Pipe Flow < 1.5 cfs".
- Now associate the “PIPEOUT” output relate with the new query by clicking on the Output Relate tab.
- From the drop down box at the bottom of the dialog box, select "PIPEOUT" and click on the “Add Relate” icon just to the right. “PIPEOUT” is now associated with the DB Query.
- Choose the “Verify All” button to ensure proper association to the selected output relate.

Now Build the Query Statement

- Go back to the Query Statement tab and select Pipe as the element type then click on the query builder icon  to build a query statement.
- Choose the “Clear” button at the bottom right-hand corner of the Query Builder to remove any existing query from the Query Builder.
- Click once on “PIPEOUT-> FLOW” data field. Click once on the “<” operator and type “1.5" in the entry field directly below the list of operators.
- Choose the “Add” button to add the query statement.
- Choose the “Validate” button to ensure that at least one pipe matches the criteria “FLOW < 1.5" If the “Invalid Query Statement or Empty Query Result” message appears, then change the query statement to a valid statement with a non-null result. If the “Valid Query Statement” message appears, then choose the “OK” button to close the Query Builder dialog box and return to the DB Tables dialog box.
- Choose the “Save” button to save the query and the “OK” button to close the DB Query dialog box.

Apply Query and Create a Domain

- Open the Domain Manager (Tools -> Domain -> Domain Manager).

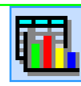
- Choose the “DB Query” option and select the LOWFLOW database query from the adjacent drop down box.
- Choose the “Add” button to highlight all pipes meeting the query criteria: Pipes with flows less than 1.5 cfs at the user-selected EPS timestep. All pipes meeting the criteria will be highlighted (default color is red – use the Preferences command to change the highlight color).
- Choose “Close” to close the Domain Manager dialog box.

Create Output Report

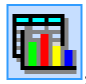
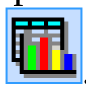
- From the **Tools** menu, select **Query Report**.
- From the Source Query ID, select the LOWFLOW database query.
- The Available Fields box will show all of the fields available from the PIPEOUT output relate while the Display Fields box shows which fields will be included in the query report.
- Use the Display buttons to add and remove fields from inclusion in the output report.
- When all desired fields have been added to the Display Fields box, click OK and the query report will be generated in the Output Report Manager.
- Review and print report as desired.

InfoSewer™

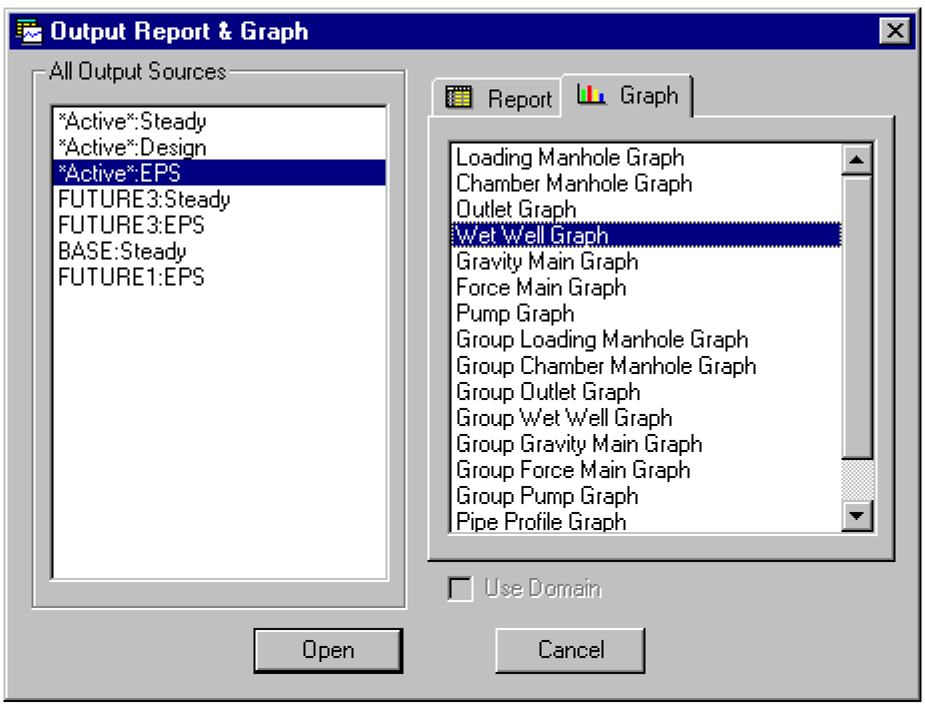
Modeling Made Easy

Output Report Manager 

The Output Report Manager is where the user can generate reports and graphs for any element(s) as modeled in InfoSewer.

To initiate the Output Report Manager , from the **Tools** menu, select **Output Report/Graph** . From the initial dialog box, the user selects the New icon and is provided with the Output Report & Graph dialog box as shown below.

Click on any portion to learn more:



All Output Sources

The All Output Sources section is used to select the output source for which the user wishes to generate a report or graph. The sequence for creating a report or graph is to:

- Select the desired output source.

- Select the desired output report or graph for viewing.

(Note: Output sources are made available from scenarios by running the model for each model. Once a model from a custom scenario is run, it will appear in the All Output Sources box. Click here to learn more about output sources.)

Facility Manager

The Facility Manager is used to create and maintain the active facility set. The active facility set defines the network components in a current model that will be considered during the next simulation run(s). Facility sets can also be associated with a scenario via the Scenario Manager. To assign a facility set to a scenario, click here.

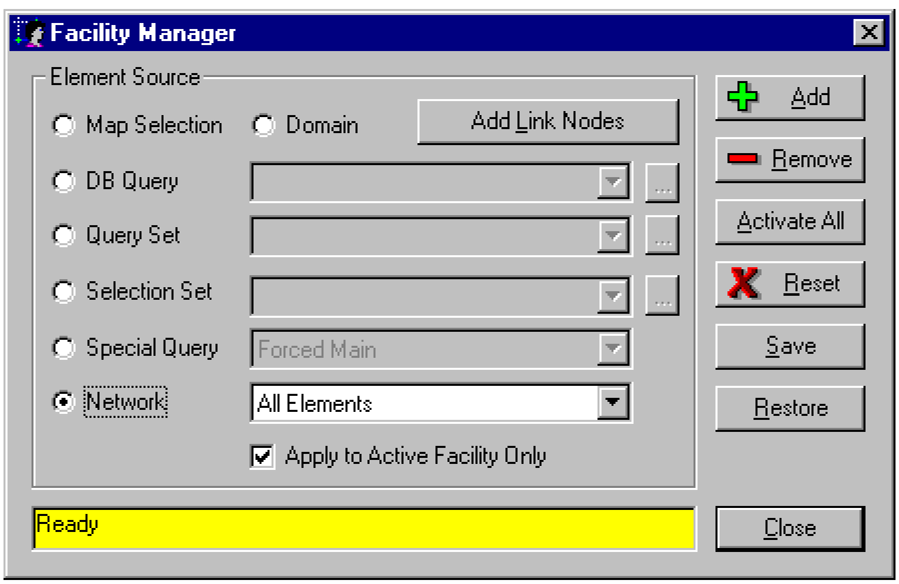
At any point in time, only the active facility set is displayed on the screen. Inactive components are hidden. (To display inactive facilities, from the **Tools** menu, select **Preferences**. Go to the Display tab and check "Show Inactive Elements").

How Do I...

- Create an Active Facility?
- Clear a Facility Set?

From the **Tools** menu, point to **Facility** and select **Facility Manager** in order to activate the Facility Manager as seen below:

To make any set of facilities *active*, use any of the options available from the Facility Manager to "activate" a set of facilities. Click on any portion of the dialog box below to learn more about the Facility Manager



Create an Active Facility Set

To create an active facility set, from the **Tools** menu, select **Facility Manager**. At the Facility Manager dialog box, the user will need to decide if the desired active facility set to be created should be done from some sort of query or graphical selection. Once the user has made a selection, choosing the Add button will add those facilities to the active facility set.

At this point, only those elements activated will retain their color, all others will turn grey or turn off (depending on the options specified under Preferences). If a model were run at this point, only those elements selected would be modeled (as long as the scenario facility option for the currently *active* scenario under the Scenario Manager was set to "Active Network" - click here to learn more.)

Clear a Facility Set

To clear a facility set, from the **Tools** menu, select **Facility Manager** and click on the Reset button or the Activate All button. Doing this will clear the selected facility set and return the elements to their normal color, activating all elements.

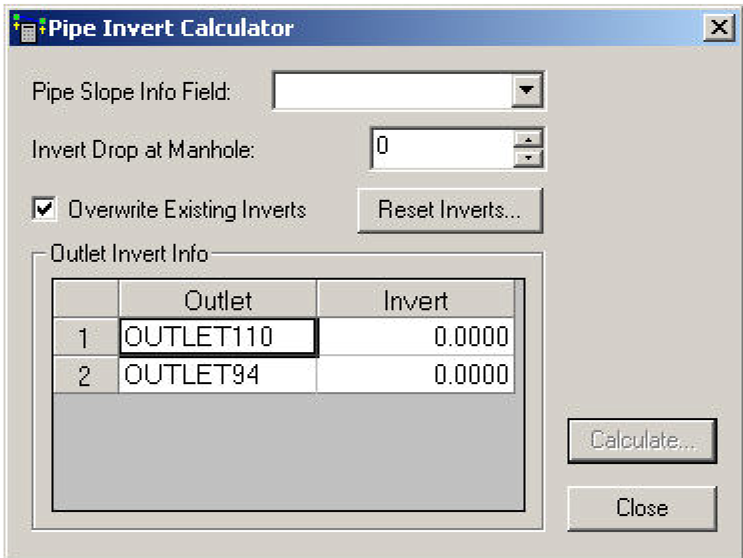
Other Related Topics - Domain Methodology, Facility Methodology, Facility vs Domain, Facility Manager

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Pipe Invert Calculator

The pipe invert calculator feature of InfoSewer allows the user to generate pipe inverts from known slope data for any given number of pipes. The calculator will also retain known pipe inverts during the calculation process, thus providing a complete tool that creates required modeling data from scarce data sources.

When the command is initialized, the following dialog box is displayed:



How to Use the Pipe Invert Calculator

The following procedure illustrates how to create and populate slope information and utilize that data for the invert calculation process.

- Using the DB Editor, open the Pipe Information table from folder 2. Element Information Table.
- Create and append a field in this table and name the new field "Slope". Make this field numeric with at least 6 decimal places.
- Save and close the Pipe Information table.
- Populate the newly created "Slope" field for all pipes in the system where invert data is not available.

- Open the Pipe Invert Calculator and in the Pipe Slope Field drop down box, select the SLOPE field.
- Specify any elevation drops across manholes in the system. Most sewer systems in the United States are designed with a 0.1' elevation drop across the manhole.
- Specify whether you like to use existing pipe invert elevations in the calculation or overwrite all existing pipe invert data. To retain known pipe invert elevations during the calculation, make sure the "Overwrite Existing Inverts" check box is NOT checked.
- To erase all inverts in the system, click on the Reset Inverts button.
- Once you have specified these options, enter the invert elevation for all outlets listed in the sewer system. This elevation is usually the invert elevation of the headworks at the sewer treatment plant.
- Once you have specified the desired options, click the Calculate button to run the Pipe Invert Calculator and populate all invert data for the sewer collection system.

NOTE: If you have missed or not populated a pipe slope, the slope value is automatically given a value of 0.0000. Thus, the pipe will be calculated as being flat. Depending on your hydraulic condition, this may have adverse affects while running a hydraulic simulation.

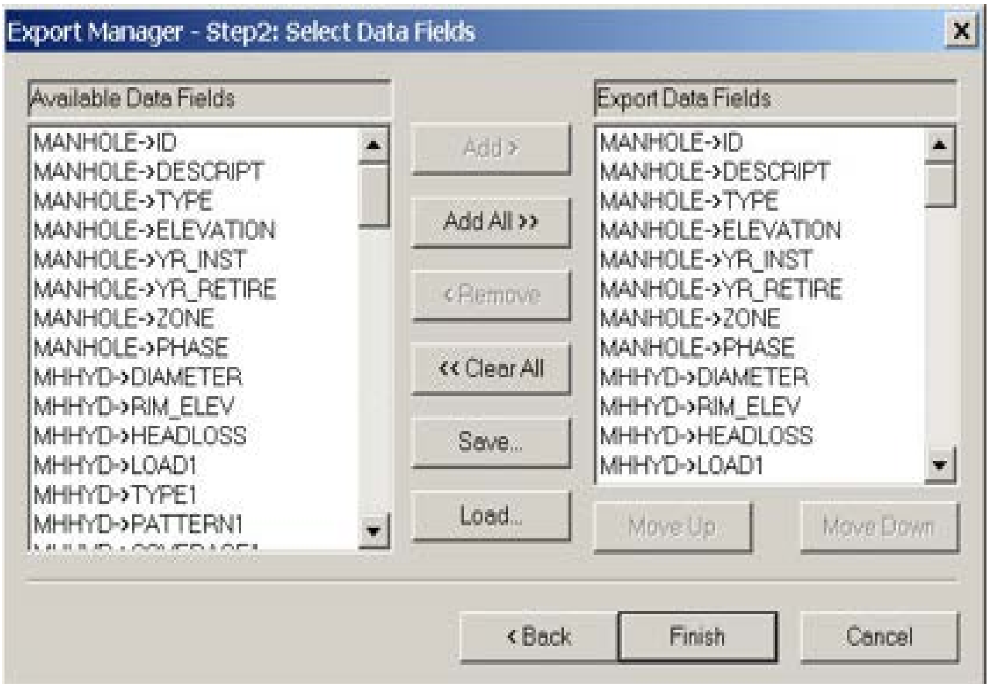
Other Related Topics - Design Non-Circular Cross-Sections, Extra Load, Flow Split, Infiltration, Inflow, Peaking Curve, Pipe Constraints, Pipe Design

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Home > Pipe Invert/Direction Translator

Pipe Invert/Direction Translator

The Pipe Invert/Direction Translator provides a way to "translate" invert information from manhole to pipe



When both fields are "N/A" for one of the pipe inverts, the translation will be skipped.

When either field is valid, the translation will do the necessary calculation and update the related pipe invert field.

Skip Pipes in Domain - This feature allows the user to protect some pipes from being translated.

Apply to Active Facilities Only - will ignore inactive facilities if selected.

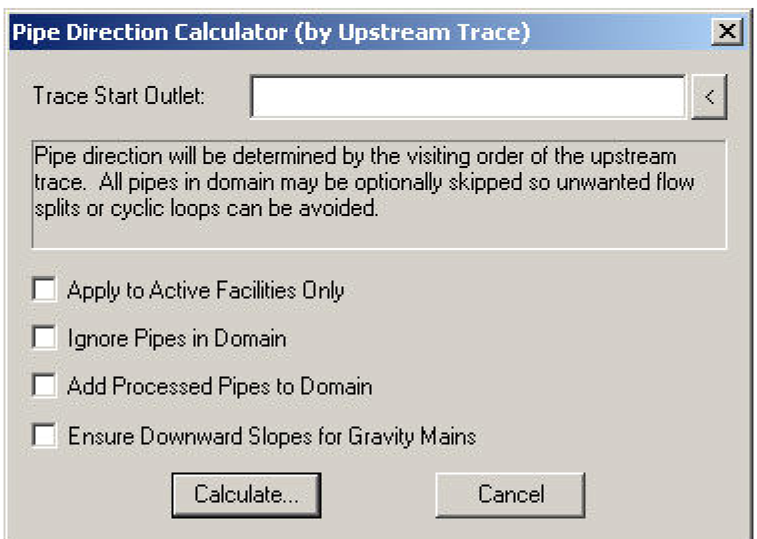
Add Modified Pipes to Domain - Adds all translated pipes to the current domain.

Ensure Downward Slopes for Gravity Mains - When selected, the direction of a gravity main (order of FROM and TO) will be adjusted based on the calculated inverts to ensure a downward slope. This option can work by itself to ensure all eligible Gravity Mains have the proper downward slopes. It adjusts Pipe Direction and its vertex order based on its new or existing From and To Inverts.

Pipe Direction Calculator

Pipe direction is calculated by tracing the pipe direction starting with the downstream and working upstream through the connected pipes.

Pipe Direction Calculator is a material trace tool that can fix the pipe direction (From / To) as well as its corresponding inverts (From Invert / To Invert). It is designed to avoid unwanted flow splits and cyclic loops, and can support network with multiple outlets. Its function relies on the definition of a domain. The key flow split pipes shall be selected into a domain before starting the calculation. The selection shall be based on the design considerations of the system. The main power of this tool is bidirectional trace. The trace is bidirectional thus it disregards all currently defined direction and it is able to calculate a new pipe direction.



Trace Start Outlet - Provide the ID of a downstream outlet or use the browse button to select an outlet from the map view.

Apply to Active Facilities Only - Apply the trace only to active facilities. If not selected active and inactive facilities will be traced.

Ignore Pipes in Domain - By selecting this option and establishing a domain of pipes, it allows the user to exclude some pipes from the direction calculator process.

Add Processed Pipes to Domain - When completed, pipes processed by the Pipe Direction Calculator will be placed in a domain

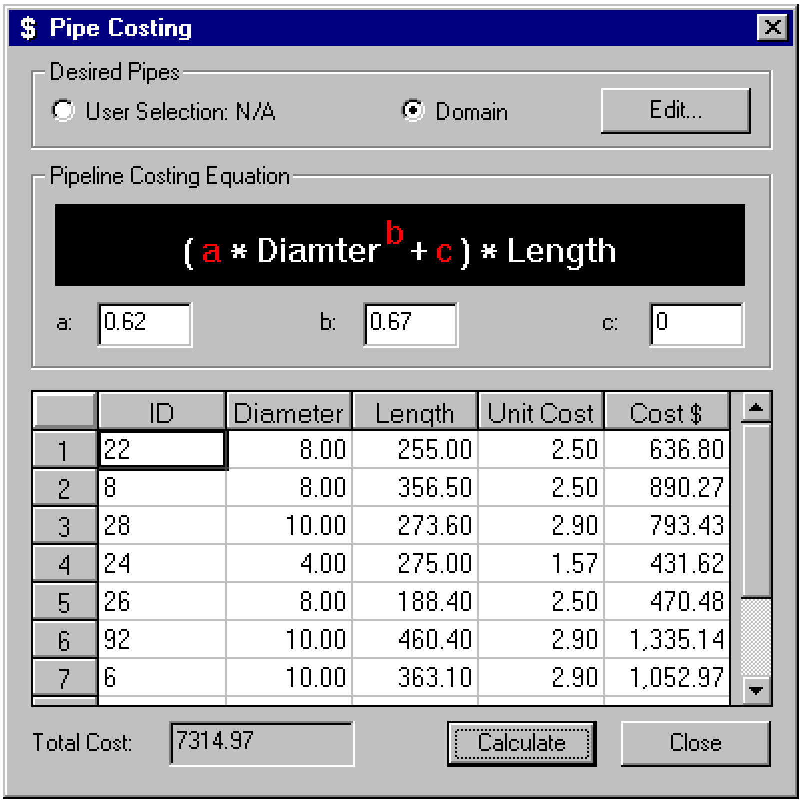
Ensure Downward Slopes for Gravity Mains - When selected, the direction of a gravity main (order of FROM and TO) will be adjusted based on the calculated inverts to ensure a downward slope.

Select **Calculate...** to run the pipe direction calculation.

Select **Cancel** to return to the Map Display.

Pipeline Costing

The ability to determine costs for replacing sewer collection infrastructure can be one of the most timely and difficult tasks in preparing a master plan. The pipeline costing function has been provided to assist the user in establishing these costs for replacing certain pipes as based on the following:



Desired Pipes - Using either a graphical selection or by creating a domain, select the pipes for which a cost analysis is to be performed.

Pipeline Costing Equation - This is the equation for which pipe costs are determined. What will vary is the a, b, and c values supplied by the user that are unique to each costing analysis. The user may wish to apply different costing equations to different domains as created by the user to represent differing parts of the system (ex. freeway versus open field).

Calculate - When the user has selected the pipes and has entered the appropriate a, b, and c values, clicking the calculate button will create a pipeline cost summary for the selected elements. The user can now highlight the desired data and right-mouse click on the selected area. By choosing the copy option, the user can now paste that data into a third party package like Excel or Word.

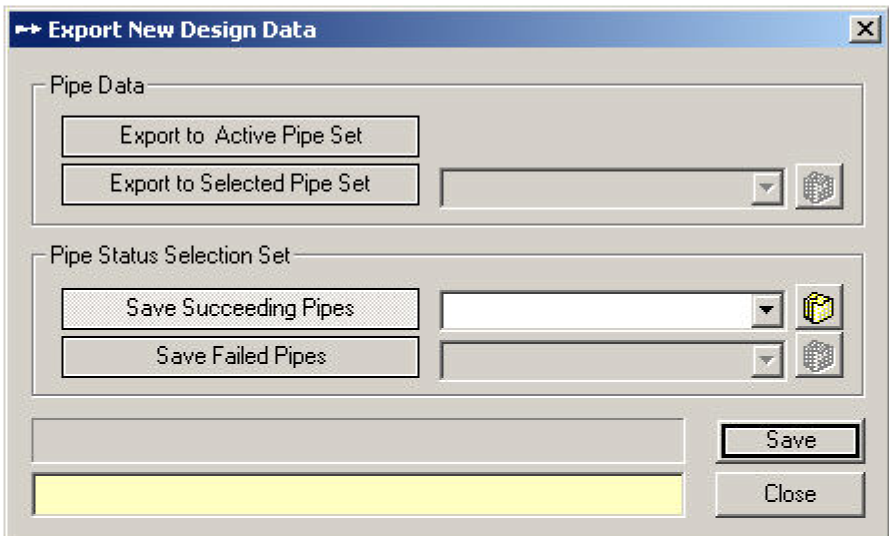
InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Export New Design Data

This tool exports design results - specifically pipe diameter and pipe invert elevations and manning's n for new pipes - to the specified model database enabling the user to build a working model from the design result. Selection sets can also be created from pipes that succeeded and failed during the design simulation.



Export to Active Pipe Set - This option exports design attributes from the design simulation to the pipe set associated with the current scenario.

Export to Selected Pipe Set - This option exports design attributes from the design simulation to the pipe set specified using the drop down. Press to access the data set editor.

Save Succeeding Pipes - This option creates a selection set using the ID's of pipes that succeeded during the design simulation Press to access the selection set editor.

Save Failed Pipes - This option creates a selection set using the ID's of pipes that failed during the design simulation Press to access the selection set editor.

Export - This button appears when one of the pipe data options is selected. Press this button to export the currently selected option to the specified pipe set.

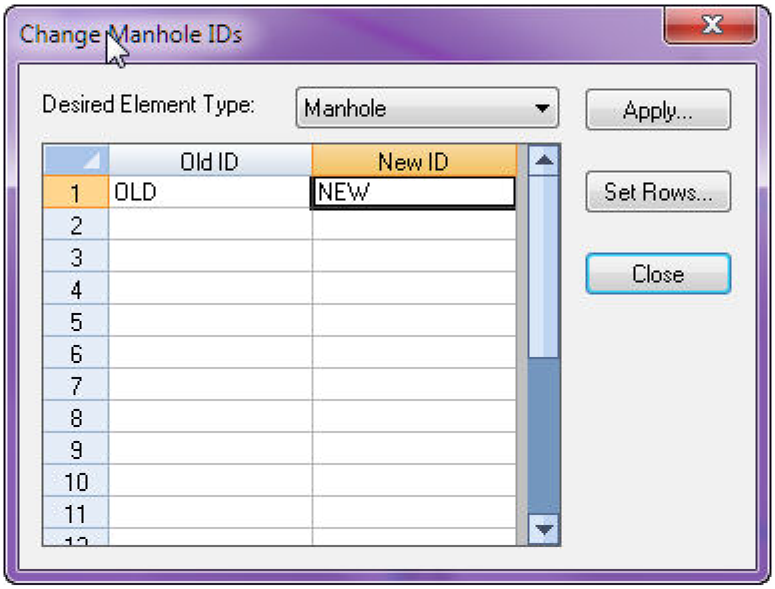
Save - This button appears when one of the pipe status selection set options is selected. Press this button to save the currently selected option to the specified selection set.

Close - Press this button to close this dialog.

InfoSewer™ Modeling Made Easy

Change ID

The Change ID dialog box allows the user to change the ID for any selected data element.



- **Desired Element Type** - From the drop down box, select the H2OMAP Sewer data element for which an ID change is to occur.
- **Old ID** - Enter the current ID.
- **New ID** - Enter the new ID value.
- **Apply** - Click apply to make the ID change.
- **Set Rows** - Specify the number of rows for mass editing (twenty is the default).

Performing a Large Change ID

To perform a mass edit, use the DB Editor and open the database where the subject ID's are stored. Highlight and copy the ID's into the Windows clipboard and paste the values into a third-party software like Microsoft Excel. Next to each ID, enter the new value for the ID - Using excel functions like "mid" and "concatenate" to help you in the mass edit. Once you have the old and new ID's, determine how many rows are being used in

Excel. Highlight all old and new ID's in Excel and use Ctrl+C to copy the highlighted area.

Go back to InfoSewer, use the Set Rows command to make the rows the same as those in the Windows clipboard. Once this is done, highlight the first cell in the Change ID dialog box and use the Ctrl+V function to paste the values from the clipboard. You have now greatly reduced your time from having to edit each ID individually. Click Apply to change the ID's and then Close to close the dialog box.

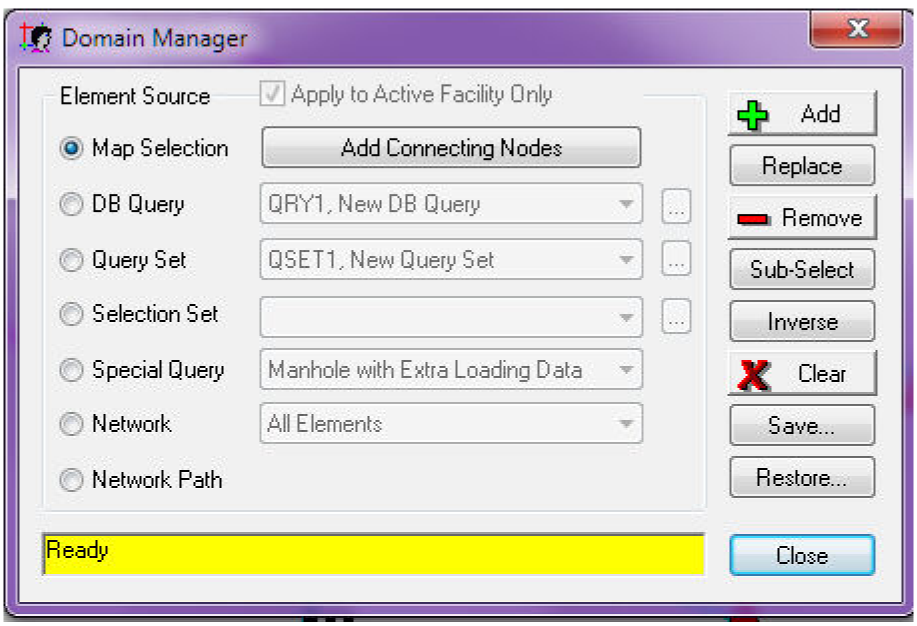
Domain Manager

A domain is a *temporarily* selected subset of network components. The domain can be used for a variety of purposes including group editing operations, mapping, contouring, etc. The Domain Manager is used to create, edit and delete elements contained within a domain. In conjunction with the Domain Manager, InfoSewer also has the Edit Selection Attributes and Edit Domain Attribute icons.

How Do I...

- Create a Domain?
- Clear a Domain?

From the **Tools** menu, point to **Domain** and select **Domain Manager** in order to activate the Domain Manager as seen below. Click on any portion to learn more.



(Note: The Run Manager disregards any selected domain during a model run. If you wish to model a subset of your network, use the Facility Manager to create a facility set rather than a domain.)

Create a Domain

To create a domain, from the **Tools** menu, select **Domain Manager**. At the Domain Manager dialog box, the user will need to decide if the desired domain to be created should be done from some sort of query or graphical selection. Once the user has made a selection, choosing the Add button will create the domain.

At this point, only those elements selected will turn red. By editing the database tables related to these elements, the user will only see the domain affected database records and not the entire database (as long as the domain option is checked under the DBEditor).

Clear a Domain

To clear a domain, from the **Tools** menu, select the **Domain Manager**. With the dialog box open, click on the Reset button. Doing this will clear the selected domain and return the elements to their normal color.

Note: The Run Manager disregards any selected domain during a model run. If you wish to model a subset of your network, use the Facility Manager to create a facility set rather than a domain.

Other Related Topics - Domain Methodology, Facility Methodology, Facility vs Domain, Facility Manager

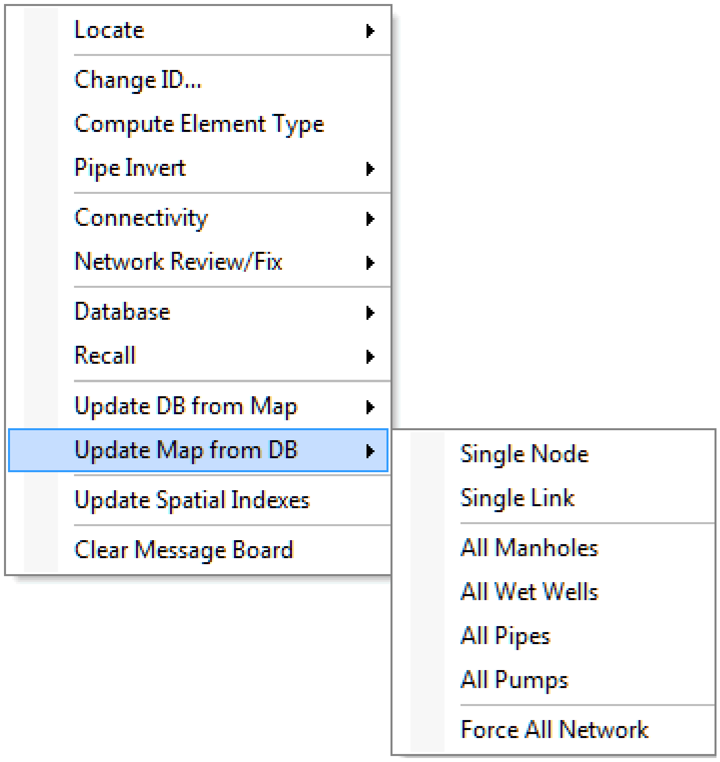
More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Update DB to Map

The DB to MAP command is used to redraw the current InfoSewer project's network map based on the following values in the InfoSewer project database:

- X,Y Coordinates stored with manholes, wetwells, pipes and pumps.
- Connectivity (from and to node designations) and shape (intermediate X,Y shape-defining vertices) for pipes. The length value in the model input database is not used to update the network map.
- The Update MAP from Database command should be used to update the InfoSewer project map in the following situations:
- When records from a non-InfoSewer database (GIS, facility inventory databases, etc.) are imported into InfoSewer using the delimited text (CSV) option. Importing comma-delimited text files updates existing records and/or adds new records to the database for the current InfoSewer project but does not update the corresponding elements on the map display.

To run an update, from the **InfoSewer Control Center -> InfoSewer** button **-> Utilities** menu, select **Update DB to Map**.



- **Single Node** - Updates a single node (**Wet Wells or Manholes**) based on a record in the associated database table.
- **Single Link** - Updates a single link based on a record in the associated database table.
- **All Manholes** - Updates all junction Manholes based on records in the Manholes database table.
- **All Wet Wells** - Updates all **Wet Wells** based on records in the **Wet Well** database table.
- **All Pumps** - Updates all pumps based on records in the pumps database table.
- **All Pipes** - Updates all pipes based on records in the pipes database table.
- **Force All Network** - All drawing elements are removed from the reserved InfoSewer drawing layers and redrawn.

Note - If you have developed custom scenarios in the current InfoSewer project or alternately, are taking advantage of InfoSewer's facility set activation feature, you should be aware of the following:

- The Update Database to Map command updates all network components, not just those activated as part of the current scenario. Changes to inactive components will be shown when those components are activated as part of another custom scenario.
- The information required by the Update Database to Map command are stored in geometry tables stored external to scenario data sets. Therefore the command disregards custom scenario definitions and related data.

Other Related Topics - Update Map to DB

Update Map to DB (database)

The update Map to Database command is used to update the current project's database tables based on the current state of the InfoSewer network drawing. This command should be used in the following situations:

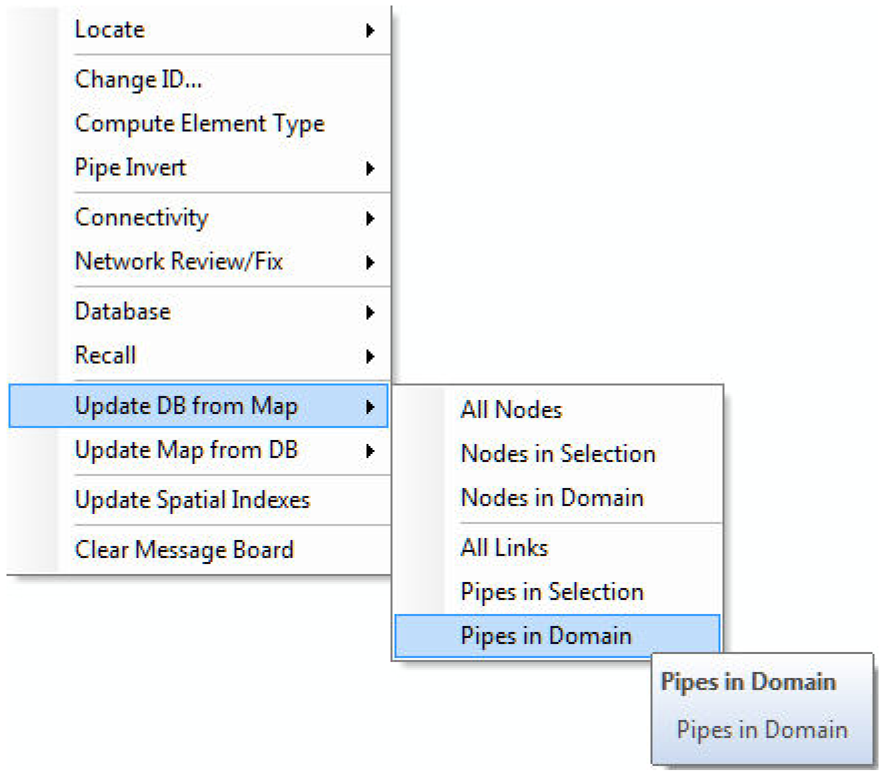
- If an ESRI generate file containing a different configuration than the original network is imported into the active InfoSewer project, replacing existing components in the network drawing.
- If a database record appears to be corrupted or has erroneous data that differs from the InfoSewer project.

Upon terminating the selection process, InfoSewer updates the database records corresponding to the selected components. The Update MAP to Database command updates the following in the InfoSewer project database:

- Nodes X,Y coordinate location.
- Pipes Connectivity (from and to node designation), shape (intermediate X,Y shape-defining vertices), and length for pipes. To

have InfoSewer recalculate pipe lengths, be sure that the Auto-Length Calculation preference is set to ON prior to running the Update Database from ACAD command.

To run an update, from the **InfoSewer Control Center -> InfoSewer** button **->** **Utilities** menu, select **Update Map to DB**.



Note - If you have developed custom scenarios in the current InfoSewer project or alternately, are taking advantage of InfoSewer's facility set activation feature, you should be aware of the following:

- You can only select network components from the active facility set when updating the InfoSewer database tables from the network drawing.
- InfoSewer considers pipe length a modeling attribute and therefore stores pipe length in scenario pipe sets. The update Map to DB command updates the records in the active pipe set only. Records corresponding to the selected pipes in other (inactive) user-defined pipe sets will not be updated with this command.

Other Related Topics - Update DB to MAP

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

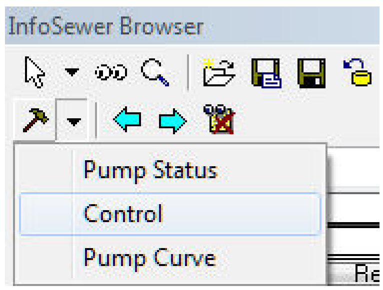
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

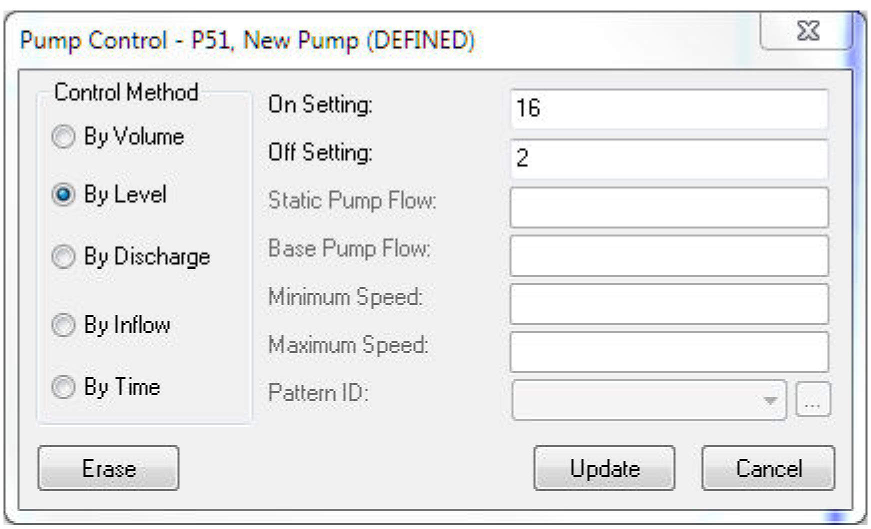
Pump Controls

Pump Control

A pump control allows the user to modify the initial status of a pump during a hydraulic simulation - depending on the state of a wet well in the system as specified by the user. Actual field operations may consist of having a pump turn on and off, depending upon the high and low water levels (or volumes) in a wet well. To model this type of situation, the user would place a simple control on the pump in question. Pump controls apply to extended period simulations only.



InfoSewer provides five control alternatives as shown below.



The alternatives and the variables are described below.

Control Method - The control method specifies the criteria that dictates the ON and OFF settings of a pump.

- **By Volume** - The pump operates depending on the volume of sewage in the wet well. The user should provide the on/off volumes.
- **By Level** - The pump may turn on or off based on water level in the wet well. The on/off levels need to be specified for this control option.
- **By Discharge** - This control method dictates a mechanism in which a user supplied pattern of targeted pump discharged flow is maintained. The software adjusts the speed of the pump to insure that the desired amount of flow is pumped. This option is available for one-point and exponential three-point pumps only, and is not valid for lift (pump) stations with parallel pumps. The actual pumped flow could differ from the desired flow if the minimum and the maximum water levels set for the wet well are violated during the simulation duration. In other words, the pump may turn off if the water level in the wet well falls below the minimum wet well level allowed in spite of the desired flow determined based on the pattern. Like wise, the pump may turn on if the water level in the wet well exceeds the maximum allowed water level even if the targeted flow is zero.
- **By Inflow** - Under this control alternative, the level of water in the wet well remains constant during the simulation duration. The pump turns on if there is inflow to the wet well. The discharged flow is equal to the inflow to the wet well. This control alternative is ideally suited for wet wells that do not have enough storage volume. This option is available for one-point and exponential three-point pumps only, and is not valid for lift (pump) stations with parallel pumps.
- **By Time** - The "By Time" control option offers the user the flexibility to turn the pump on/off at any time of a day. The model accepts the operational schedule in the form of a speed pattern. The pump turns off if the speed setting is zero, and turns on otherwise.

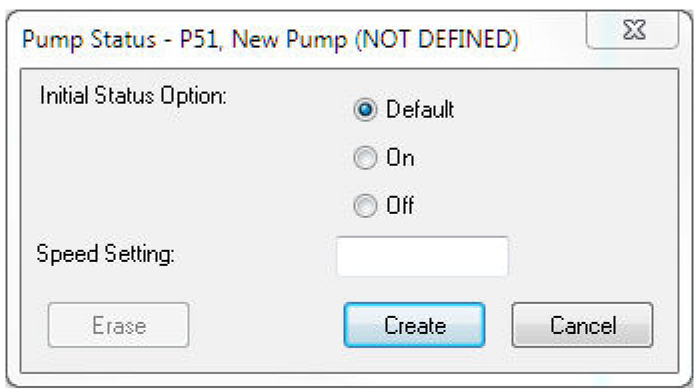
- **On Setting** - The volume or height of water that turns the pump on. This setting is valid for " By Volume" and " By Level" control options.
- **Off Setting** - The volume or height of water that turns the pump off. This setting is valid for " By Volume" and " By Level" control options.
- **Static Pump Flow** - This variable is used for " By Discharge " control method, and is used during steady state simulations only. It represents the actual amount of sewage that the user wants to be pumped during steady state simulations.
- **Base Pump Flow** - This variable is required for " By Discharge " control during EPS/Dynamic simulations alone, and represents the base flow to be multiplied by the pattern factors to determine the actual flow to be lifted at the time step being analyzed. For example, if the pattern factor at hour two is 1.2 and base flow for the pump is 5 cfs, the actual pump discharge at hour two would be 6 cfs (i.e., 5 cfs* 1.2).
- **Minimum Speed** - Represents minimum speed of a pump, and is required for the "By Discharge" and "By Inflow" control options. The default setting is zero.
- **Maximum Speed** - Represents the maximum speed that a pump could sustain, and is required for the "By Discharge" and "By Inflow" control options.
- **Pattern ID** - Pattern is required for the "By Discharge" and "By Time" control options. The former requires flow pattern, while the latter control option needs speed pattern.
- **Erase** - Delete the current control setting.
- **Update** - Change the control setting with the new criteria specified. Once a control has been created, click on the Update button to place the control statement into the Control Rules box and exit the dialog box.

- **Cancel** - Quit the command with saving changes (so long as update was not pressed first).

The procedure has to be repeated for all pumps in the collection system. Pump controls could also be assigned to multiple pumps at a time using the group editing feature.

Initial Status and Setting

Speed and initial on-off status of pumps in the collection system can be specified by first selecting the, and then selecting **Initial Status** command from Tools Icon  drop down box located the top of the Attribute Browser. The default pump speed setting is one (pump speed ratio of 1). The initial pump status is overwritten by the operational controls during an extended period simulation. The following dialog box is activated once the "Initial Status" button is clicked, and could be edited to reflect the desired settings.



Saving Control Sets

After a simple control is created, it is stored as a record in the Standard Control database (see DBEditor). This database is by default, the *active* database for the control set. You may wish to have different controls applied to the same pump depending on which scenario is being analyzed. This is done by cloning the *active* control set to a new control set which makes an exact copy of the *active* database. By assigning a unique name to this new control set, the user can now assign that control set to any scenario. As that scenario is activated in the future, it will overwrite any pre-existing records in the *active* control set with the contents of the saved control set database.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Critical Depth

The Froude number , NF, is defined as the ratio of inertial forces to gravity forces: $N_F = \frac{v}{\sqrt{g\frac{A}{T}}}$ (6) $v = \frac{Q}{A}$ (7) where

v = velocity, ft/s (m/s)

T = width of the free surface of the water at the top of the pipe, ft (m) g = gravitational constant, ft2/s (m2/s)

When the Froude number is equal to 1.0, the flow is called critical flow. When NF < 1.0, the flow is subcritical and when NF > 1.0, the flow is supercritical. The depth corresponding to the minimum specific energy is therefore called the critical depth yc. The resulting criterion for determining the critical depth using Equations (6) and (7) is: $\frac{Q^2W}{gA^3} = 1$ (8) It is also important to distinguish between the following types of flows:

- **Full Flow**: represents the flow when the pipe is full, i.e. where the wetted perimeter is equal to the entire pipe perimeter.
- **Maximum Flow**: represents the maximum theoretical flow occurring in a pipe when the normal depth is equal to 0.938 x Diameter. Any increase in depth will decrease the flow, which is why the full flow is less than the maximum discharge for a circular pipe.
- **Analysis Flow**: represents the flow occurring in a pipe as derived from the analysis criteria curve (See CRITERIA CURVES section).
- **Design Flow**: represents the flow occurring in a pipe as derived from the design criteria curve (See CRITERIA CURVES section).

The computed results for pipes include:

- The total flow and velocity
- The full, maximum, design and analysis flows
- The excess full, design and analysis capacities

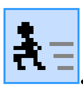
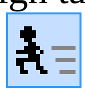
- The normal and critical depths
- The actual, design, and analysis d/D ratios

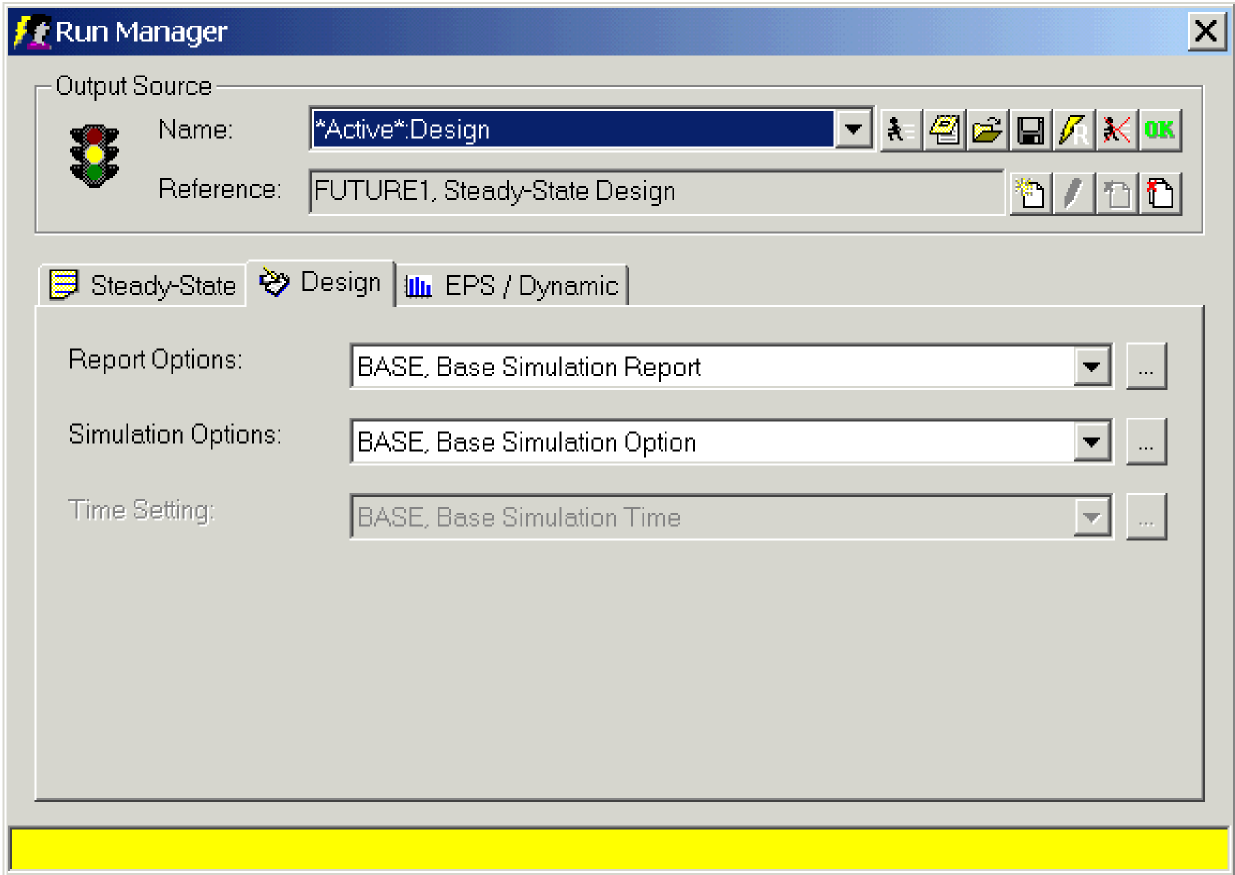
Design Simulations

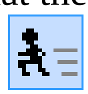
A design simulation performs the same hydraulic analysis as a steady state analysis. It is an instantaneous "snap shot" of the collection system that evaluates sewage flows against analysis and design criteria.

A design simulation will evaluate the collection system against design and analysis criteria that is provided in the form of curves (for circular pipes) or design tables (for non-circular pipes). The steady state design simulation goes one step further than a steady state analysis in that recommended pipes are calculated for parallel and replacement pipes where the pipe d/D value exceeds the analysis criteria, and/or the velocity criteria are not satisfied. The user can also generate costs for these replacement facilities by establishing Replacement and Parallel cost values. Click here to learn more about using the Run Manager.

Running a Steady State Model

To run a steady state model, go to the **Tools** menu and select **Run Manager** . Once there, select the Design tab, verify report and simulation options, and click the Run Manager icon  to begin the simulation.



Upon successful completion of a steady state simulation, results are stored in the *Active*.Steady Output Source. Use the Active Output pull down box to ensure that the *Active*.Steady output is enabled prior to using the Report Manager  to open and display the results of the simulation. Click here to learn more about a steady state model result report.

InfoSewer™

Modeling Made Easy

InfoSewer™
Modeling Made Easy

Design Report

The following variables are reported from the Output Report Manager. Note that the design status can be used to color code the map using the map display.

ID	From ID	To ID	Length (ft)	Design Flow (cfs)	Diameter (in)	slope	d/D Ratio	q/Q Ratio	v/V Ratio	Velocity (ft/s)	From Invert (ft)	To Invert (ft)

U/S Manhole Invert (ft)	D/S Manhole Invert (ft)	U/S Cover Depth	D/S Cover Depth (ft)	Cost ($)	Design Status

ID: Pipe ID

From ID: Upstream node ID

To ID: Downstream node ID

Length: Pipe Length (it is an input) **Design Flow:** Actual flow through the pipe computed from average loads and peaking information **Diameter:** Diameter designed for the pipe **Slope:** Slope designed for the pipe

d/D ratio: flow depth to diameter ratio **q/Q:** design flow to full flow ratio **v/V:** actual velocity to full flow velocity ratio **Velocity:** Velocity through the pipe **From Invert:** From Invert Elevation of the pipe **To Invert:** To invert elevation of the pipe **U/S Manhole Invert:** Invert elevation of upstream manhole **D/S Manhole Invert:** Invert elevation of downstream manhole **U/S Cover Depth:** Cover depth at upstream end of the pipe measured from U/S manhole rim elevation **D/S Cover Depth:** Cover depth at downstream end of the pipe measured from D/S manhole rim elevation **Cost:** Estimated pipe cost estimated based on designed cover depth, pipe length, and specified cost information **Design Status:** Indicates design status for the pipe. Options are:

- Successful -> if the pipe size and slope that meet all the design criteria is determined
- Not designed -> pipe is not designed due to design failure of upstream pipe(s)

- Failed: High d/D and Low Velocity

- Failed: High d/D and High Velocity

- Failed: High d/D
- Failed: Low Velocity
- Failed: High Velocity
- Failed: High Crown Drop

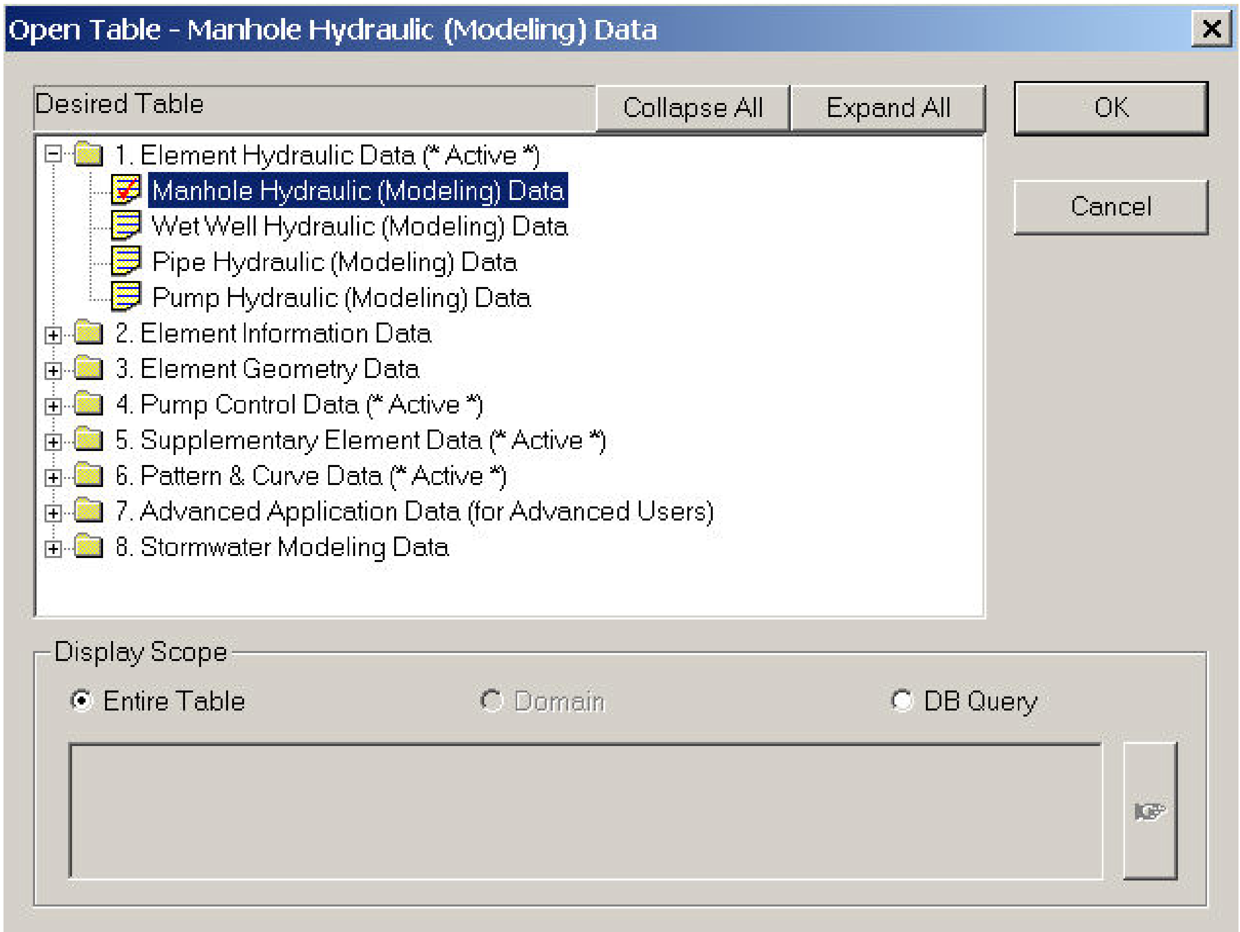
DB Tables

Use this to launch the InfoSewer DB Editor dialog box. All the InfoSewer Project related data are stored in the DB Tables and the DB Editor provides a means to select, view, edit and maintain this data. Click here for more information.

All the InfoSewer project data including data related to the different InfoSewerdata elements such as pipes, pumps, manholes and wetwells are stored in the DB Tables. The Database Editor allows the user to open any InfoSewer database and edit/modify or delete user input fields (except ID). To access the DB Tables, from the **InfoSewer Control Center ->** InfoSewerbutton **->** **Edit** menu, select **DB Editor** to see the Open Table dialog box below.

From the **Open Table** dialog box, you can see that InfoSewer stores database tables in eight separate folders. After selecting the desired table from under the relevant folder, you have the option of choosing a display scope found at the bottom of the dialog box to open the desired records of the database. Once this is done, click the OK button to open the database table.

Click on any portion for more details:



Other Related Topics - Advanced Application Data Tables, Pump Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Pattern & Curve Data Tables, Supplementary Data Tables, DB Editor, DB Tables, Stormwater Modeling Data

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Advanced Force Main Network Solution

The Force Main Network Solution allows the simulation of multiple upstream and downstream force mains entering and leaving one chamber junction during an Extended Period Dynamic Simulation or EPS solution in Sewer. All of the force mains, pumps, wet wells and force main chamber junctions that are connected are considered as one force main network in the EPS solution. You can have more than one force main network in a large Sewer model separated by gravity pipes and loading manholes. The individual force main networks are solved iteratively with different upstream head and downstream tail manholes which connect the force main network(s) to the rest of the network.

A force main network consists of the following elements:

- Wet well
- Pump
- Junction Chamber
- Head Manhole where flow from other parts of the sewer system enters the force main network
- Tail manhole where the flow leaves the force main network.

The head and tail manhole for one force main network is determined by the program based on the geometry of the network. The force main network starts at a wet well, includes the pumps connecting the wet well to the force main links and also includes the actual force main links and force main connecting junction chambers. You can also connect a force main to the gravity mains without an intermediate wet well and pump(s).

The boundary conditions of the force main network are:

- Water heads at the wet wells which vary according to the inflow from the upstream sections of the sewer network and outflow to the force main network
- Water head at the tail manholes which are calculated as the maximum discharge head (invert + diameter) of all the force mains that end at

that manhole. Water entering the tail manholes will be routed downstream after the force main network flows are calculated.

For example, assuming there are n1 wet wells, n2 head manholes, n3 tail manholes, n4 junction chambers and p1 pumps and p2 force mains, the program must solve the network hydraulics to get n2+n4 water head values and p1+p2 flow values iteratively using the Newton-Raphson method. The solution iterates until the mass and energy of the force main network is in balance.

The hydraulic equations used in the solution are:

- Head/Flow relationship of the force mains and pumps (p1+p2 equations)
- Mass balance at head nodes and junction chambers (n2+n4 equations)

For head nodes, water entering the network from other sections of the sewer system must equal the flow sum of force mains that connect to it:

$$\sum_{i \in G_v} Q_i + \sum_{j \in G_f} Q_j = 0$$

Where Q = Flow; G_v = group of gravity pipes connecting to the head manhole; and G_f = group of force mains connecting to the head manhole. The sum of the gravity flow into the wet well or head manholes is balanced by the sum or flow out of the force main network in the force main pipes.

For junction chambers, which are connected to only force main pipes:

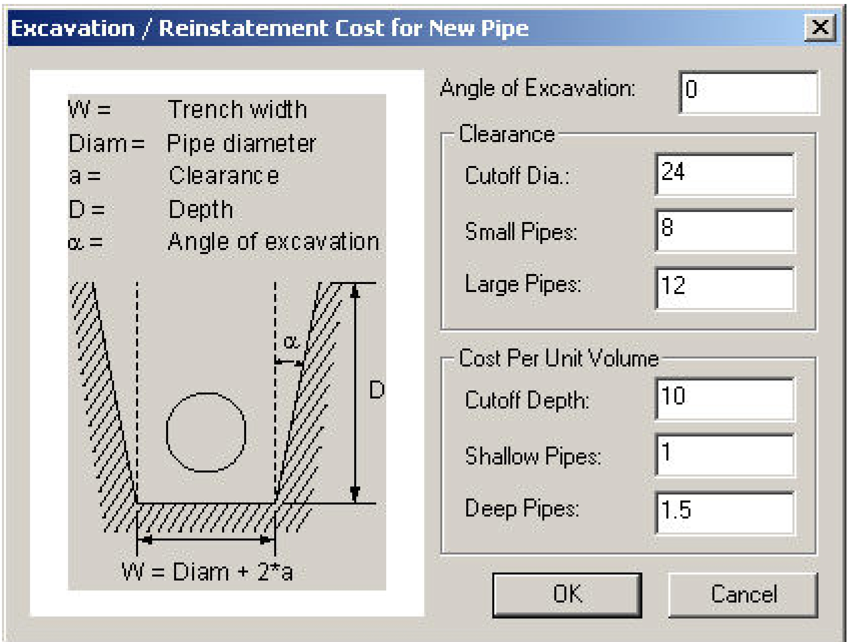
$$\sum_{j \in G_f} Q_j = 0$$

For force mains, Hazen-Williams equation describes the flow/head loss relationship within a force main. The flow out of and the flow into the junction chamber is in balance. The head at the junction manhole is iterated until the flows are in balance.

For pumps that are neither Inflow Control nor Discharge Control, the pump curve is used to estimate the flow and head gain relationship within a pump. For Inflow Control and Discharge Control pump, pump flow as control values are fixed and the equation $Q = Q_{control}$, where $Q_{control}$ is the controlling pump value. For such situations, the pump is actually modeled as variable speed pump and pump speed will be calculated with Newton-Raphson method to achieve the flow control objective.

Define Cost

This dialog aids in the development of costs for pipe size and construction parameters using the dialog below.



Angle of Excavation: Refers to slope of the trench excavation angle which depends on soil type. This value defined here may be utilized for all pipes in the system for which angle of excavation is not defined locally.

Clearance Cutoff Diameter: Small size pipes may require small clearance as compared to large size pipes. The clearance cutoff diameter refers to pipe size that distinguishes pipe size beyond which a different clearance has to be used. Default diameter is 24 in and 600 mm in US customary and SI units, respectively. Pipes with a diameter less than the cutoff diameter will use the small pipes clearance value, while pipes with a diameter greater than the cutoff diameter will use the large pipes clearance value

Cost Cutoff Depth: Unit costs of excavation and reinstatement may vary depending on depth. The cutoff depth refers to the depth beyond which a different cost rate needs to be used. The unit is ft and meter in US customary and SI systems, respectively. The default cutoff depth is 10 ft and 3 m in US customary and SI systems, respectively. Pipes at a cover depth less than the cutoff depth will use the shallow pipes value, while pipes at a cover depth greater than the cutoff depth will use the deep pipes value.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

EPS/Dynamic Modeling

An extended period and dynamic simulation (EPS/Dynamic) is any modeling run over a duration greater than a single hydraulic timestep (steady state). During dynamic simulations, the model uses the Muskingum-Cunge's flow routing algorithm to solve the St. Venant's equations, whereas the EPS approach analyses the system using a sequence of state state models, discretized at smaller simulation time intervals. The user is provided many tools in H2OMAP Sewer that allow for the customization of an EPS/Dynamic simulation. Click on any of the links below to learn more.

EPS/Dynamic Simulation - A hydraulic simulation that evaluates the collection system at set intervals as specified through the simulation time command. An EPS/Dynamic model determines the d/D ratio, flow, and velocity for each pipe in the system.

- Simulation Time - Control the timesteps for hydraulic analyses and the starting clock time.

Controls

Controls allow the user to create an EPS/Dynamic model that is an "actual" computer representation of field conditions. Through control statements, the user can "turn on" and "turn off" data elements as other system criteria are satisfied.

- Simple Controls - Basic "If, Then" statements that enable one action to occur for only one event (ex. pump on when wet well is full).

Water Quality - H2OMAP Sewer provides five options for sewage quality analysis in conjunction with a hydraulic simulation. The purpose of the quality model is to simulate pollutant loading and buildup, washoff, as well as individual domestic, commercial and industrial contributions, and transport.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

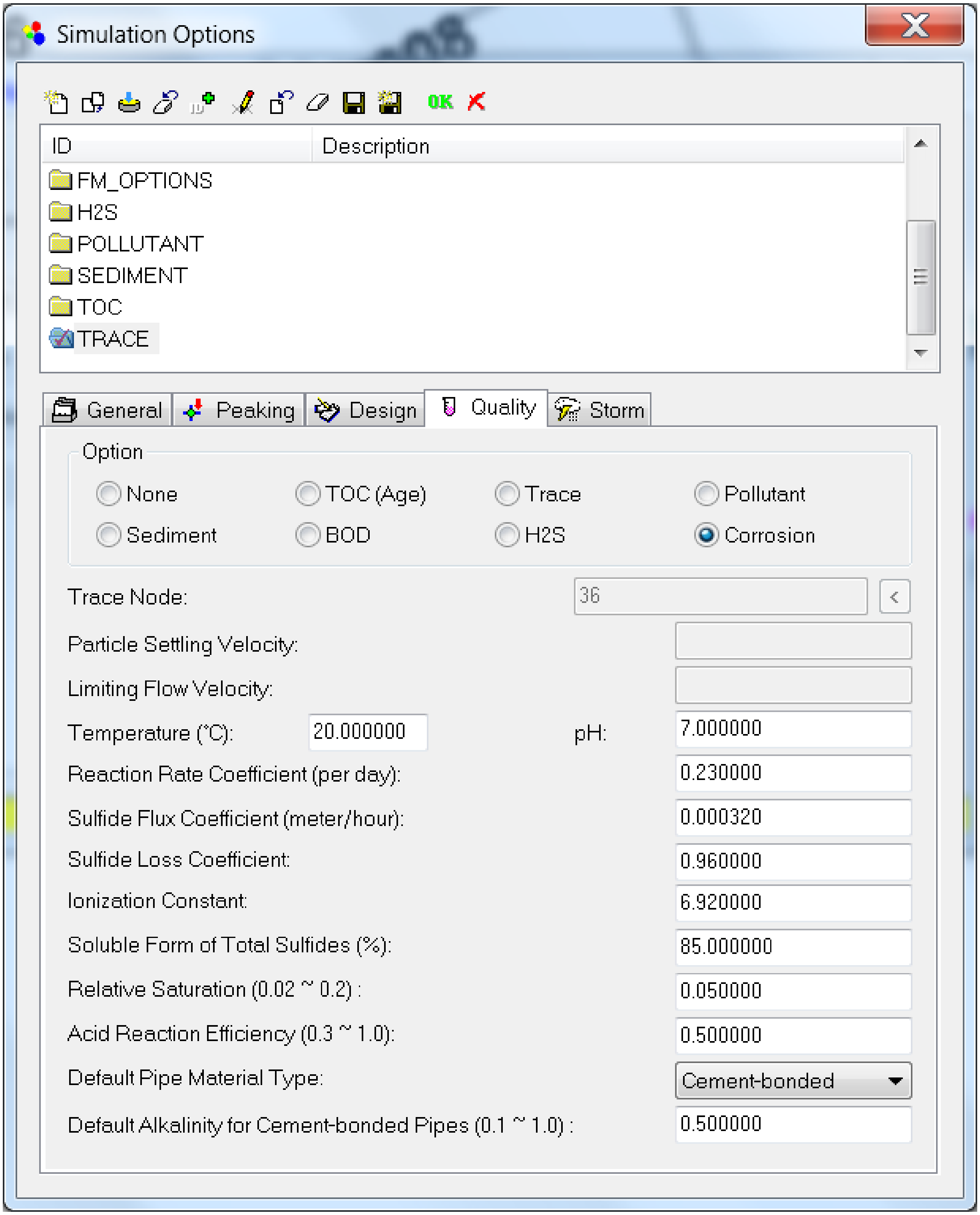
Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Water Quality

InfoSewer provides seven options for sewage quality analysis in conjunction with a hydraulic simulation. The purpose of the quality model is to simulate loading, buildup and washoff, and transport of various water quality constituents resulting from domestic, commercial, and industrial wastes. Water Quality options include None, TOC(Age), Trace, Pollutant, Sediment, BOD, H2S and Corrosion.

These options are available only for EPS/Dynamic runs, and may be chosen on the Quality panel from the Simulation Options dialog box given below.



Time of Concentration

Time of concentration (TOC) is the time spent by a sewage flow parcel in the network (i.e., the time of flow in the sewerage system). New sewage entering the network from loading manholes is assigned age of zero. When the model is run under constant hydraulic conditions, the age of sewage at any location in the network can be interpreted as the time of travel to that location. This parameter is useful to address important water quality and safety issues such as generation of sulfide that may occur in a sanitary sewer system, and its subsequent results such as corrosion and odor issues.

Time of concentration analysis could be performed by clicking TOC (Age) on the quality panel of the simulation options dialog box. No additional input parameter is required.

Source Tracing

Source tracing tracks over time what percent of sewage reaching any pipe or manhole in the network had its origin at a particular source node. The source node can be any manhole in the network, including wet-wells. Source tracing is very useful in sewer collection systems, and could be used for (1) tracking changes in sewage flow contribution (and associated constituents) over space and time; (2) predicting impact of industrial and commercial waste discharges on performance of wastewater treatment plants; (3) determining contaminant level that causes a wastewater treatment plant to be in violation of its discharge permits; (4) and developing appropriate user charges based on wasteloads and level of contaminant.

Source Tracing simulation is initiated by clicking on the Trace tab of the quality panel (shown above). As soon as the trace tab is selected, trace node specification tab is activated enabling the user to select the desired source Node. Only one source node can be selected for a given simulation. InfoSewer treats the source node as a constant source of a non-reacting constituent that enters the network with a concentration of 100% throughout the simulation period. Temporal trend of the percentage of the sewage flow originating from the source node is provided as output for pipes and manholes draining the source node.

Pollutant Transport

InfoSewercan also track the movement of conservative constituents (e.g., chloride, bromide, sulfate, boron, sorbed trace metals) flowing through the network over time. The dynamic quality simulation model is predicated on conservation of mass coupled with reaction kinetics and consists essentially of three processes: advection in pipes, mixing at sewer manholes and wet-wells, and kinetic reaction mechanism. The ability to model pollutant transport in sewer collection systems is useful in determining the amount of pollutant that is transported to the wastewater treatment plant and assessing impact on the receiving waters.

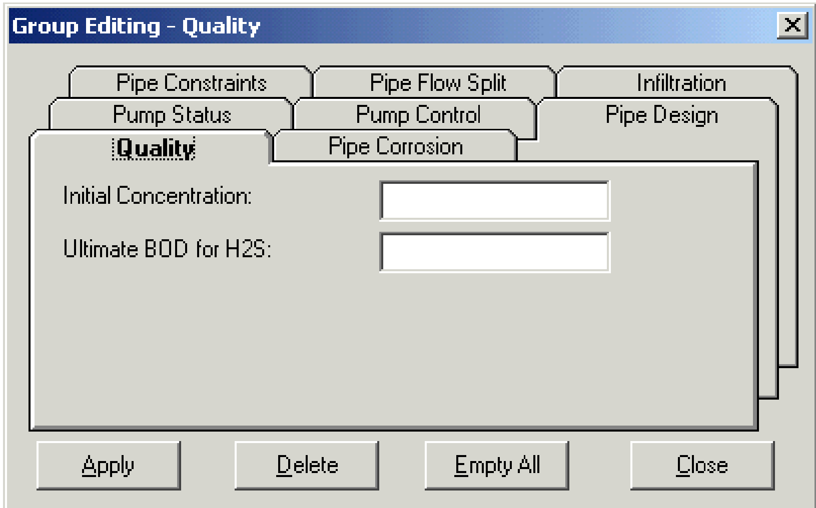
Using InfoSewer, the user can model transport of conservative pollutants by first clicking on the Pollutant tab of the quality dialog box given below. Unlike the source tracing analysis described above, source nodes could be multiple for pollutant transport simulation. Initial concentration of the source nodes could also vary from a source node to another.

Initial concentrations for the desired source node(s) can be specified using either of the following two ways.

In the first approach, the user has to select a source node (only one node at a time), and then has to click on the Quality icon ( ) located at the top of the Attribute Browser. The quality icon initiates the following dialog box through which the initial concentration (in mg/l) could be edited. Note that ultimate BOD data is not required for pollutant transport modeling. This process needs to be repeated for all source nodes.

The second approach involves creating domain of the source nodes. Once the domain is created, the user has to perform the following tasks.

- Click on edit from the Menu bar and then on Group Editing on Domain ( ). As an alternative, Group Editing on Domain could be directly accessed from the tool bar.
- Choose quality from the Group Editing dialog box. Then specify the initial concentration in mg/l. Performing these two steps implies that all nodes in the domain are source nodes, and that they all have the same initial concentration. Note that ultimate BOD data is not required for pollutant transport modeling.



Sediment Transport

Sanitary sewer systems can carry substantial loads of suspended solids (waste solids) which can accumulate and cause blockages thereby impairing the hydraulic capacity of the sewer pipes (by restricting their flow area and increasing the bed friction resistance). InfoSewer can simulate the transport and gravitational settling of sediments (total suspended solids including grit) over time throughout the sewer collection system under varying hydraulic conditions. As long as flow velocity exceeds the critical/terminal velocity, InfoSewer assumes that the sewage flow has the capacity to transport all incoming sediments. Deposited sediment particles are also assumed to be scoured and transported downstream when velocity of the sewage flow exceeds the terminal velocity. Settling starts when flow velocity falls below the critical velocity. In the model, transport of the sediment particles is governed by advection implying that the particles are transported at local flow velocity.

Sediment transport modeling using InfoSewer requires only few inputs, namely limiting flow velocity, particle settling velocity, and source node(s) and initial sediment concentrations (in mg/l) at the source nodes.

In order to specify the first two inputs (i.e., limiting flow velocity and particle settling velocity), the user should first select 

from the quality tab which in turn activates the editing tabs for particle settling velocity and limiting flow velocity. Specification of source node(s) and its/their initial concentration is similar to the method described above in relation to pollutant transport. The default values used by the model for limiting flow velocity and particle settling velocity are 2 ft/s and 0.1 ft/s, respectively. User specified values over rid these default figures.

Sediment deposition (in kg) in pipes and sediment concentration (in mg/l) at manholes, wet wells, and outlets are the outputs reported following successful simulation of sediment transport for a collection system.

Note that the ultimate BOD data is not required for modeling of sediment transport and deposition.

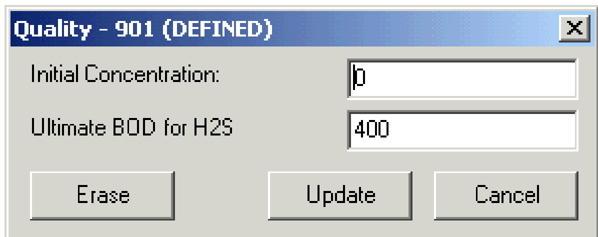
Biochemical Oxygen Demand

Biochemical Oxygen Demand (BOD) is the most widely used parameter of organic pollution in sanitary sewer systems. The ability to model BOD is of great importance in wastewater engineering to assist wastewater utilities in (1) estimating the quantity of oxygen required to biologically stabilize the organic matter present; (2) determining the size of wastewater treatment facilities; (3) evaluating the efficiency of the treatment process; and (4) ensuring compliance with wastewater discharge permits. InfoSewer models the rate of BOD oxidation (exertion) using first-order kinetics with the rate of oxygen utilization being proportional to the difference between the amount of oxygen used and the ultimate BOD (UBOD).

UBOD at source node(s), average daily temperature (in ^{o}C) of the study area at the simulation day, and the first-order reaction rate constant corresponding to temperature of 20^{o}C are the input parameters required by the model to simulate BOD for the sewer collection system. Internally, InfoSewerdetermines and uses appropriate value of the reaction rate constant depending on the user-specified temperature and the kinetic rate constant corresponding to temperature of 20^{o}C. The default values used by InfoSewer for first-order reaction rate constant and temperature are 0.23/day and 20^{o}C, respectively.

Average daily temperature and first-order reaction rate constant are specified by first clicking on 

which automatically activates the editing tabs for the two input parameters. Source nodes and their corresponding UBOD could be provided following the procedure given above in connection with pollutant transport. Here it should be noted that, the initial concentration data is what is regarded as UBOD during BOD modeling. The potentially confusing additional "Ultimate BOD for H2S" data is not required during BOD modeling. This information is used for hydrogen sulfide (H2S) modeling and corrosion prediction.



Note: Source nodes can only be loading manholes and/or wet wells during source tracing analysis, and modeling of pollutant transport, sediment transport, and BOD.

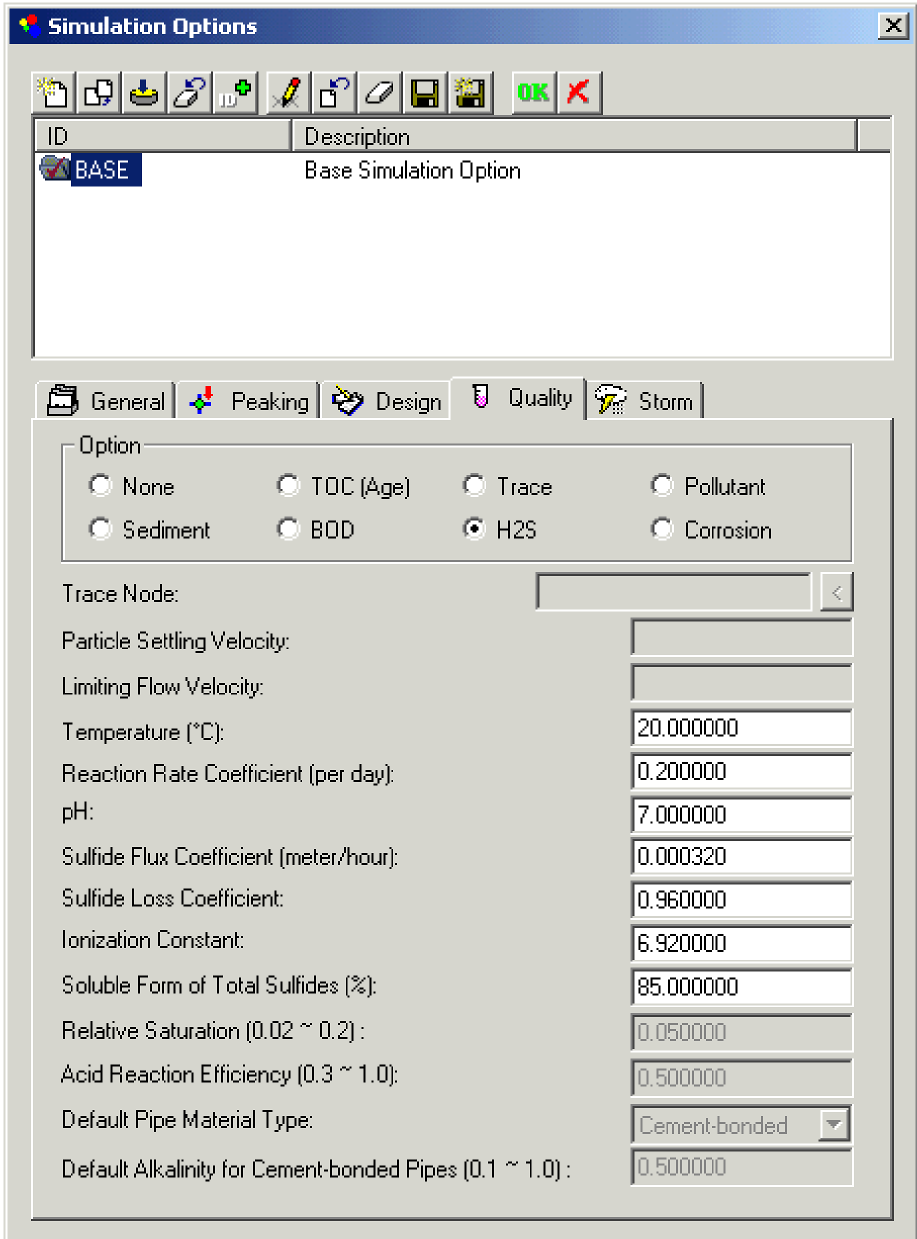
Hydrogen Sulfide

The control of odorous gases and the corrosion of sewers are the two most important problems in operating wastewater collection systems. H_2S is the most commonly known and prevalent odorous gas associated with domestic wastewater collection and treatment systems. *H_2S modeling* for InfoSewer suite gives wastewater engineers a powerful Operations and Maintenance (O&M) tool to readily model and analyze entire sewer collection systems for sulfide generation and corrosion potential under varying conditions anticipated throughout the life of their systems. *H_2S Detector* predicts sulfide buildup in sewer collection systems for gravity sewers, force mains, and wet wells using the Pomeroy-Parkhurst equations. This unique and powerful tool enables wastewater utilities to pinpoint odor and corrosion problems, develop effective monitoring programs, alert plant operators and sewer maintenance workers to potential danger and the need to observe safety practices, and evaluate and implement effective control system such as aeration, chlorination, and mechanical cleaning.

Modeling of hydrogen sulfide requires minimal input data from the user. The required data include:

- average daily temperature for the region (in ^{o}C).
- reaction rate coefficient (per day) which was described above in relation to BOD modeling.
- pH of the wastewater. The normal pH range of municipal wastewater is 6.0 to 8.0.
- effective sulfide flux coefficient for sulfide generation by the slime layer in gravity sewers (meter/hour). For conservative analysis (i.e., observed sulfide buildup generally less than predicted), the suggested values of this parameter is 0.00032.
- a dimensionless coefficient to account for sulfide losses by oxidation and escape to atmosphere. For conservative analysis (i.e., observed sulfide buildup generally less than predicted), the suggested values of this parameter is 0.64. For moderately conservative analysis a value of 0.96 is suggested.
- logarithmic ionization constant for hydrogen sulfide (unit less), a function of temperature and specific electrical conductance of the waste water. Its value generally varies from 6.67 (at a temperature of $40^{o}C$ and specific electrical conductance of 50, 000 micromhos/cm) to 7.74 (at a temperature of $10^{o}C$ and specific electrical conductance of 0 micromhos/cm).
- percent of total sulfides that occur in the soluble (dissolved) form for the wastewater, most frequently known to vary from 70 to 90 percent.

The default values used by the model for these inputs are shown in the following dialog box. In addition to the listed data, the user has to supply initial concentration of total sulfides and ultimate BOD of the wastewater. These two variables could be assigned using the quality tab available at the top of the attribute browser, for one source node at a time, or using the " Group Edit on Domain" feature that enables simultaneous assignment of the two variables for all source nodes in the domain.



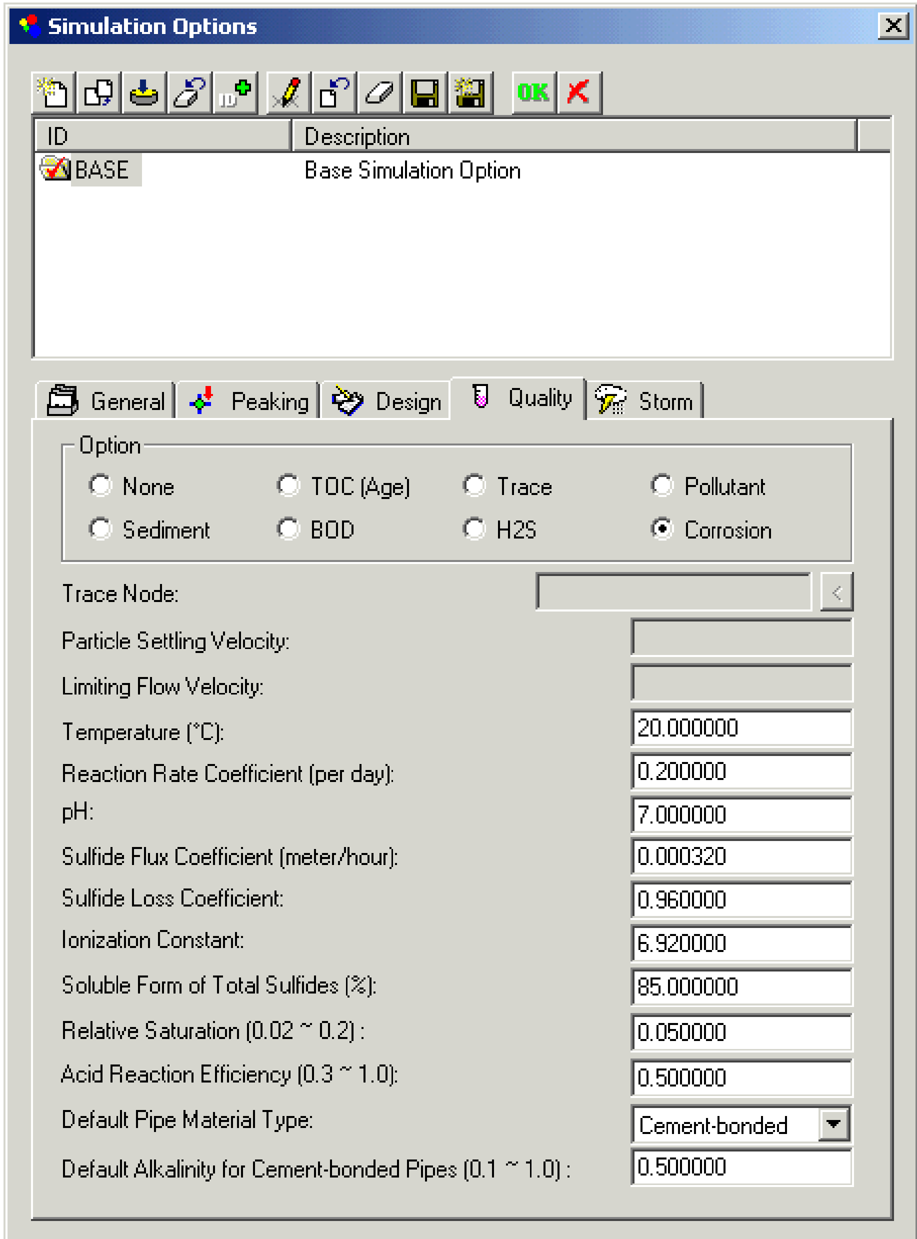
Corrosion

Corrosion in sewers causes large annual capital expenditures. Corroded sewer pipes may allow greater inflow and infiltration into the collection system causing surcharges and overflows, ultimately calling for premature replacement of sewer pipes. Corrosion of unprotected concrete or metal surfaces is primarily due to the production of sulfuric acid in sewer systems through oxidation of hydrogen sulfide gas by bacterial action on the exposed surfaces under aerobic conditions. *Corrosion Predictor*^{TM} extension for

InfoSewer suite helps wastewater engineers to predict the rate of corrosion of the sewer pipes in their collection systems under varying environmental and hydraulic conditions enabling them to pinpoint pipes that are subjected to severe corrosion problems.

The rate of pipe corrosion depends upon the rate of sulfuric acid production, the amount of the produced sulfuric acid that reacts with the pipe material, and the alkalinity of the pipe material. Sulfuric acid production is related to the amount of hydrogen sulfide gas that escapes to the sewer atmosphere, which in turn depends on a number of environmental and hydraulic conditions previously described in relation to hydrogen sulfide buildup, partitioning, and release. *Corrosion Predictor*^{TM} determines rate of corrosion for cementitious (cement-bonded) materials and ferrous materials in gravity sewers using Pomeroy equation. In force mains, where the lines flow full, there is generally no internal corrosion since generation of sulfuric acid is prevented.

Some of the parameters required during prediction of corrosion rates are shown in the following dialog box.

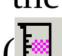
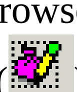


In order to predict rate of corrosion in sewer systems using *Corrosion Predictor*^{TM} , one has to first model the build up, partitioning, and emission of hydrogen sulfide. Therefore, all the data listed and described above in relation to hydrogen sulfide modeling is essential for predicting corrosion rate in sewer systems. This data include

- initial concentration of total sulfides at source manholes.
- ultimate BOD of the wastewater for every source node.

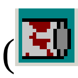
- average daily temperature for the region (in ^{o}C).
- reaction rate coefficient (per day) which was described above in relation to BOD modeling.
- pH of the wastewater. The normal pH range of municipal wastewater is 6.0 to 8.0.
- effective sulfide flux coefficient for sulfide generation by the slime layer in gravity sewers (meter/hour). For conservative analysis (i.e., observed sulfide buildup generally less than predicted), the suggested values of this parameter is 0.00032.
- a dimensionless coefficient to account for sulfide losses by oxidation and escape to atmosphere. For conservative analysis (i.e., observed sulfide buildup generally less than predicted), the suggested values of this parameter is 0.64. For moderately conservative analysis a value of 0.96 is suggested.
- logarithmic ionization constant for hydrogen sulfide (unit less), a function of temperature and specific electrical conductance of the waste water. Its value generally varies from 6.67 (at a temperature of 40^{o}C and specific electrical conductance of 50, 000 micromhos/cm) to 7.74 (at a temperature of 10^{o}C and specific electrical conductance of 0 micromhos/cm).
- percent of total sulfides that occur in the soluble (dissolved) form for the wastewater, most frequently known to vary from 70 to 90 percent.

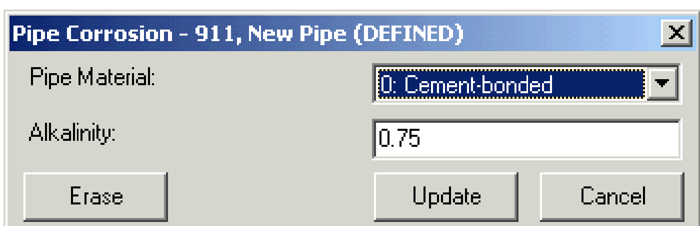
Note: Since corrosion is a result of long term environmental effects, the above described data (e.g. temperature, ultimate BOD, initial concentration of total sulfides, pH, etc) should be assigned accounting for their seasonal fluctuations.

Initial concentration of total sulfides and ultimate BOD of the wastewater for every source node could be supplied using the quality tab () located at the top of the attribute browser, or using the group editing features (i.e., group editing on selection () and group editing on domain ()) for a group of source nodes simultaneously.

In addition to the above described inputs, corrosion prediction requires specification of the following data.

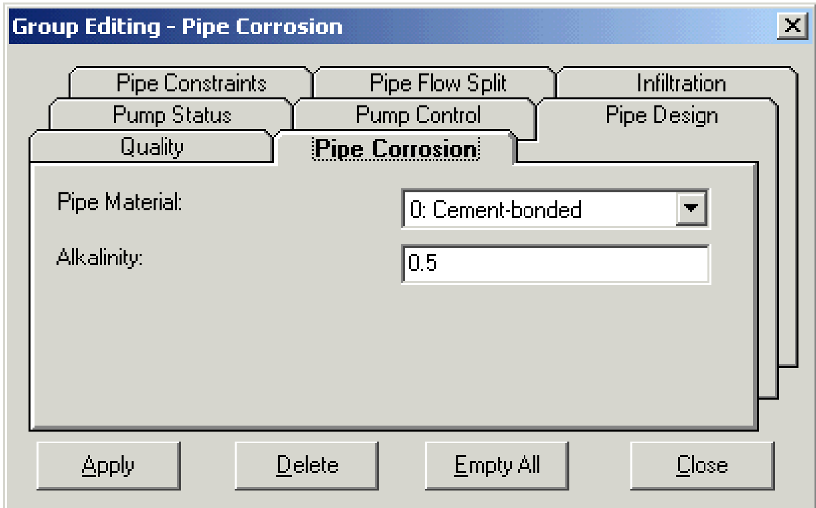
- pipe material (i.e., cement-bonded or ferrous) for gravity pipes.
- alkalinity of the pipe material expressed as $CaCO_3$ equivalents (for cement-bonded materials only). This value approximately ranges from 0.18 to 0.23 for granitic aggregate concrete, 0.9 for calcareous aggregate, 0.4 for mortar linings, and 0.5 for asbestos cement.
- coefficient of efficiency for acid reaction considering the estimated fraction of acid remaining on the wall. This value could be as low as 0.3 and will approach 1.0 for a complete acid reaction.
- relative saturation of H_2S in the air compared to equilibrium concentration (typically 2 to 20 percent), expressed as decimal fraction.

Pipe material type and the associated alkalinity could be specified for one pipe at a time using the corrosion tab () located at the top corner of the attribute browser. To use this option, select the pipe, click on the corrosion tab, and then specify the material type and the alkalinity (for cement-bonded materials only) on the initialed dialog box (shown below).



These two input data could also be specified for a group of pipes simultaneously using either of the group editing features (i.e., group editing on selection () and group editing on domain ()). To use the group editing on domain feature, for example, one has to first create a domain.

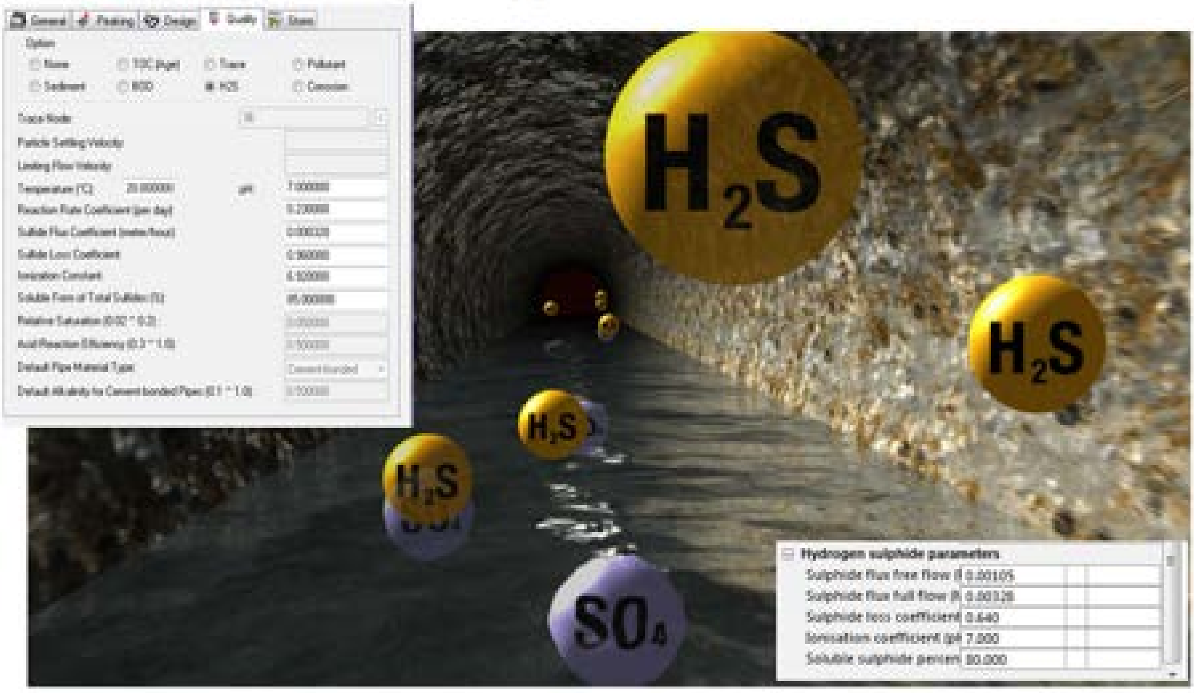
Once domain is created, the  icon should be clicked. This process initiates the group editing dialog box. On this dialog box, click on the "pipe Corrosion" option, and then specify the material type and the alkalinity (for cement-bonded materials only), as shown below.



The default values used by the *Corrosion Predictor*^{TM} for pipe material, alkalinity of pipes made from cement-bonded materials, coefficient of acid reaction efficiency, and the relative saturation of H_2S are: cement-bonded, 0.5, 0.5, and 0.05, respectively.

Note: The model uses the default material type and the default alkalinity for all gravity pipes in the system for which material type and alkalinity are not specified.

On the upper walls of the sewers, microbes take up the H_2S and oxidise it with air to form sulfuric acid, H_2SO_4. This is extremely powerful in corroding concrete, which is the most common material used in large sewer pipes.



The production of hydrogen sulfide (H2S) from sulfate (SO4) in the water, and its subsequent release to sewer air and oxidation to sulfuric acid at concrete surface corrodes the concrete pipes. University of Queensland

InfoSewer™

Modeling Made Easy

Introduction to Stormwater Modeling

Determination of stormwater or runoff resulting from a single or multiple rainfall event(s) is the most important component of stormwater management, and is commonly known as rainfall-runoff modeling or stormwater modeling. H2OMAP Sewer Pro adds comprehensive rainfall-runoff and infiltration/inflow modeling capability on top of the sanitary sewer simulation features provided by H2OMAP Sewer, equipping the industry with a single powerful tool that can be used for planning, design, and operational management of sanitary sewer systems, storm sewer systems, and combined sewer systems.

H2OMAP Sewer Pro shares all the functionality of H2OMAP Sewer including the methodologies used to model the physical and the non-physical components of a collection system, steady state and dynamic analysis of sanitary loads, steady state design, dynamic routing of flows through the collection system, calculation of HGLs, and water quality modeling. This section presents the theory pertinent to H2OMAP Sewer Pro.

There are various rainfall-runoff-modeling techniques. The choice of the method to use depends, among other considerations, on the type of analysis (e.g., steady state or dynamic simulation) and the information required (e.g., peak flow or complete hydrograph). Peak flow information is appropriate for steady state simulation of sewer systems whereas knowledge of the complete hydrograph is essential for dynamic simulations.

For estimation of peak flow during steady state simulations, H2OMAP Sewer Pro applies the Rational method, which is a very popular tool of choice used by many practicing engineers across the globe. For dynamic simulation, complete hydrographs are derived based on the practical and highly effective unit hydrograph theory. The unit hydrograph can be observed or synthetic. This method allows the entire hydrograph (including peak and volume) resulting from an actual or design storm event to be adequately simulated.

The links given above and/or below may be used to access details (theory as well as working mechanism) on topics of special interest.

- Steady State Modeling of Stormwater
- Dynamic (EPS) Modeling of Stormwater
- Theory of Stormwater Modeling

Note: Stormwater modeling feature is available for **H2OMAP Sewer Pro** only.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

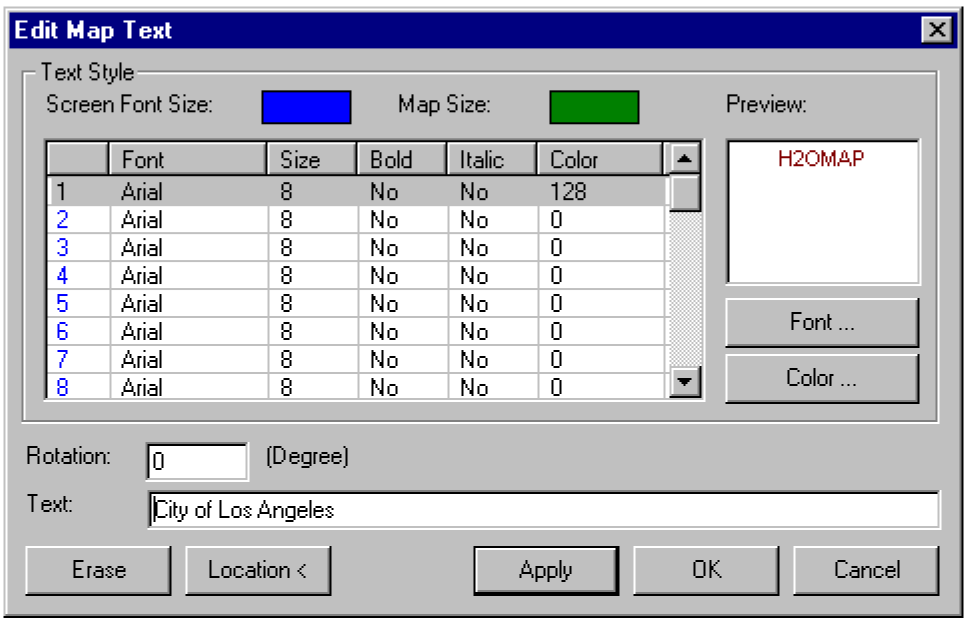
More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Edit Text

The Edit Text feature of InfoSewer allows the user to place text anywhere in the graphics display. There are two types of text the user can place. The first is "Screen Font" text which varies in size while the second is "Map Size" text that, once placed, is fixed in height. Text can be edited and moved at any time using the Edit Text command.

Creating Text

From the **Edit** menu, select **Edit Text**. Once the command is initialized, the following dialog box will appear:



- **Screen Font Size** - Use a screen font (fonts with a blue number) to have text size adjusted in relation to the zoom factor. For example, if an 8 pt. font is selected, no matter the zoom scale, the font will always remain 8 pt. in relation to the zoom scale.
- **Map Size** - Using the scroll bar to arrow down, the user is able to see that the second set of font "layers" have a green number. This means that these font layers have their text tied to a map size (in units). For example, if an 8 pt. font is used, the text size will remain

constant, meaning that as the user zooms in and out, the font will grow larger and smaller.

- **Preview/Font/Color** - Use the font and color buttons to change the appearance of the selected layer. By default there are 20 Screen Fonts (blue numbers) and 20 Map Fonts (green colors).
- **Rotation** - Enter the degree angle to rotate the text (0-360 degrees).
- **Text** - Enter the text value to place into the H2OMAP Sewer display.
- **Erase** - Click this button to remove the selected text.
- **Location** - Click this button to move the text from one location to another. Once clicked, the mouse icon will acquire a question mark. Click a new location to place the text and the Edit Text Map dialog box will reappear. Use the Apply button to see the change.
- **Apply** - Use this button to see a change in the selected text without having to exit the Edit Map Text dialog box. For example, to move the text and change the color of the font, use the Location and Color buttons. To see the modifications, click the Apply button.
- **OK** - Click the OK button to exit the Edit Map Text dialog box.
- **Cancel** - Cancel all changes and close the Edit Map Text dialog box.

InfoSewer™

Modeling Made Easy

Group Editing

The Group Editing command allows you to specify modeling properties of multiple network components in a single operation.

To run the command, from the **Edit** menu, select **Group Editing on Selection** or **Group Editing on Domain**. If you have created a domain, the group editing dialog box will appear once the Group Editing on Domain command is initiated. The group editing on selection option allows for a graphical selection to be made. Once initiated, a "?" will appear next to the mouse cursor. Drag a selection window across the screen to select the data elements for group editing.

Once any of the elements selected for a group edit are populated with data, the information input by the user will also appear in various database tables under the DBEditor. For example, when a group of pipes are assigned a set of pipe constraints, those pipes will be added to the Pipe Design Constraints database found under the DBEditor - Folder 5. Supplementary Element Data.

Click on any of the tabs to learn more.

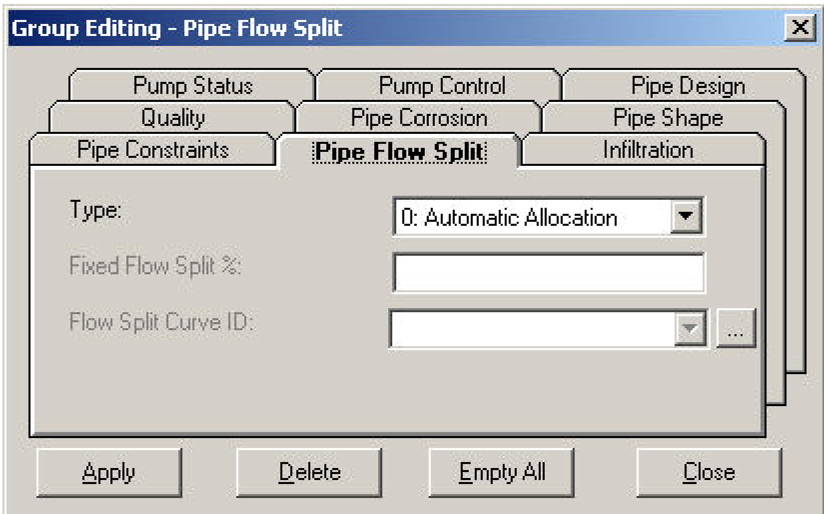
The Pipe Constraints Tab

The pipe constraints tab is used to specify the maximum and minimum velocity constraints to be applied during design simulation. Specifying a pipe constraint on a single pipe will override the global constraint for that pipe as assigned in the Simulation Options -> Design tab.

- **Maximum Velocity** - Input a high end value for which pipe velocities will be analyzed during a steady state design analysis.
- **Minimum Velocity** - Input a low end value for which pipe velocities will be analyzed during a steady state design analysis.
- **Maximum Cover** - The maximum cover criteria to be used during a steady state design simulation.
- **Minimum Cover** - The minimum cover criteria to be used during a steady state design simulation.

The Pipe Flow Split Tab

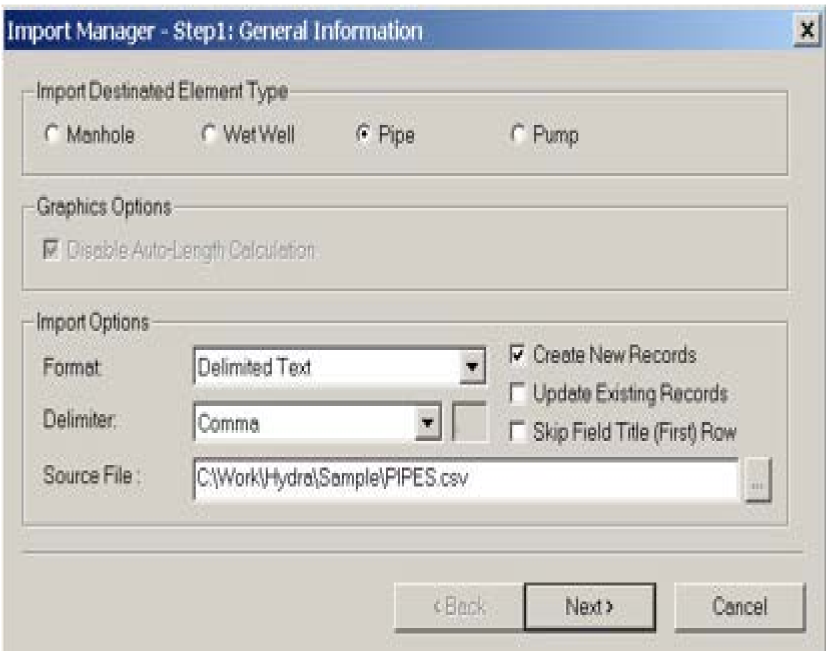
The pipe flow split tab allows the user to edit the method for analyzing a flow split.



- **Type** - Specify whether the calculation will be automatic (0), a fixed percentage (1), assigned a flow split curve (2), or assigned an Inflow-Outflow curve (3).
- **Fixed Flow Split %** - If option 1 is selected, the user must specify the percentage to split the flow.
- **Flow Split Curve ID** - If option 2 or 3 is selected, the user must specify the curve ID to be used in the flow split analysis.

The Pipe Design Tab

The pipe design tab is used to specify the criteria and cost curves for a steady state analysis and steady state design simulation. Specifying a pipe constraint on a single pipe will override the global constraint for that pipe as assigned in the Simulation Options -> Design tab.



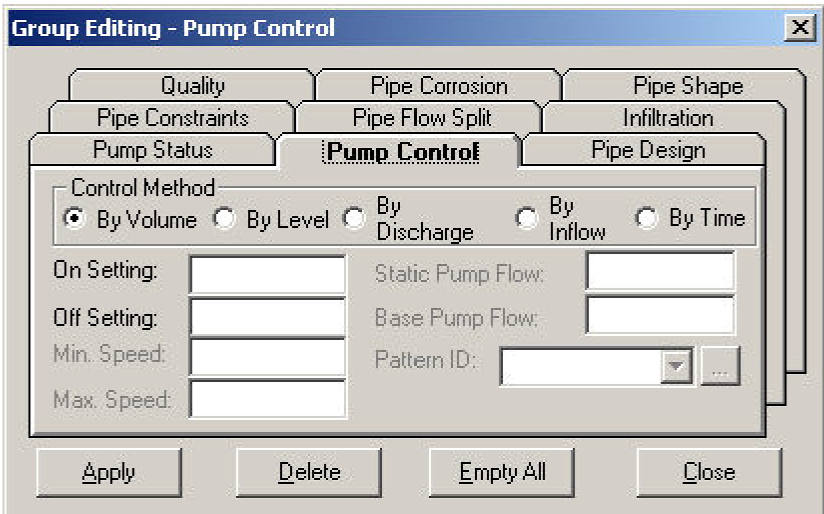
- **Design Criteria Curve ID** - The d/D criteria curve to be used in analyzing a replacement or parallel pipe during a steady state simulation.
- **Analysis Criteria Curve ID** - The d/D criteria curve to be used in analyzing an existing pipe during a steady state simulation.
- **Replacement Cost Curve ID** - The cost curve to be used for recommended replacement pipes during a steady state design simulation.
- **Parallel Cost Curve ID** - The cost curve to be used for recommended parallel pipes during a steady state design simulation.
- **Maximum Allowed Diameter -** Applies to design of new systems, and it refers to the maximum possible size that the pipe is allowed to take. This variable is also used as a flag indicating

whether changing pipe size to meet the design criteria that are not met by changing pipe slope alone is allowed. If this input parameter is not defined for a pipe (i.e., if a value greater than zero is not assigned), the pipe will take the larger of the original diameter assigned to the pipe (if defined), or size of the upstream pipe(s).

- **Minorloss Crown Drop -** At manholes, pipes that change flow direction may exhibit head loss. This head loss may be accounted for by dropping the pipe crown by the amount of the anticipated headloss, which is commonly assumed to be 0.1 ft (i.e. 30 mm). Minor loss crown drop is considered only for pipes that change flow direction but not pipe size (i.e. if a pipe changes direction and its diameter remains the same as upstream pipe diameter), crown elevation of the downstream pipe may be lowered by the Minorloss Crown Drop amount. The unit is ft and mm in US customary and SI systems, respectively. The default value is 0.1 ft and 30 mm in US customary and SI systems, respectively.
- **Default D vs. Min Slope Design Curve -** The minimum slope allowed for various conduit sizes is supplied to the model using this curve. X-axis values are pipe diameters and Y-axis values are minimum slope. The curve defined here will be applied to the selected pipe in the collection system. A default curve is available for both US customary units and SI units.
- **Angle of Excavation -** Refers to slope of the trench excavation angle which depends on soil type. This value defined here may be utilized for all pipes in the system for which angle of excavation is not defined locally.

The Pump Control Tab

The pump control tab allows the user to specify pump settings for selected pumps.

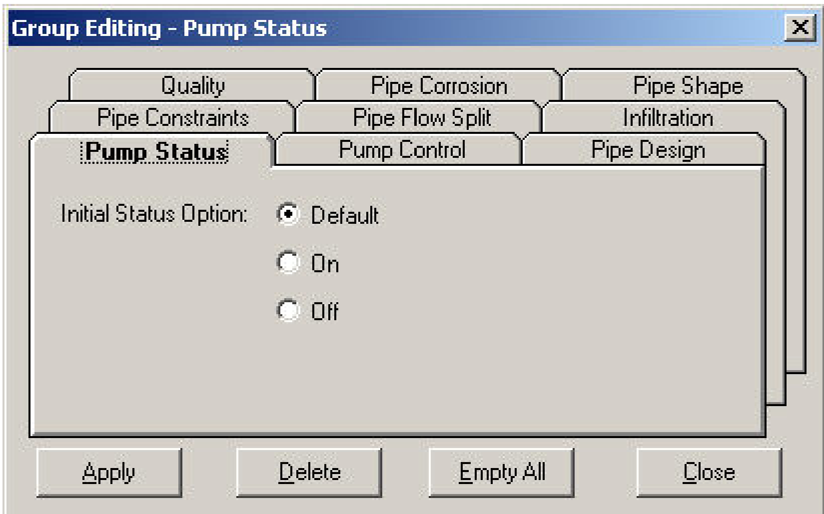


- **Control Method** - Specify the desired control method.
- **On/Off Setting** - If the selected control method is "By Volume" or "By Level", assign the on and off settings for the pump.
- **Min/Max Speed** - These settings are required if the specified control method is either " By Discharge" or "By Inflow".
- **Static/Base Pump Flow** - These settings are required only for " By Discharge" and "By Inflow" control methods.
- **Pattern ID** - Pattern must be assigned if the selected control method is "By Discharge" or " By Time". By Discharge control requires flow pattern. Speed pattern should be provided for the "By Time" control method.

Technical details of the control methods is available at pump controls.

The Pump Status Tab

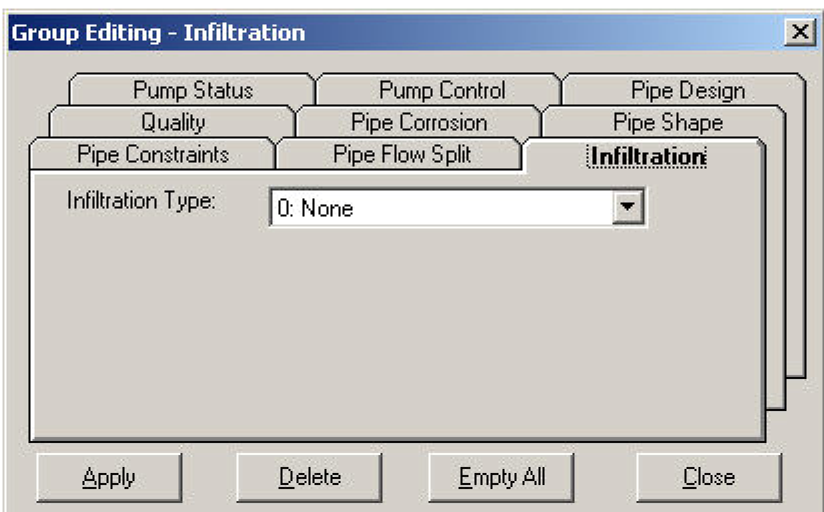
The pump status tab allows the user to specify the initial status of a pump at the beginning of an EPS/Dynamic simulation.



- **Initial Status Option** - Specify whether the initial status of the pump is on or off.

Infiltration Tab

The infiltration tab enables specification of the desired method to model infiltration in to sewer pipes that are buried under ground.



For pipes believed to have infiltration, the model offers five modeling alternatives. These are:

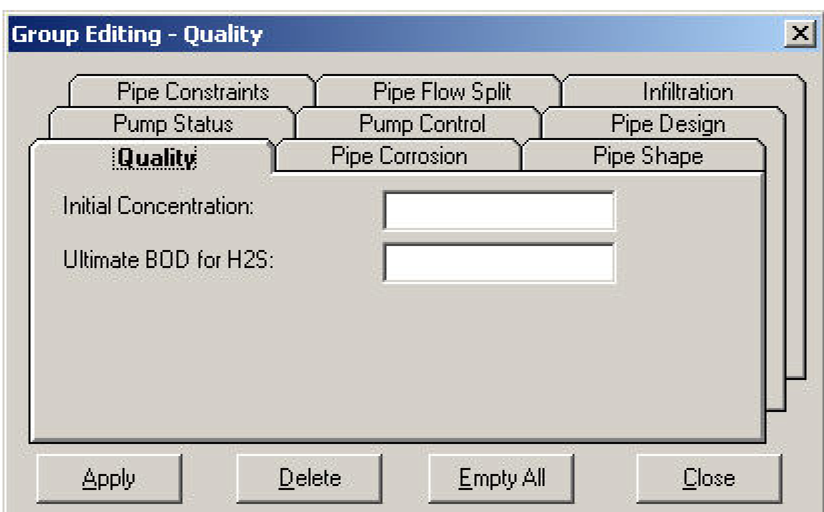
- **Pipe Length** - Infiltration is estimated as a function of length of a pipe. The user has to specify infiltration rate.
- **Pipe Diameter-Length** - Under this option, infiltration is modeled as a function of pipe diameter and pipe length. The user is expected to

provide infiltration rate under this option as well.

- **Pipe Surface Area** - The third option determines infiltration in terms of surface area of the pipe based on user specified infiltration rate.
- **Count Based** - This technique is based on the number of faulty spots on the pipe through which infiltration is subsurface flow is expected to enter the pipe. Here, the user has to specify infiltration rate and the infiltration unit count.
- **Pattern Based** - This option enables the user to specify infiltration flow in the form of a pattern. Base infiltration rate and pattern should if this option is used.

Quality Tab

Quality tab allows specification of initial concentration of conservative contaminant, sediment, or Biochemical Oxygen Demand, initial concentration of total sulfides (during H_2S modeling and corrosion rate prediction) which are required during modeling of pollutant transport, sediment transport, BOD, H_2S, and corrosion rate, respectively.

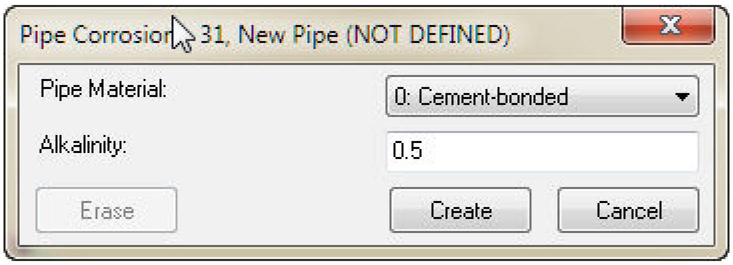


- **Initial Concentration** - This input stands for initial concentration (in mg/l) of conservative contaminant, sediment, BOD, and total dissolved sulfides.

- **Ultimate BOD for H2S** - This input refers to ultimate BOD of the waste water. It is required during H_2S modeling and corrosion rate prediction only.

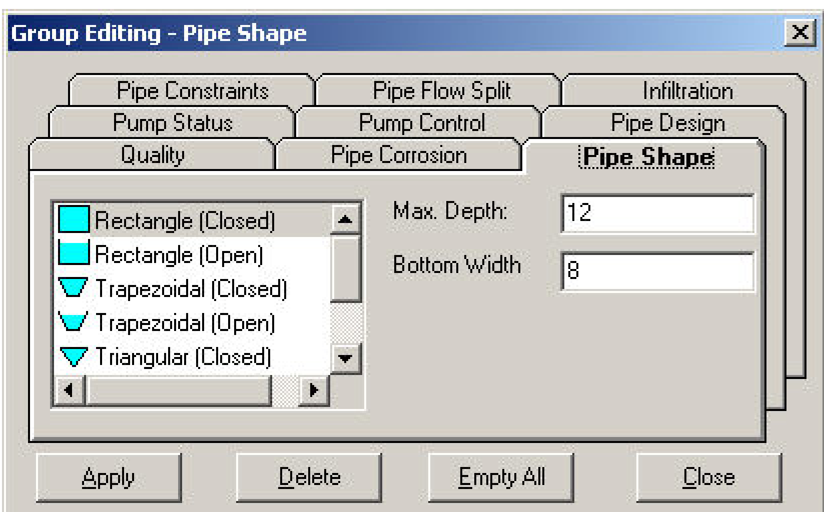
Pipe Corrosion Tab

This tab allows specification of **pipe material** type and **alkalinity** of the pipe material. These inputs are needed during corrosion prediction only.

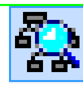


Conduit Shape Tab

This tab enables specification of shape type and associated sizes **for non-circular conduits**.



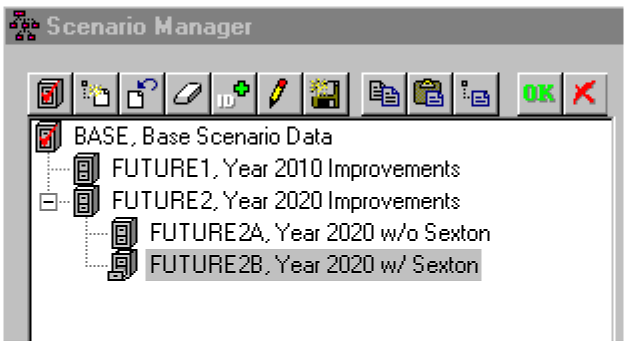
More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Scenario Manager 

The Scenario Manger is where InfoSewer allows the user to create, delete and modify scenarios. The Scenario Manager is what allows the user to create "what if" situations throughout a sewage collection system.

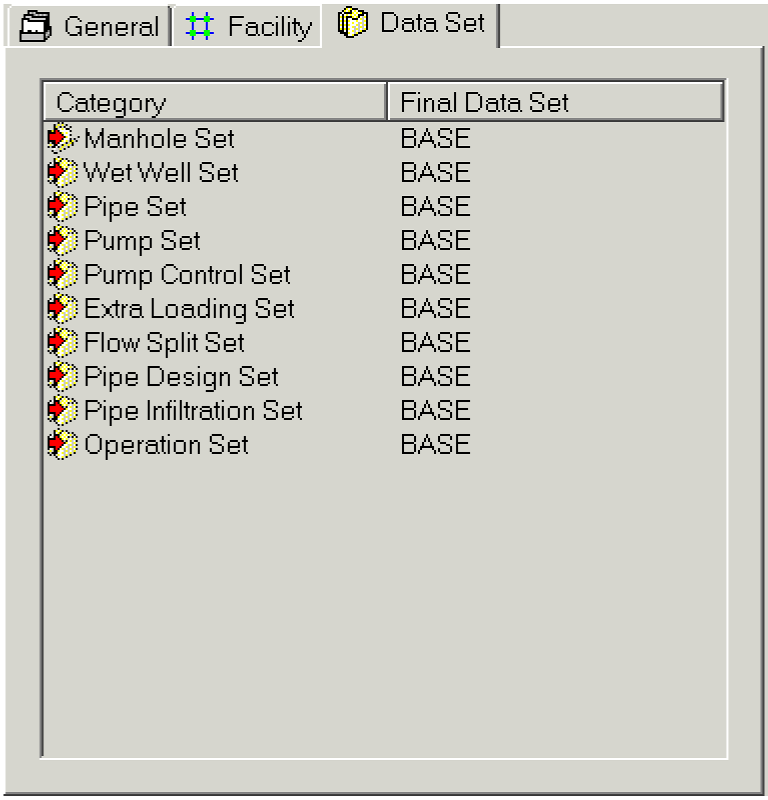
What is a Scenario?

A scenario is a group of InfoSewer facilities, data sets and conditions that are created to reflect a specific modeling situation. With a scenario, you can develop multiple models that are specific to your sewage collection system (ex. Average day loading for a specific service area with unique reporting options).



A scenario is comprised of the following three items:

- **Simulation sets** - General tab
- **Facility set** - Facility tab
- **Data sets** - Data Set tab



Components of an InfoSewer scenario

Each of the three components of a scenario can be further defined as follows:

- **Simulation set** – Created through the General tab, they define the simulation options (durations, timesteps, analysis parameters, etc.) associated with the scenario. There are three different option set types, each storing a logical grouping of simulation options. Click here for more information on simulation sets.
- **Facility set** – Defines the network facilities (components such as pipes, pumps, wet wells and manholes) to be used in a simulation. Only one facility set can be active at a time (facility sets are created through the Facility tab). Click here for more information on facility sets.
- **Data set** – Stores modeling data (pipe diameter, invert elevations, manhole loadings, etc.) associated with each facility in a separate external database. There are nine different data set types (as seen

above), each storing its own unique logical grouping of modeling data. Click here for more information on data sets.

When you define a scenario, you pick the facility, data, and option sets that comprise that scenario. When picking data sets for inclusion in a scenario, you may either specify that a data set associated with a given scenario is included in that scenario independent from other scenarios or alternately may specify that the given data set inherits its contents – properties – from a “parent” scenario. Click here to learn more about Parent-Child relationships and how they are built.

Once you have configured and created a scenario, you can activate that scenario at any time. Once a scenario is activated, any modifications made to any of the databases related to InfoSewer facilities will be changed, but only for the data sets that are related to, and dependent upon, the active scenario. Click here to learn how to activate a scenario.

InfoSewer™
Modeling Made Easy

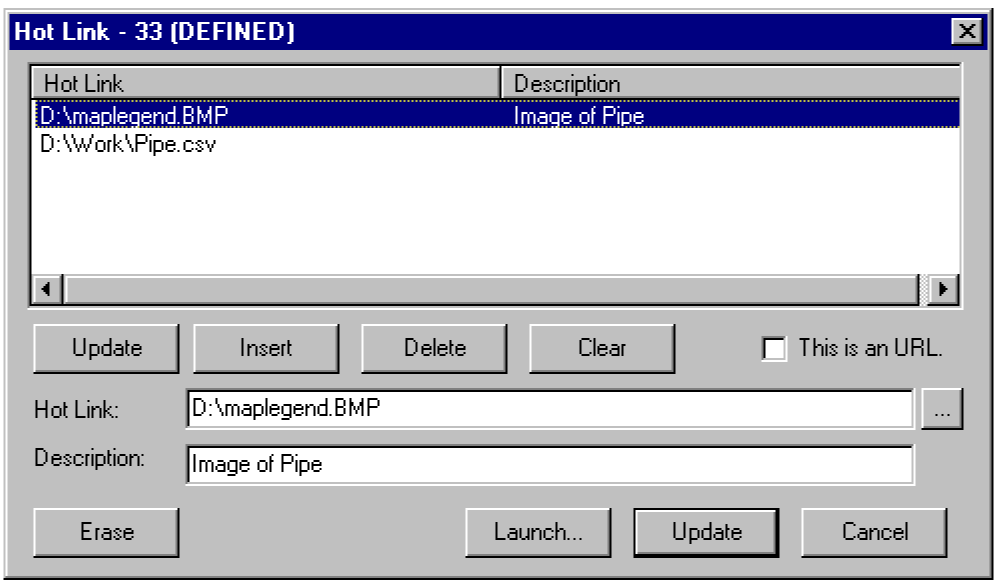
Hot Links

Hot links enable the user to attach an external file to an InfoSewer element to be viewed at a later time. A hot linked file may contain a photograph of the element, a work order history sheet, a cost spreadsheet or a raster image of a construction drawing.

Creating and viewing a hot linked file is a two-fold process. First, a user must attached a hot linked file to the element. Second, the must initialize the hot link execute command in order to view the selected file. The following illustrates the process:

Attaching a File

Select the element in the InfoSewer display for which a hot link is to be created. Click on the hot link icon at the top of the Attribute Browser. The following dialog box will be displayed:



Use the [...] to browse and locate the file on the network. Once the file is located, click the Insert button to associate this file with the InfoSewer element. If you wish to reference a URL, ensure that the "This is a URL" box is checked. Once a file has been attached, click the cancel button to exit the dialog box.

Viewing a File

To view a file, merely click on the Execute Hot Link icon located on the Standard toolbar or click on the Hot Link icon at the top of the Attribute Browser. Once the Hot Link dialog box is opened, highlight the desired file and click on the Launch button to see the hot linked file. When complete, click on the Cancel button to close the Hot Link dialog box.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™
Modeling Made Easy

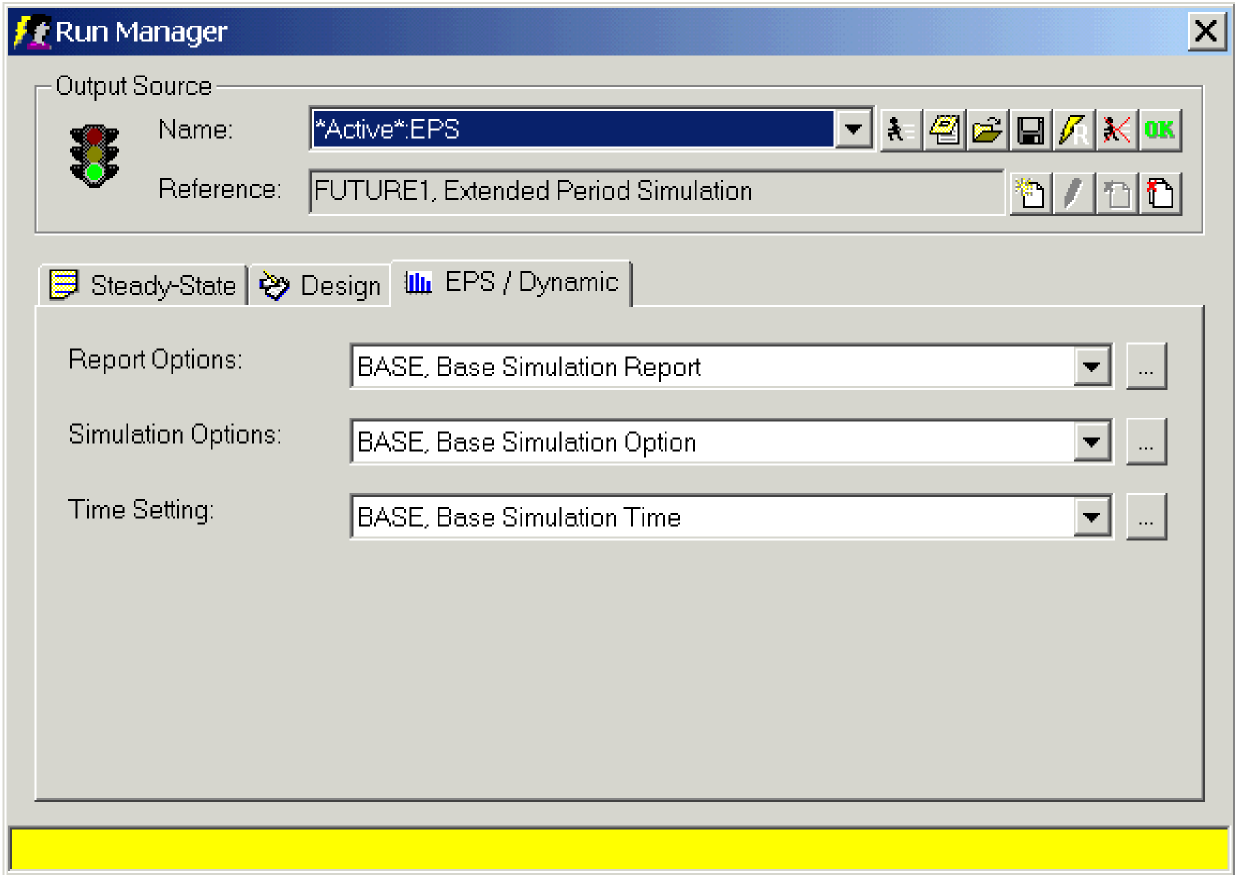
Extended Period and Dynamic Simulations

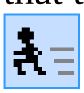
Extended period simulations and dynamic simulations involve loading the collection system, via a base load and diurnal curve, over a set period of timesteps. An EPS/Dynamic model does not involve any peaking equation, criteria or cost curves. The advantage of an EPS/Dynamic simulation is that it enables the user to see how flows are collected and routed over a series of timesteps, thus demonstrating the operation of the collection system. Issues such as wet well cycling and pump operation are easily answered through an **EPS/Dynamic** simulation.

Prior to running an EPS/Dynamic model, the user will need to ensure that loads are properly determined and diurnal patterns have been applied to all flows. The user must also ensure that the Simulation Time option has been used to create the time conditions for an EPS/Dynamic simulation.

Running an EPS/Dynamic Simulation

To run an EPS/Dynamic model, from the **Tools** menu select **Run Manager**. Once there, select the EPS/Dynamic tab, verify report, simulation and time options, and click the Run Manager icon to begin the simulation.



Upon successful completion of a EPS/Dynamic run, results are stored in the active.EPS Output Source. Use the Active Output pull down box to ensure that the *Active*.EPS output is enabled prior to using the Report Manager  to open and display the results of the simulation. Click here to learn more about EPS model result reports.

InfoSewer™

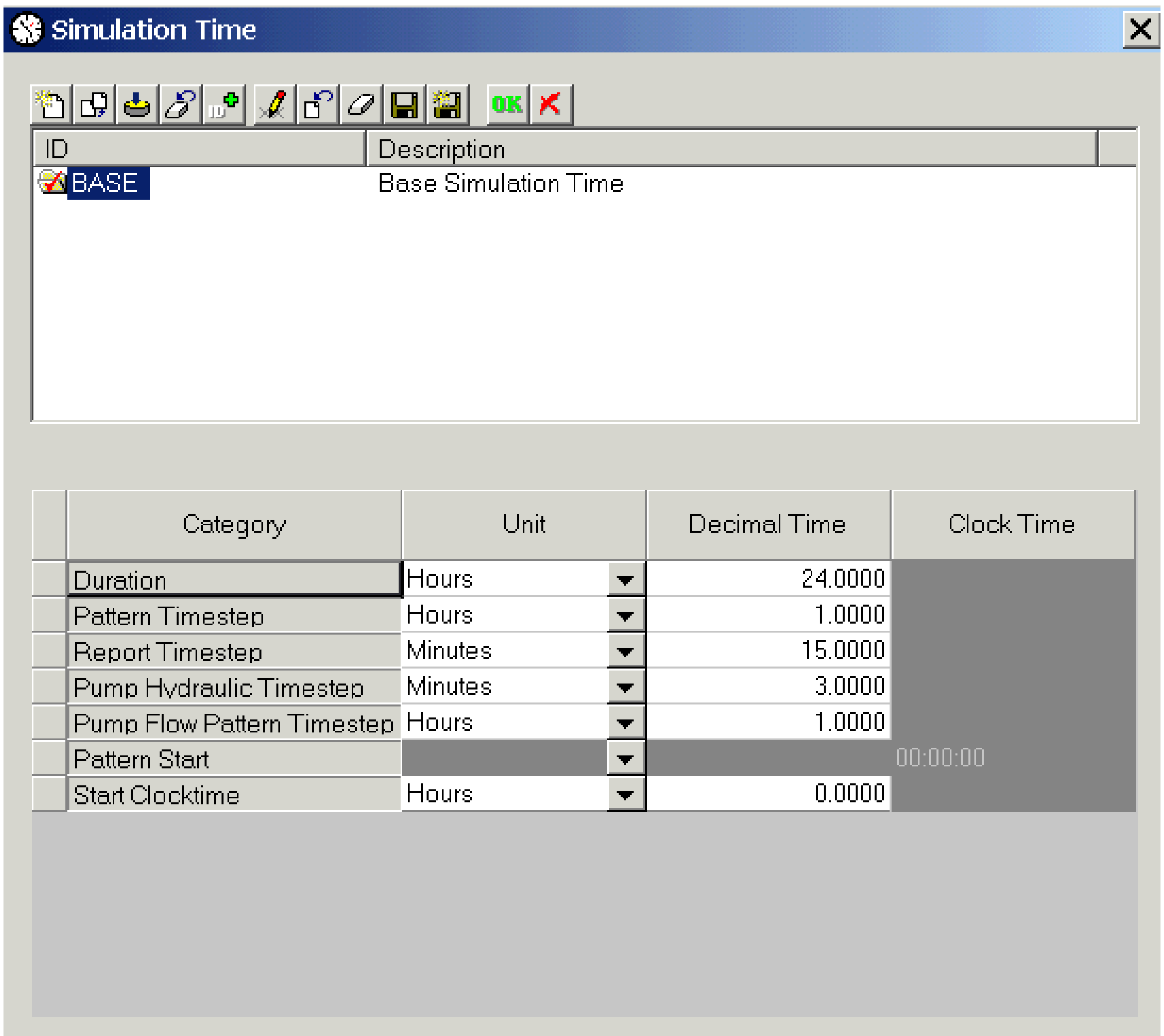
Modeling Made Easy

Simulation Time

The Simulation Time feature of InfoSewer allows the user to change the time-step at which an EPS/Dynamic simulation is to be analyzed. If the user selects the Steady State check box, then only a steady state model will be analyzed. A simulation time option can be set up for each unique scenario within InfoSewer.

Create Simulation Time

To create a new simulation time, from the Control Center click on the Operation Data tab and select Simulation Time. Click on the New button at the top of the Control Center dialog box and supply a unique name and description for the simulation option. When the OK button is chosen, the following dialog box will appear.



Simulation Time Options

When performing an EPS/Dynamic, many modeling parameters are dependent on analysis timesteps (intervals) and duration. The following durations and time steps may be specified in InfoSewer:

- **Duration** – Total length of an EPS/dynamic simulation in hours.
- **Pattern Timestep** – Time interval to be used with all time patterns. Normal default is 1 hour.
- **Report Timestep** – Time interval which analysis results are reported. Normal default is 1 hour. To achieve greater accuracy it is recommended that you use shorter report time step intervals.
- **Pump Hydraulic Timestep** – Time interval for which a hydraulic condition is evaluated for a pump. Decreasing this number will increase result accuracy.
- **Pump Flow pattern Timestep** – Time step for pump discharge and pump speed patterns that are used for pumps controlled based on time

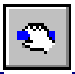
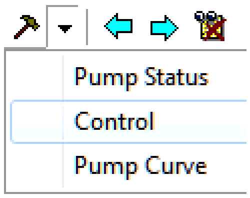
or by desired downstream discharged flows.

- **Pattern Start** –The pattern hour at which the simulation is to evaluate (e.g., a value of 2 means that the simulation begins with all time patterns starting at their second hour). Normal default is 0.
- **Start Clocktime** – Clock time (e.g., 7:30 am, 10:00 pm) at which simulation begins. Default is 12:00 am (00:00 - midnight).

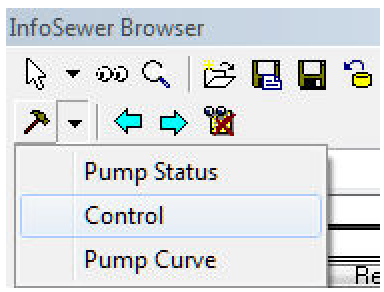
Once a simulation time is created, it can be associated with any scenario specified by the user or evaluated at any time via selection from the Run Manager.

InfoSewer™ Modeling Made Easy

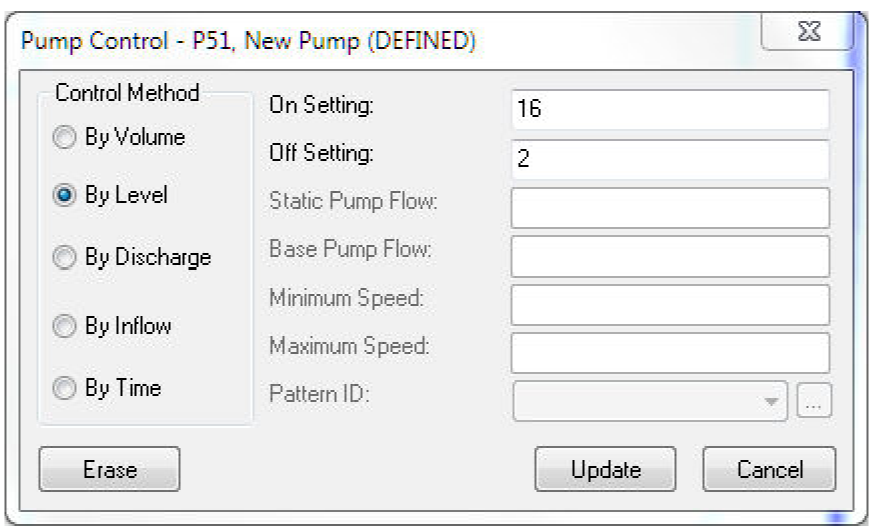
Pump Controls

Pump Control  or 

A pump control allows the user to modify the initial status of a pump during a hydraulic simulation - depending on the state of a wet well in the system as specified by the user. Actual field operations may consist of having a pump turn on and off, depending upon the high and low water levels (or volumes) in a wet well. To model this type of situation, the user would place a simple control on the pump in question. Pump controls apply to extended period simulations only.



InfoSewer provides five control alternatives as shown below.



The alternatives and the variables are described below.

Control Method - The control method specifies the criteria that dictates the ON and OFF settings of a pump.

- **By Volume** - The pump operates depending on the volume of sewage in the wet well. The user should provide the on/off volumes.
- **By Level** - The pump may turn on or off based on water level in the wet well. The on/off levels need to be specified for this control option.
- **By Discharge** - This control method dictates a mechanism in which a user supplied pattern of targeted pump discharged flow is maintained. The software adjusts the speed of the pump to insure that the desired amount of flow is pumped. This option is available for one-point and exponential three-point pumps only, and is not valid for lift (pump) stations with parallel pumps. The actual pumped flow could differ from the desired flow if the minimum and the maximum water levels set for the wet well are violated during the simulation duration. In other words, the pump may turn off if the water level in the wet well falls below the minimum wet well level allowed in spite of the desired flow determined based on the pattern. Like wise, the pump may turn on if the water level in the wet well exceeds the maximum allowed water level even if the targeted flow is zero.
- **By Inflow** - Under this control alternative, the level of water in the wet well remains constant during the simulation duration. The pump turns on if there is inflow to the wet well. The discharged flow is equal to the inflow to the wet well. This control alternative is ideally suited for wet wells that do not have enough storage volume. This option is available for one-point and exponential three-point pumps only, and is not valid for lift (pump) stations with parallel pumps.
- **By Time** - The "By Time" control option offers the user the flexibility to turn the pump on/off at any time of a day. The

model accepts the operational schedule in the form of a speed pattern. The pump turns off if the speed setting is zero, and turns on otherwise.

- **On Setting** - The volume or height of water that turns the pump on. This setting is valid for " By Volume" and " By Level" control options.
- **Off Setting** - The volume or height of water that turns the pump off. This setting is valid for " By Volume" and " By Level" control options.
- **Static Pump Flow** - This variable is used for " By Discharge " control method, and is used during steady state simulations only. It represents the actual amount of sewage that the user wants to be pumped during steady state simulations.
- **Base Pump Flow** - This variable is required for " By Discharge " control during EPS/Dynamic simulations alone, and represents the base flow to be multiplied by the pattern factors to determine the actual flow to be lifted at the time step being analyzed. For example, if the pattern factor at hour two is 1.2 and base flow for the pump is 5 cfs, the actual pump discharge at hour two would be 6 cfs (i.e., 5 cfs* 1.2).
- **Minimum Speed** - Represents minimum speed of a pump, and is required for the "By Discharge" and "By Inflow" control options. The default setting is zero.
- **Maximum Speed** - Represents the maximum speed that a pump could sustain, and is required for the "By Discharge" and "By Inflow" control options.
- **Pattern ID** - Pattern is required for the "By Discharge" and "By Time" control options. The former requires flow pattern, while the latter control option needs speed pattern.
- **Erase** - Delete the current control setting.
- **Update** - Change the control setting with the new criteria specified. Once a control has been created, click on the Update button to place

the control statement into the Control Rules box and exit the dialog box.

- **Cancel** - Quit the command with saving changes (so long as update was not pressed first).

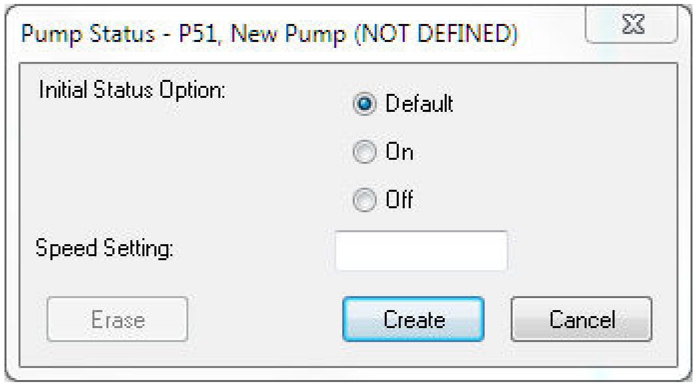
The described dialog box could be accessed from the attribute browser.

- Select the pump to which the control is to be assigned, and then click on the control icon ( ).
- Select the desired control and supply the corresponding data. As an example, for "By Discharge" control, chose the control type. If dynamic simulation is desired, assign the appropriate pattern ID and the base pump flow value. Steady state and design simulations demand only static pump flow value.
- Click on the update button.

The procedure has to be repeated for all pumps in the collection system. Pump controls could also be assigned to multiple pumps at a time using the group editing feature.

Initial Status and Setting

Speed and initial on-off status of pumps in the collection system can be specified by first selecting the, and then selecting **Initial Status** command from Tools Icon drop down box located the top of the Attribute Browser. The default pump speed setting is one (pump speed ratio of 1). The initial pump status is overwritten by the operational controls during an extended period simulation. The following dialog box is activated once the "Initial Status" button is clicked, and could be edited to reflect the desired settings.



Saving Control Sets

After a simple control is created, it is stored as a record in the Standard Control database (see DBEditor). This database is by default, the *active* database for the control set. You may wish to have different controls applied to the same pump depending on which scenario is being analyzed. This is done by cloning the *active* control set to a new control set which makes an exact copy of the *active* database. By assigning a unique name to this new control set, the user can now assign that control set to any scenario. As that scenario is activated in the future, it will overwrite any pre-existing records in the *active* control set with the contents of the saved control set database.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Water Quality

With InfoSewer, you can perform several types of water quality analyses in conjunction with a hydraulic simulation. The purpose of the quality model is to simulate pollutant loading and buildup, washoff, as well as individual domestic, commercial and industrial contributions, and transport. Five different types of quality analyses can be carried out by InfoSewer and are explained below.

POLLUTANT TRANSPORT

InfoSewer can effectively simulate the transport of dissolved pollutants throughout the sewer collection system. It tracks the movement of conservative constituents (e.g., chloride, bromide, sulfate, boron, sorbed trace metals) flowing through the network over time. The dynamic quality simulation model is predicated on conservation of mass coupled with reaction kinetics and consists essentially of three processes: advection in pipes, mixing at sewer manholes and wet-wells, and kinetic reaction mechanism. Longitudinal dispersion due to concentration gradient is neglected which means that there is no intermixing of mass between adjacent sewage parcels traveling down a pipe. Advective transport within a sewer pipe is represented with the following equation:

$$\frac{\partial C_i}{\partial t} = -u_i \frac{\partial C_i}{\partial x} + r(C_i) \quad (24)$$

where C_i is the concentration (mass/volume) in pipe i as a function of distance x and time t; u_i is the flow velocity (length/time) in pipe I; and r is the rate of reaction (mass/volume/time) as a function of concentration.

For conservative (inert) pollutants the rate of reaction (r) is set to zero. For pumps, instantaneous substance advection is assumed.

At sewer manholes and wet-wells, the mixing of fluid is taken to be complete and instantaneous. Thus the concentration of a substance in sewage leaving the manhole or wet-well is simply the flow-weighted sum of the concentrations from the incoming pipe(s) and is described by the following equation:

$$C_{out} = \frac{\sum Q_{in} C_{in}}{\sum Q_{in}} \quad (25)$$

where $Q_{in}$is the incoming flow (volume/time); C_{in} is the pollutant concentration of the incoming flow; and C_{out} is the concentration of the pollutant leaving the manhole.

Under completely mixed conditions, the concentration throughout the wet-well is a blend of the current contents and that of any entering sewage and is represented by the following equation:

$$\frac{\partial(V_{ww} C_{ww})}{\partial t} = \sum Q_{in} C_{in} - \sum Q_{out} C_{ww} \quad (26)$$

where $V_{WW}$is the wet-well volume at time t, C_{WW} is the wet-well concentration; and the remaining terms are as defined above.

The ability to model pollutant transport in sewer collection systems is useful in determining the amount of pollutants that is transported to the wastewater treatment plant and assessing impact on the receiving waters.

TIME OF CONCENTRATION

InfoSewer can model the changes in the age of sewage flow (time of concentration) throughout a collection system. Time of concentration is the time spent by a sewage flow parcel in the network (i.e., the time of flow in the sewerage system). This parameter is useful to address important water quality and safety issues such as generation of sulfide that may occur in a sanitary sewer system (which manifest itself in corrosion and odor issues).

In InfoSewer , new sewage entering the network from loading manholes enters with age of zero. As this sewage moves through the collection system it splits apart and blends together with sewage flow parcels of varying age at manholes and wet-wells. InfoSewer provides automatic modeling of sewage age. Internally, it treats age as a reactive constituent whose growth follows zero-order kinetics with a rate constant (r =1.0) equal to 1 (i.e., each second the sewage becomes a second older). Time of concentration for a manhole is thus calculated as a flow-weighted average sewage age value of flows entering the manhole. Travel time of a sewage flow parcel through a sewer pipe is computed based on flow velocity and pipe length. For the pipes leaving a manhole, the average sewage age is increased by the travel time to the next downstream manhole.

SOURCE TRACING

InfoSewer can also perform sophisticated source tracing calculations. Source tracing tracks over time what percent of sewage reaching any pipe or manhole in the network had its origin at a particular source node. The source node can be any manhole in the network, including wet-wells. Source tracing is very useful in sewer collection systems, and could be used for (1) tracking changes in sewage flow contribution (and associated constituents) over space and time; (2) predicting impact of industrial and commercial waste discharges on performance of wastewater treatment plants; (3) determining contaminant level that causes a wastewater treatment plant to be in violation of its discharge permits; (4) and developing appropriate user charges based on wasteloads and level of contaminant. Internally, InfoSewer treats the source node as a constant source of a non-reacting constituent that enters the network with a concentration of 100.

BIOCHEMICAL OXYGEN DEMAND

Biochemical Oxygen Demand (BOD) is the most widely used parameter of organic pollution in sanitary sewer systems. The ability to model BOD is of great importance in wastewater engineering to assist wastewater utilities in (1) estimating the quantity of oxygen required to biologically stabilize the organic matter present; (2) determining the size of wastewater treatment facilities; (3) evaluating the efficiency of the treatment process; and (4) ensuring compliance with wastewater discharge permits.

InfoSewer models the rate of BOD oxidation (exertion) throughout the collection system using first-order kinetics with the rate of oxygen utilization being proportional to the difference between the amount of oxygen used and the ultimate BOD as:

$$\frac{dBOD}{dt} = -kBOD \quad (27)$$

or

$$BOD = UBOD(1 - e^{-kt}) \quad (28)$$

where

BOD = *BOD* exerted at time *t*, mg/L

UBOD = total or ultimate carbonaceous *BOD*, mg/L

k = first-order reaction rate constant, 1/day

t = time, day

The value of the first-order reaction rate constant *k* varies with the type of waste and normally ranges from 0.05/day to 0.46/day or more, with a typical value of 0.23/day. This constant can also be expressed as a function of temperature as:

$$k = k_{20}\theta^{T-20} \quad (29)$$

where k_{20} is the first-order reaction rate constant at 20^{o}C (1/day) and *T* is the temperature (degree Celsius). The value of θ is 1.056 in the temperature range between 20^{o}C and 30^{o}C and 1.135 in the temperature range between 4^{o}C and 20^{o}C, with a typical value of 1.047. InfoSewer determines and uses appropriate values of k and θ based on the user-specified temperature. The default values used by InfoSewer for k, θ, and T are 0.23/day, 1.047, and 20^{o}C, respectively.

SEDIMENT TRANSPORT

Sanitary sewer systems can carry substantial loads of suspended solids (waste solids). These sediments are complex mixtures of cohesive (organic) and non-cohesive (minerals) materials and exhibit a wide range of particle sizes and densities. They can collect causing blockages (shock loading under periods of low flow) and overflow events, as well as impairing the hydraulic capacity of the sewer pipes (by restricting their flow area and increasing the bed friction resistance). In addition, when waste solids are intermittently agitated and moved along the pipes, sulfide generation is increased which may cause various problems including odor, hazard to maintenance crews, and corrosion of unprotected sewer pipes produced from cementitious materials and metals. Limiting velocity criteria are generally adopted when designing sanitary sewer systems to control sediment deposition in the pipes.

Wastewater flow velocity and sediment transport in sewer systems are interdependent. As the flow increases from zero, flow-induced forces (lift and drag) acting on the sediment particle increase. When these forces exceed the submerged weight of the particle, the sediments start to move. With a further increase in velocity, the particles will be suspended by eddies of fluid turbulence and move downstream with the wastewater. With a subsequent decrease in flow velocity (beyond the limiting or terminal velocity), sediment particles will start to settle by gravity at a rate proportional to their settling velocity. A further increase in flow velocity provides the energy to scour and transport the deposited material along the sewer pipes. This dynamic process continues based on the flow conditions inherent in the sewer system.

InfoSewer can simulate the transport and gravitational settling of sediments (total suspended solids including grit) over time throughout the sewer collection system under varying hydraulic conditions. As long as flow velocity exceeds the critical/terminal velocity, InfoSewer assumes that the sewage flow has the capacity to transport all incoming sediments. Deposited sediment particles are also assumed to be scoured and transported downstream when velocity of the sewage flow exceeds the terminal velocity. Settling starts when flow velocity falls below the critical velocity. In the model, transport of the sediment particles is governed by advection (Eq. 24) implying that the particles are transported at local flow velocity. The model assumes that the sewer pipe diameter and roughness coefficient remain constant and are unaffected by sediment deposition.

HYDROGEN SULFIDE

The control of odorous gases and the corrosion of sewers are the two most important problems in operating wastewater collection systems. Evaluation of existing or potential odor or corrosion problems, and identification of where such problems will occur is, therefore, highly essential. In wastewater collection systems, odors are produced as a result of biological decomposition of organic matter, particularly those containing sulfur and nitrogen, under anaerobic conditions prevailing in the slime layer of gravity pipes, force mains, and wet wells. Hydrogen sulfide (H_2S) and ammonia are the only malodorous inorganic gases produced from the decomposition. Other odor producing substances include organic vapors such as idoles, skatoles, mercaptans and nitrogen-bearing organics. However, H_2S is the most commonly known and prevalent odorous gas associated with domestic wastewater collection and treatment systems. *H_2S Detector*™ extension for H2OMAP Sewer suite gives wastewater engineers a powerful Operations and Maintenance (O&M) tool to readily model and analyze entire sewer collection systems for sulfide generation and corrosion potential under varying conditions anticipated throughout the life of their systems.

Hydrogen sulfide has a characteristic rotten egg odor, is extremely toxic, is corrosive to metals, and is a precursor to the formation of sulfuric acid (which corrodes concrete, lead-based paints, metals, and other materials). The conditions leading to formation of H_2S generally favor the production of other odorous organic compounds. Therefore, investigation of the conditions favoring H_2S formation not only helps to quantify the potential for odor generation from other compounds, but also it aids in identifying potential corrosion problems in the collection system.

The occurrence of H_2S in wastewater collection systems, other than that added from industrial sources and infiltrated groundwater, is primarily the result of the reduction of sulfate ion ($SO^{=}_4$), one of the most universal anions occurring in natural waters, under anaerobic conditions, as shown by the following reaction.

$$SO^{=}_4 + \text{organic matter} \xrightarrow[\text{bacteria}]{\text{anaerobic}} S^{=} + H_2O + CO_2$$

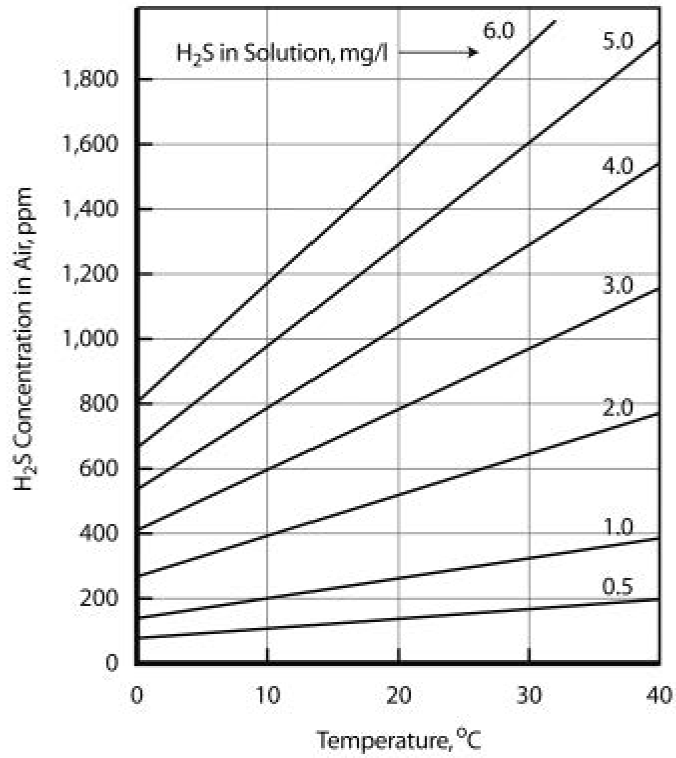
$$S^{=} + 2H^{+} \longrightarrow H_2S$$

The molecular H_2S, formed from sulfate reduction, dissolves in the waste water and dissociates in accordance with reversible ionization reactions, expressed as:

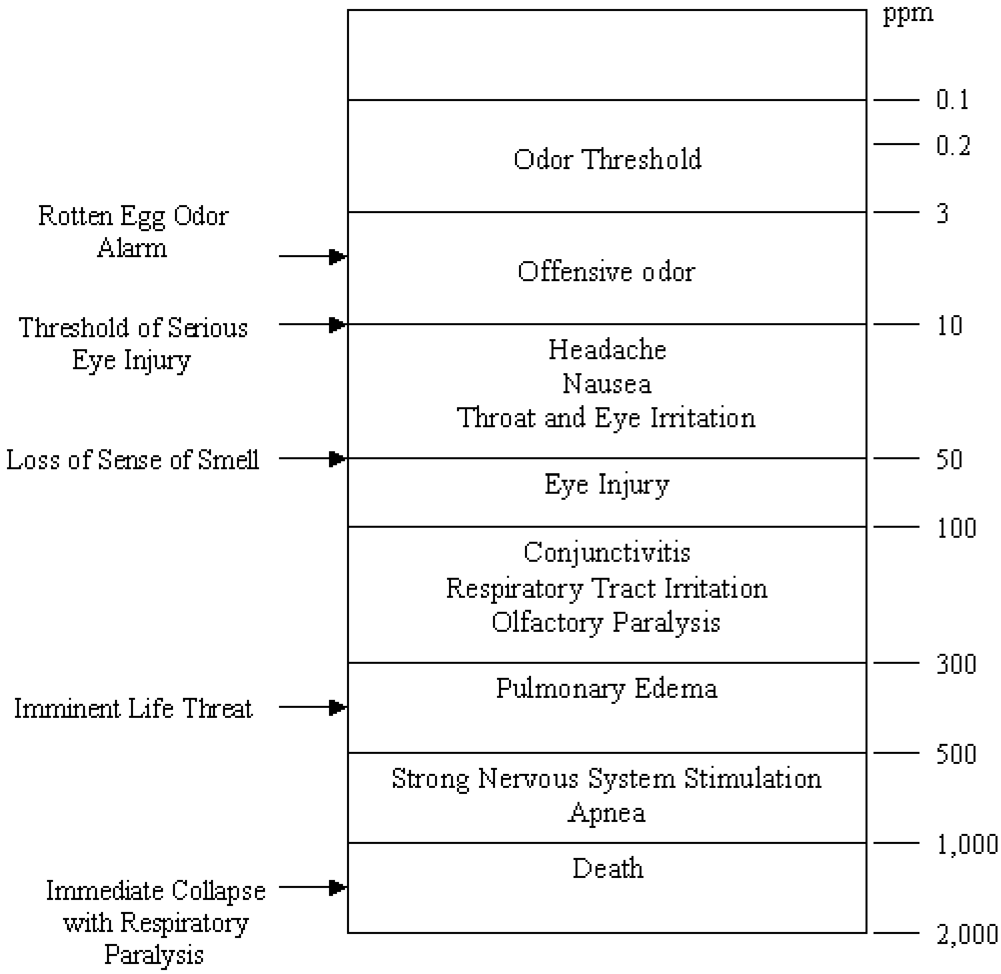
$$H_2S \longleftrightarrow HS^{-} + H^{+}$$

$$HS^{-} \longleftrightarrow S^{=} + H^{+}$$

The partitioning of the hydrogen sulfide into these components (i.e., (H_2S)aqueous, HS^- ion, and $S^=$ ion) depends primarily on the temperature and the pH of the wastewater, although ionic strength, as represented by dissolved solids or electrical conductivity, also affects the partitioning. The HS^- ion and $S^=$ ion produce no odors. Some of the aqueous H_2S will escape into the sewer atmosphere causing the odor problem. The concentration of H_2S gas in the atmosphere will vary with the concentration of (H_2S) aqueous according to Henry's law. The rate of escape of H_2S gas is a function of the difference between the saturation or equilibrium concentration determined by Henry's law and the actual concentration of H_2S in the sewer atmosphere. The EPA provides the figure below that shows H_2S in the sewer atmosphere in equilibrium with the given concentrations of aqueous H_2S concentration in the wastewater at the respective temperatures, for a pressure of one atmosphere.



Prediction of the rate of sulfide buildup and corrosion potential is an essential element in the design of new sewer systems as well as in the evaluation of existing systems. The rate of sulfide buildup depends on a number of environmental conditions, including, concentration of organic material and nutrients, sulfate concentration, dissolved oxygen (DO), pH, temperature, stream velocity, surface area, and detention (residence) time. Accounting for all these environmental conditions, H_2S Detector predicts sulfide buildup in sewer collection systems for gravity sewers, force mains, and wet wells using the Pomeroy-Parkhurst equations. *H_2S Detector* enables wastewater utilities to pinpoint odor and corrosion problems, develop effective monitoring programs, alert plant operators and sewer maintenance workers to potential danger and the need to observe safety practices, and evaluate and implement effective control system such as aeration, chlorination, and mechanical cleaning. As described above, H_2S is an acutely toxic material. It can cause serious health hazards even at very low concentrations. The physiological effects (i.e., toxicity spectrum) ofH_2S are summarized in the following figure provided by EPA.


ppm
0.1
Odor Threshold
0.2
3
Rotten Egg Odor Alarm
Offensive odor
Threshold of Serious Eye Injury
10
Headache
Nausea
Throat and Eye Irritation
Loss of Sense of Smell
50
Eye Injury
100
Conjunctivitis
Respiratory Tract Irritation
Olfactory Paralysis
300
Imminent Life Threat
Pulmonary Edema
500
Strong Nervous System Stimulation
Apnea
1,000
Immediate Collapse with Respiratory Paralysis
Death
2,000

CORROSION PREDICTION

Corrosion is one of the primary reasons that sewer systems lose their structural integrity. Corroded sewer pipes may allow greater inflow and infiltration into the collection system, further deteriorating reliability of the network by causing undesirable conditions such as surcharges and overflows, ultimately requiring premature replacement of the pipes. Corrosion of unprotected concrete or metal surfaces is primarily due to the production of sulfuric acid in sewer systems through oxidation of hydrogen sulfide gas by bacterial action on the exposed surfaces under aerobic conditions. This type of corrosion is commonly referred to as microbially induced corrosion. *Corrosion Predictor*^{TM} extension for *H2OMAP Sewer* suite helps wastewater engineers to predict the rates of corrosion of the sewer pipes in their collection systems under varying environmental and hydraulic conditions. It enables them to pinpoint corrosion problems, prioritize repairs, specify corrosion resistant materials or select other forms of corrosion protection (e.g., protective linings).

As previously described in the hydrogen sulfide section, some of the soluble H_2S available in pipes may escape into the sewer atmosphere and transferred to the pipe walls above the wastewater surface. It will then be taken up as it comes in contact with the damp surfaces of pipes. The H_2S retained in this dampness is then converted to sulfuric acid by aerobic bacteria, as described in the following reaction:

$$H_2S + 2O_2 \xrightarrow{\text{bacteria}} H_2SO_4$$

Next, some or all of the produced sulfuric acid reacts with the pipe material causing corrosion, mainly at the inside pipe wall above the wastewater flow line. The amount of sulfuric acid that reacts with the pipe material depends on the rate of production of the acid, which in turn depends on moisture, the presence of oxygen, and the mass emission of sulfide gas. If the rate of acid production is slow, almost all of the acid will react with the pipe material. If the rate of production is rapid, much of the acid will not be able to diffuse through the material. Consequently, it will be carried down the walls of the pipe and into the flowing wastewater stream where the sulfuric acid reacts with alkalinity producing sulfate ion.

In addition to the concentration of acid present, the corrosive effect of sulfuric acid varies according to the type of pipe material used and the ambient temperature. Cementitious pipes, including ferrous pipes with mortar lining, experience a reaction that converts the surface material into a pasty mass, which is primarily a calcium sulfate ($CaSO_4$), commonly referred to as gypsum. This pasty mass may fall away and expose new surfaces to corrosive attack. Ferrous pipe materials may experience surface reaction in which a portion of the material is dissolved and a portion is converted to iron sulfide, yielding a hard bulky mass that forms on the exposed surface. A warm and humid environment creates good condition for microbial induced corrosion.

The rate of pipe corrosion depends upon the rate of sulfuric acid production, the amount of the produced sulfuric acid that reacts with the pipe material, and the alkalinity of the pipe material. Sulfuric acid production is related to the amount of hydrogen sulfide gas that escapes to the sewer atmosphere, which in turn depends on a number of environmental and hydraulic conditions previously described in relation to hydrogen sulfide buildup, partitioning, and release. *Corrosion Predictor*^{TM} uses the following equations to estimate rate of corrosion for cementitious (cement-bonded) materials and ferrous materials in gravity sewers. In force mains, where the lines flow full, there is generally no internal corrosion since generation of sulfuric acid is prevented.

For cement-bonded materials (Metcalf & Eddy 1981),

$$C = 11.4 k \phi_{sw} \frac{1}{A} \quad (30)$$

where

C = average rate of penetration, inch/year

k = coefficient of efficiency for acid reaction considering the estimated fraction of acid

remaining on the wall. May be as low as 0.3 and approaches 1.0 for complete acid

reaction

ϕ_{sw} = flux of H2S to the pipe wall, g/m2.hr.

A = alkalinity of the cement-bonded material, expressed as $CaCO_3$ equivalents.

Approximately 0.18 to 0.23 for granitic aggregate concrete, 0.9 for calcareous

aggregate, 0.4 for mortar linings and 0.5 for asbestos cement (USEPA 1985).

For ferrous materials (Metcalf & Eddy 1981),

$$C = 2.04k\phi_{sw} \quad (31)$$

where *C*, *k*, and ϕ_{sw} are consistent with the definitions given above. The default values used by *Corrosion Predictor*^{TM} for *k* and *A* are 0.5 and 0.5, respectively.

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing us at Support@Innovyze.com

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™
Modeling Made Easy

Reports, Graphs and Output

After a simulation is run, the next step is to view the results. H2OMAP Sewer allows you to view data in report or graph form. You can also compare data between two separate model runs (scenarios), import a model run from another H2OMAP Sewer session or customize a report. Click on any of the following to learn more.

- Output Report Manager - Allows the user to view a report or graph for any *active* or previously run model (output source).
- Available Report Types - Displays the full selection and options available for viewing output in a report format.
- Available Graph Types - Displays the full selection and options available for viewing output in a graph format.
- How to Compare Output Data - A brief tutorial of how to compare hydraulic model results between two models.
- Animation Viewer - Allows the user to display simulation results for EPS timesteps in succession, thereby creating a "movie" of model results.
- Query Report - Allows refined reporting through the use of database queries.
- Output Relate - Creates an external database file that contains the results for a hydraulic simulation. Required for viewing results in third party software applications.
- Output Source - Every time a model is run it is stored internally as an output source for future viewing and comparison of results.
- Simulation Report - Allows the user to have the native analysis model report exported to a specific ASCII text file that differs from the default file.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™
Modeling Made Easy

Rational Method

The Rational method is the most widely used technique for estimation of peak flows from urban and rural drainage basins. The concept of the Rational method is attractive and easy to understand. If rainfall occurs over a watershed at a constant intensity for a period of time that is sufficiently long to produce steady state runoff at a desired design point, then the peak flow rate will be proportional to the product of rainfall intensity and watershed area.

Mathematically, the Rational formula is expressed as:

$$Q = CIA$$

where Q = peak runoff rate (flow unit)

C = runoff coefficient (unitless)

I = rainfall intensity (intensity unit)

A = watershed area (area unit).

The units need to be consistent. For example, if intensity is in ft/s and area is in ft2, the resulting peak flow rate would be in cfs. H2OMAP Sewer Pro allows several intensity unit alternatives (in/hr, mm/hr, in/min, mm/min). The area should be in acres if US Customary units are used, and in square meters for SI units. H2OMAP Sewer Pro converts the units internally to preserve consistency.

TIME OF CONCENTRATION

The Rational method is based on the idea that peak runoff occurs at a given design point when the rainfall duration is sufficiently long so that all locations in the watershed draining to the point contribute flow, and that intensity of the rainfall is constant for that duration. Some locations of the contributing subbasins are hydraulically closer to the design point and may yield runoff to the point sooner than other locations. The Rational method uses the concept of time of concentration to ensure contribution of all locations in the upstream subwatersheds to the design point. Time of concentration, from the perspective of the Rational method and storm runoff (not to be confused with the water quality definition given in preceding sections), is defined as the time it takes for a drop of water falling on the most remote point of upstream subwatersheds to reach the design point. Remoteness refers to hydraulic travel time rather than distance.

There are a number of empirical methods for estimation of time of concentration. The techniques are formulated as a function of subwatershed characteristics such as travel distance and subwatershed slope. H2OMAP Sewer Pro calculates time of concentration as the sum of overland flow travel time and channel (pipe) flow travel time.

$$T_c = t_o + t_c$$

where T_c = time of concentration for the manhole

t_o = overland flow travel time

t_c = channel flow travel time

Pipe flow travel time is calculated using the flow velocity of the pipe and the pipe length. Overland flow travel time is calculated using the method derived by Kirpich, a technique commonly used in USA. Kirpich’s formula is given as:

$$t_o = 0.0078L^{0.77}S^{-0.385}$$

where t_o = overland flow travel time for a subwatershed (in minutes)

L = length of flow path from the remotest spot in the subwatershed (in feet)

S = average slope of the subwatershed

There could be multiple flow paths from upstream subwatershed to a given manhole leading to different flow times. The time of concentration used by H2OMAP Sewer Pro is the longest flow time among different possible flow paths to the manhole. In order to avoid unreasonably low storm duration and unreasonably high rainfall intensities, H2OMAP Sewer Pro allows the user to specify a minimum time of concentration, and uses the maximum of the user-specified minimum time of concentration and an internally calculated time of concentration. The default minimum time of concentration used by the model is 10 minutes.

RAINFALL INTENSITY

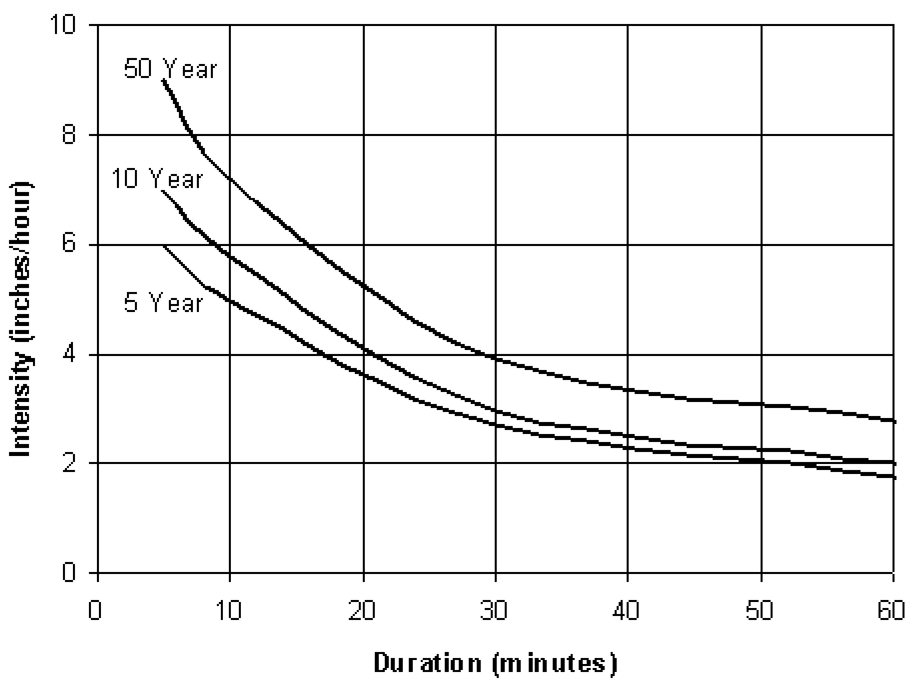
Rainfall intensity is defined as the depth of rain that falls in a given duration of time divided by that duration. It may rain harder (high rainfall intensity) for a short period of time, but the longer a rainfall event lasts, the lower its average rainfall intensity, implying that rainfall intensity is inversely related to duration of a rainfall event.

Intensity of a storm is also a function of return period or frequency of the rainfall occurrence. Intense rainfall events occur less frequently than less intense storms. Return period, also known as recurrence interval, is defined as the expected length of time (e.g., T years) between occurrences of two storm events that exceed a given intensity. Return period does not imply that multiple rainfall events exceeding a given intensity will not occur within T years, nor does it assure occurrence of a storm event exceeding the intensity within T years. All it means is that the rainfall event will occur at an average rate of once in T years.

The higher the return period (i.e., the less frequent the storm event), the higher the intensity of the rainfall event would be. As an example, a storm with 10-year return period represents a rainfall event whose magnitude is expected to be exceeded once in ten years. This implies that the frequency of the storm event is 1/10, or that there is a 10% probability for the intensity of the storm event to be exceeded in any given year.

INTENSITY-DURATION-FREQUENCY CURVE

The relationship between rainfall intensity, rainfall duration, and frequency of occurrence of storm events is often presented for a certain region in the form of an Intensity-Duration-Frequency Curve, commonly referred to as an IDF curve. A sample IDF curve is given below for return periods of 5, 10, and 50 years. InfoSewer calculates the rainfall intensity used by the Rational method based on a user-specified IDF curve. The user should provide an IDF curve for a desired return period. The model derives rainfall intensity from the IDF curve corresponding to the duration equal to time of concentration calculated for the manhole following the techniques described above.



RUNOFF COEFFICIENT

Runoff coefficient is loosely defined as the ratio of runoff to rainfall, and is a function of watershed characteristics including land use, soil type, and slope of the watershed. The value of runoff coefficient ranges between 0.0 and 1.0. A value of 0.0 means that all of the rainfall is lost in the form of abstractions such as infiltration, interception, and evaporation and none of the rainfall is converted to runoff. The value of 1.0 implies that all the rainfall is converted to runoff and is discharged from the watershed. As an example, most of the rain that falls on impervious areas such as pavement and roof would be immediately converted to runoff. A value of C for such land uses is close to 1.0. Runoff coefficient values recommended by the American Society of Civil Engineers and Water Environment Federation for return periods not exceeding 10 years are given below for various land uses, soil types, and slope conditions. For return periods that exceed 10 years, the runoff coefficient from the table should be multiplied by a frequency adjustment factor, Cf , given below.

DESCRIPTION OF AREA	RUNOFF COEFFICIENT
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	
Pavement	
Asphaltic and concrete	0.70 - 0.95
Brick	0.70 - 0.85
Roofs	0.75 - 0.95
Lawns, sandy soil	
Flat, 2%	0.05 - 0.10
Average, 2-7 %	0.10 - 0.15
Steep, 7%	0.15 - 0.20
Lawns, heavy soil	
Flat, 2%	0.13 - 0.17
Average, 2-7 %	0.18 - 0.22
Steep, 7%	0.25 - 0.35

Frequency (Years)	C_f
25	1.1
50	1.2
100	1.25

For a subwatershed composed of multiple land uses, a composite runoff coefficient should be determined by weighting C values of each of the land uses by their corresponding area according to the following equation.

$$C_w = \sum_{i=1}^{N} (C_i A_i)$$

where C_i = runoff coefficient for individual land use in the subwatershed.

A_i = area of the individual land use in the subwatershed.

N = total number of land uses in the subwatershed

ASSUMPTIONS OF THE RATIONAL METHOD

For credible engineering application, the modeler needs to understand the following basic assumptions of the Rational method.

- The rainfall intensity is constant throughout the watershed over a period of time that

equals the time of concentration of the analysis site.

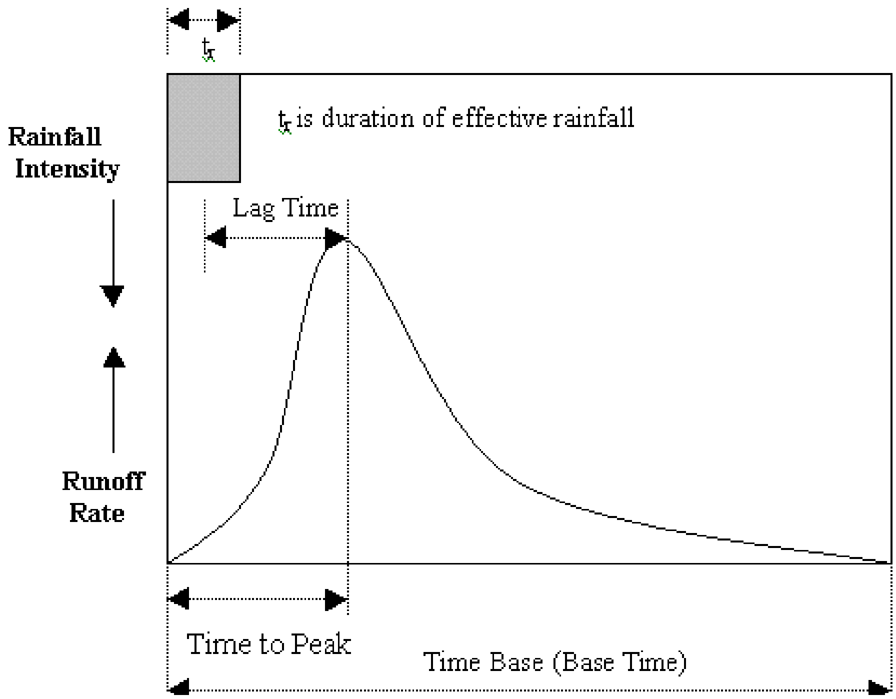
- The runoff coefficient is invariant, regardless of season of the year or intensity

of rainfall.

- Area of the contributing watershed is small (not more than 300 acres).

UNIT HYDROGRAPH

The unit hydrograph method is the most commonly used method of stormwater modeling for applications that require generation of a complete hydrograph with reasonable accuracy. A unit hydrograph is defined as the direct runoff hydrograph resulting from a unit depth of excess (effective) rainfall produced by a storm of uniform intensity and specified duration. Unit hydrograph is a specific type of hydrograph that represents the effects of the physical characteristics of the watershed on the input rainfall excess. A typical unit hydrograph is shown below.



Basic terminology and assumptions of the unit hydrograph method are presented below, followed by a description of the technique used by InfoSewer for generation of unit hydrographs and runoff hydrographs.

EXCESS RAINFALL

During a storm event, not all of the rainfall is converted to runoff. Part of the rainfall is “lost” in the form of deep infiltration (percolation), evaporation, interception, and depression storage. The amount of the precipitation (rainfall) actually reaching the outlet of the watershed or the subwatershed in the form of runoff is known as excess rainfall. Excess rainfall is sometimes called effective rainfall.

DIRECT RUNOFF

Direct runoff is a storm flow resulting from excess rainfall. It is an aggregate of surface runoff and quick subsurface runoff. Surface runoff is an overland flow that occurs when rainfall intensity exceeds infiltration capacity of the soil. Surface runoff flows on the surface of the watershed and through tributary channels to the outlet or to the point of reference such as loading manhole. Quick subsurface runoff is part of an infiltrated rainfall that travels underground and contributes runoff to the location of interest during or soon after the storm event. Unit hydrographs and subsequent runoff hydrographs derived based on unit hydrographs represent direct runoff.

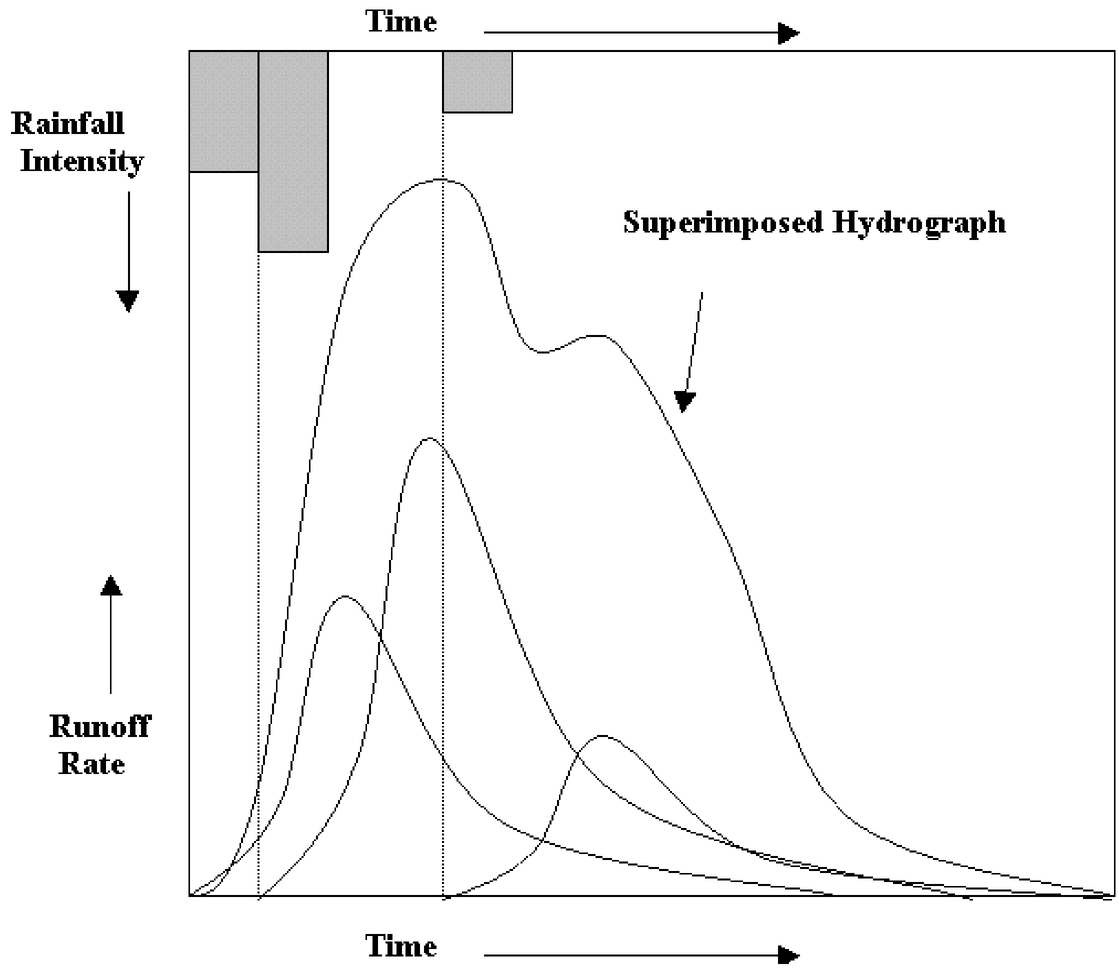
DURATION OF UNIT HYDROGRAPH

This is an important feature of a unit hydrograph, and represents duration of the rainfall excess (represented as tr in the above figure) that produced the unit hydrograph. On a given watershed or subwatershed, all storm events with a given duration of excess rainfall produce direct runoff hydrographs that have the same time to peak and same time base. If the effective duration of a storm differs from that of another on the same basin, the resulting direct runoff hydrograph will have different times to peak and different time bases. Unit hydrograph is named in terms of its duration (i.e., 15-minute unit hydrograph, 1-hour unit hydrograph). As an example, a unit hydrograph derived for a watershed from excess rainfall of a unit depth collected over 15-minute duration is known as a 15-minute unit hydrograph.

BASIC ASSUMPTIONS OF UNIT HYDROGRAPH

- Intensity of excess rainfall is constant within the duration of the excess rainfall for the entire watershed represented by the unit hydrograph.
- One of the fundamental assumptions of the unit hydrograph theory is linearity. It is assumed that an increase in depth of excess rainfall increases ordinates of the direct runoff hydrograph proportionally.
- The unit hydrograph method also assumes that for a given pattern (temporal distribution) of rainfall, the ratio of ordinates of direct runoff hydrograph to depth of effective rainfall is time invariant (e.g., seasonally and within rainfall event(s)).
- Direct runoff hydrograph resulting from a given pattern of rainfall excess can be built by superimposing the unit hydrographs resulting from the separate amounts of rainfall excess occurring in each unit period. This is known as the principle of superposition.

Application of these basic assumptions is illustrated in the figure below.


Time
Rainfall Intensity
Superimposed Hydrograph
Runoff Rate
Time

INFILTRATION LOSSES

During storm events, some of the rainfall is lost in the form of infiltration and depression/retention storage depending on soil type, land use, and topographic conditions of the modeled catchment. InfoSewer estimates part of rainfall that is lost in the form of infiltration and depression/retention storage, and uses the resulting effective (excess) rainfall to determine runoff hydrograph.

The model applies the well known and the widely used Horton's method to determine infiltration loss through pervious portions of the subwatersheds. Horton's infiltration equation is given as:

$$f_t = f_\infty + (f_i - f_\infty)e^{-\alpha t} \quad (36)$$

where, f_t = infiltration rate at time t, in inches/hour.

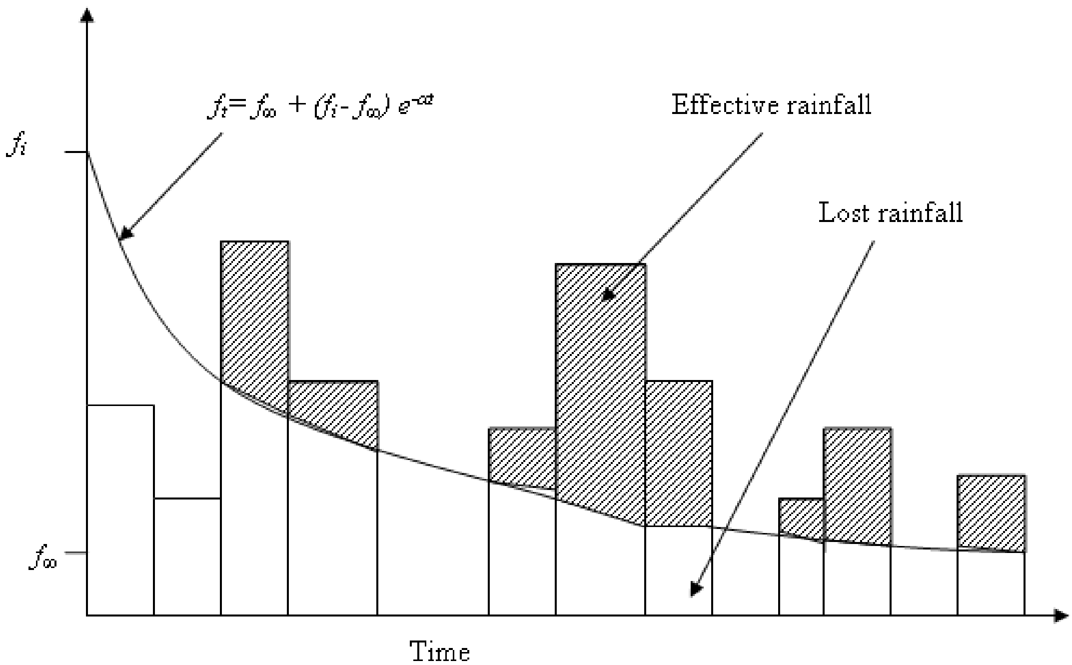
f_i = maximum or initial infiltration rate (at t = 0), in inches/hour. The default value is 3 inches/hour.

f_∞ = final infiltration rate (at t = ∞), in inches/hour. The default value is 0.5 inches/hour.

α = decay coefficient, in per second. The default value is 0.001/second.

t = time from beginning of storm, in seconds.

The actual infiltration at time *t* during the rainfall event(s) is the lesser of rainfall intensity and rate of infiltration, ft. The following figure describes Horton's infiltration model graphically. Only the shaded portion of the rainfall intensities, the region above the infiltration curve, becomes runoff. The remaining rainfall is lost in the form of infiltration.



$f_t = f_\infty + (f_i - f_\infty)\, e^{-\alpha t}$
Effective rainfall
Lost rainfall
f_i
f_∞
Time

INFILTRATION REGENERATION

Infiltration capacity can recover (regenerate) during dry periods. InfoSewer models infiltration regeneration whenever there is dry time steps (when there is no precipitation for the catchment) using the following drying curve. Modeling of infiltration regeneration enables the user to account for the effects of antecedent precipitation on rainfall abstractions, thus enhancing simulation accuracy.

$$f_t = f_i - (f_i - f_\infty)e^{-\alpha_d(t-t_w)} \quad (37)$$

where, f_t = infiltration rate at time t, in inches/hour.

f_i = maximum or initial infiltration rate (at t = 0), in inches/hour. The default value is 3 inches/hour.

f_∞ = final infiltration rate (at t = ∞), in inches/hour. The default value is 0.5 inches/hour.

α_d = decay coefficient for the recovery curve, in per second. It is generally considered to be less than
α, implying a longer drying curve than wetting curve.

t = time from beginning of storm, in seconds.

t_w = hypothetical projected time, internally computed by the model.

DEPRESSION/RETENTION STORAGE LOSS

In addition to infiltration losses, InfoSewer Pro can also model losses due to depression/retention storage. The model allows the user to input separate depression storage loss values for the pervious and for the impervious areas of the catchment. The model internally derives representative depression storage loss value for the catchment by weighting the user input depression storage loss values by respective areas of the pervious and the impervious lands in the catchment.

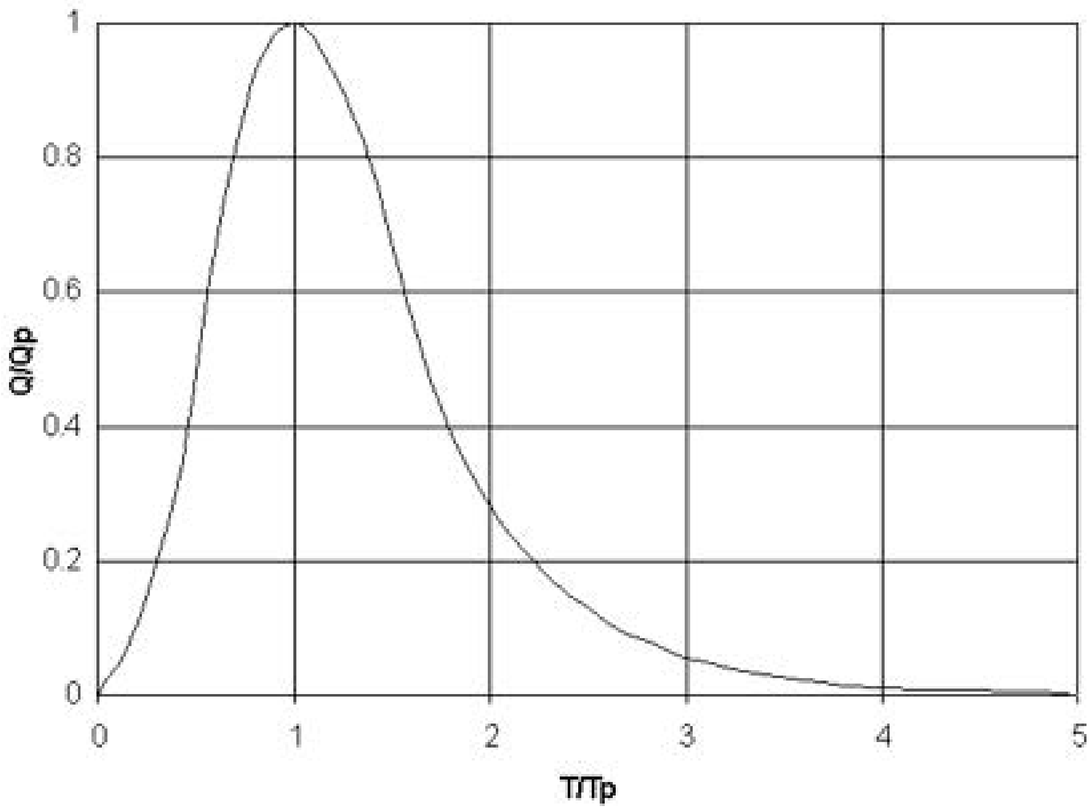
TYPES OF UNIT HYDROGRAPHS

Unit hydrographs could be natural or synthetic. Natural unit hydrographs are derived from observed data, whereas synthetic hydrographs are generated following empirical techniques based on watershed parameters and storm characteristics to simulate the natural unit hydrograph.

If a natural hydrograph is available for a watershed, InfoSewer can utilize it to generate storm runoff at different locations in the collection system. In the absence of a natural unit hydrograph, the model can derive a synthetic unit hydrograph using four different techniques. Each of the unit hydrograph synthesizing approaches is described below.

NRCS (SCS) DIMENSIONLESS UNIT HYDROGRAPH METHOD

The SCS dimensionless unit hydrograph, graphically descried below, is widely used in practice. To generate a tr-hour unit hydrograph for a watershed, time to peak (Tp) and the peak flow rate (Qp) are determined using watershed characteristics.



$$T_p = \frac{t_r}{2} + t_l \quad (38)$$

where t_r is duration of effective rainfall, and t_l is lag time of the watershed. Lag time represents the time from the center of mass of effective rainfall to the time to peak of a unit hydrograph. In other words, lag time is a delay in time, after a brief rain over a watershed, before the runoff reaches its peak. The lag time can either be specified by the user, or can be calculated by the model using the following SCS equation.

$$t_l = L^{0.8}\left(\frac{1000}{CN} - 9\right)^{0.7} \Big/ \left(19000 S^{0.5}\right) \quad (39)$$

where

t_l = lag time of the watershed in hours.

L = hydraulic length of the watershed in ft. This refers to travel distance of water from
the most upstream location of the watershed to the point where the unit hydrograph
is being derived.

CN = the SCS curve number. This is a measure of runoff generating capacity of a watershed, and it depends on the soil, the antecedent moisture condition, the cover,
and the hydrologic conditions of the watershed. Recommended *CN* values for urban areas are
given below. The SCS suggests the *CN* values to be
within 50 and 95.

S = average slope of the watershed.

COVER DISCRIPTION		CURVE NUMBER FOR HYDROLOGIC SOIL GROUP			
Cover Type and Hydrologic Condition	Average Percent Impervious Area	A	B	C	D
Fully Developed Urban Areas (Vegetation Established)					
Open Space (lawns, parks, golf courses, cemetries, etc.):					
Poor Condition (grass cover < 50%)		68	79	86	89
Fair Condition (grass cover 50% to 75%)		49	69	79	84
Good condition (grass cover > 75%)		39	61	74	80
Impervious areas:					
Paved parking lots, roofs, driveways, etc. (excluding right-of-way)		98	98	98	98
Streets and roads					
Paved; curbs, and storm sewers (excluding right-of-way)		98	98	98	98
Paved; open ditches (including right-of-way)		83	89	92	93
Gravel (including right-of-way)		76	85	89	91
Dirt (including right-of-way)		72	82	87	89
Western desert urban areas:					
Natural desert landscaping (pervious areas only)		63	77	85	88
Artificial desert landscaping (impervious weed barrier, desert shrub with I- to 2-inch sand or gravel mulch and basin borders)		96	96	96	96
Urban districts:					
Commercial and business	85	89	92	94	95
Industrial	72	81	88	91	93
Residential districts by average lot size:					
1/8 acre or less (town house)	65	77	85	90	92
1/4 acre	38	61	75	83	87
1/3 acre	30	57	72	81	86
1/2 acre	25	54	70	80	85
1 acre	20	51	68	79	84
2 acres	12	46	65	77	82
Developing urban areas:					
Newly graded areas (pervious areas only, no vegetation)		77	86	91	94

The peak flow rate is calculated as:

$$Q_p = \frac{484A}{T_p} \quad (40)$$

where

Q_p = peak flow rate in cfs.

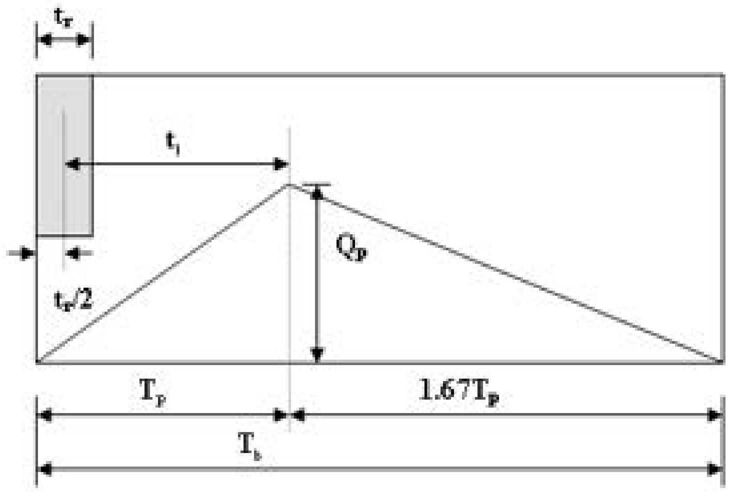
A = area of the watershed, in square miles, draining to the location of the unit hydrograph.

T_p = time to peak of the unit hydrograph in hours.

Once T_p and Q_p are known, actual time and flow rate ordinates of the t_r-hour unit hydrograph are determined by multiplying the dimensionless time (T/T_p) and the dimensionless flow rate ordinates (Q/Q_p) by T_p and Q_p , respectively.

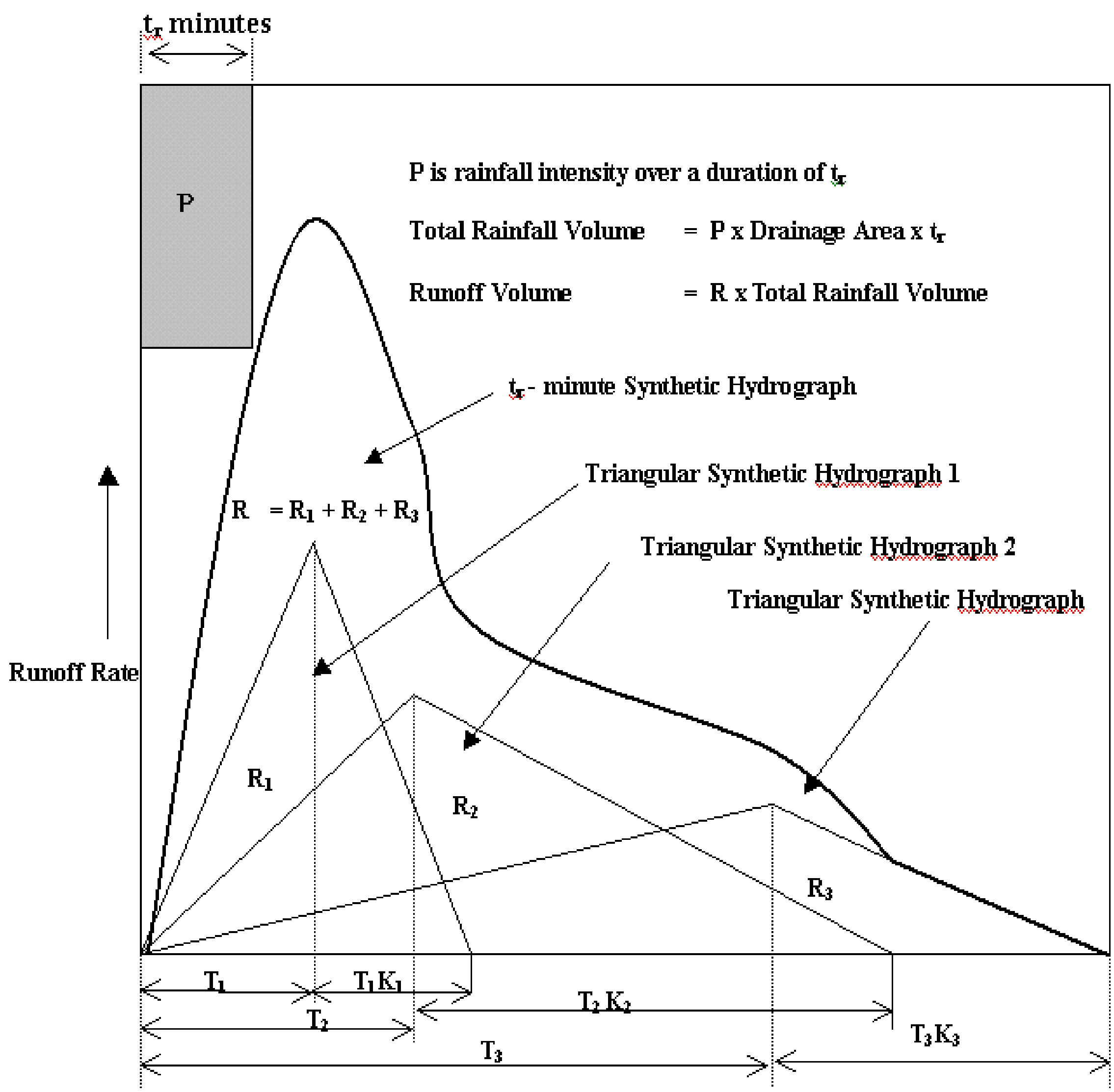
NRCS (SCS) TRIANGULAR UNIT HYDROGRAPH METHOD

The SCS has also developed a triangular unit hydrograph (see figure below) that is an approximation to the dimensionless unit hydrograph described above. The triangular unit hydrograph is entirely defined in terms of three points, Q_p, T_p, and T_b. The lag time, time to peak, and peak flow rate are calculated using the same equations as for the dimensionless unit hydrograph.



THE TRI-TRAINGLE METHOD

Tri-triangle is one of the techniques used by InfoSewer for synthesizing a hydrograph. The triangular unit hydrograph technique developed by the U.S. Soil Conservation Service (now known as Natural Resources Conservation Service NRCS) is one of the commonly used methods. As described above NRCS's triangular synthetic unit method uses a single triangle to represent a unit hydrograph. However, shape of unit hydrograph is too complex to be well captured by a single triangle. InfoSewer applies up to three triangular synthetic hydrographs, as the name implies, to simulate a hydrograph. The total synthetic hydrograph is the result of aggregating corresponding ordinates of the three triangular hydrographs. Each of these three triangular hydrographs has its own characteristic parameters, namely time to peak, recession constant, and fraction of an effective rainfall volume allocated to the triangle. The technique is graphically illustrated below.



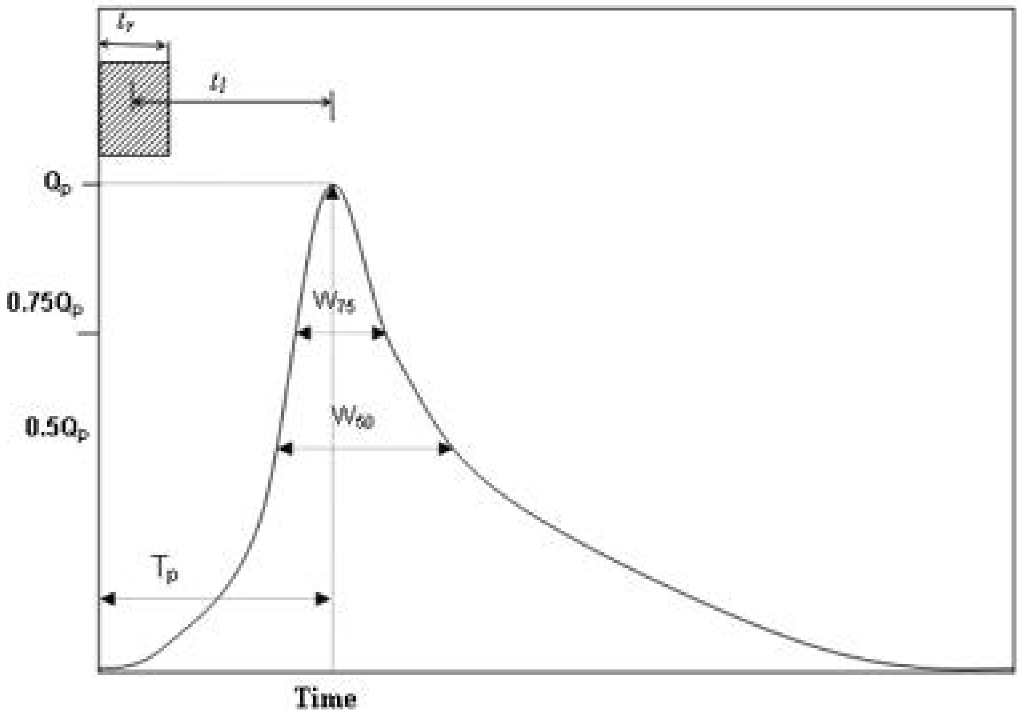
R_1, R_2, and R_3 are fractions of excess rainfall volume, R, allocated to triangular synthetic hydrographs 1, 2, and 3 respectively. T_i and K_i are time to peak and recession constants of the triangles, respectively. The three triangular synthetic hydrographs are conceptual representations of different components of direct runoff or rainfall-dependent infiltration/inflow. The first triangle represents rapidly responding (fast) components, such as contributions from pavements and rooftops, or direct inflow or rapid infiltration into separate sewer systems. The third triangle represents slow runoff components such as ground water contributions or slow infiltration into separate sewers. The second triangle represents runoff or infiltration with a medium time response.

Time to peak value of the first triangle typically varies between 1 and 2 hours, depending on the size of the tributary area in question. The second triangle takes

T values ranging from 4 to 8 hours. The third triangle parameter varies greatly depending on the infiltration characteristics of the system being modeled, and has a T value generally between 10 and 24 hours. The default values used for T_1, T_2, and T_3 are 1, 4, and 12 hours respectively. Values of K for the first triangle typically ranges between 2 and 3. The second and the third triangles take K values from 2 to 4. The default values of K are 2, 3, and 3 for the first, the second, and the third triangles, respectively.

THE COLORADO URBAN HYDROGRAPH PROCEDURE

The Colorado Urban Hydrograph Procedure (CUHP) uses the equations and procedures presented in the Urban Drainage Criteria Manual (USDCM) of the Urban Drainage and Flood Control District (UDFCD). Shape of the CUHP synthetic unit hydrograph is determined using the following equations that relate unit hydrograph parameters to catchment properties.



Lag time (t_l) of the watershed (catchment), defined as the time from the center of unit storm duration to the peak of the unit hydrograph, is determined as:

$$t_l = C_t \left(\frac{(L \cdot L_{ca})}{\sqrt{S}} \right)^{0.48} \quad (41)$$

where t_l = lag time in hours.

L = length along the drainageway path from study point to the most upstream limits of the catchment in miles.

L_{ca} = length along stream from study point to a point along stream adjacent to the centroid of the catchment in miles.

S = length weighted average slope of catchment along draiangeway path to upstream limits of the
catchment.

C_t = time to peak coefficient.

Once the lag time is known, time to peak (T_p) of the unit hydrograph could be determined by adding $0.5t_r$ to the lag time in consistent units.

Peak flow rate, Q_p, of the unit hydrograph is calculated as:

$$Q_p = \frac{640 C_p A}{T_p} \quad (42)$$

where Q_p = peak flow rate of the unit hydrograph, in cfs.

A = area of the catchment, in square miles.

C_p = unit hydrograph peaking coefficient, and is determined as:

$$C_p = P \cdot C_t \cdot A^{0.15} \quad (43)$$

where P = peaking parameter.

C_t and P are defined in terms of percent impervious (I_a) of the catchment as:

$$C_t = aI_a^2 + bI_a + c \quad (44)$$

$$P = dI_a^2 + eI_a + f \quad (45)$$

The coefficients a, b, c, d, e, and f are defined in terms of I_a in the following table.

Ia	***a***	***b***	***c***	***d***	***e***	***f***
***Ia* ≤ 10**	0.0	-0.00371	0.163	0.00245	-0.012	2.16
***10 < Ia* ≤ 40**	0.000023	-0.00224	0.146	0.00245	-0.012	2.16
***Ia* > 40**	0.0000033	-0.000801	0.120	-0.00091	0.228	-2.06

The widths of the unit hydrograph at 50% and 75% of the peak are estimated as:

$$W_{50} = \frac{500}{\left(\frac{Q_p}{A}\right)} \quad (46)$$

$$W_{75} = \frac{260}{\left(\frac{Q_p}{A}\right)} \quad (47)$$

where W_{50} = width of the unit hydrograph at 50% of the peak, in hours.

W_{75} = width of the unit hydrograph at 75% of the peak, in hours.

Q_p = peak flow rate, in cfs.

A = catchment area, in square miles.

It is recommended that a unit hydrograph duration of 5-minute be used for studies that apply the CUHP. The maximum recommended drainage area (catchment size) for any single CUHP unit hydrograph is 5 square miles. Whenever a larger watershed is studied, it needs to be subdivided into subcatchments of 5-square miles or less. For this synthetic unit hydrograph method, the minimum drainage area should be 90 acres. For catchments smaller than 90 acres, other unit hydrograph generation mechanisms should be used.

GENERATION OF RUNOFF HYDROGRAPH

InfoSewer generates a direct runoff hydrograph resulting from single or multiple storm events using a unit hydrograph synthesized according to any of the three techniques described above, or from a user-specified natural unit hydrograph. The model relies on a user input hyetograph (i.e., rainfall intensity versus time graph). The model can derive effective rainfall from user supplied total rainfall hyetographs.

An entire watershed of a collection system may not be well represented by a single unit hydrograph owing to variability in topography, land use, and soil characteristics of the subwatersheds. In the case of separate sewer systems, the magnitude and type of sources of infiltration/inflow will vary by subbasin. Accordingly, InfoSewer Pro allows usage of multiple unit hydrographs, each representing part of the watershed being modeled. The unit hydrographs could be of any duration (e.g., 15-minute, 1-hr), but the duration must be the same for all the unit hydrographs involved in modeling of the watershed.

To derive the storm hydrographs, InfoSewer Pro applies the basic assumptions of unit hydrograph theory described above (i.e., linearity, time invariance, and the principle of superposition). Storm events are assumed to have constant intensity over the duration of the unit hydrograph. Excess rainfall resulting from single or multiple storm events is discretized at intervals of unit hydrograph duration. For example, if a sewer collection system is being modeled using a 15-minute unit hydrograph, and if duration of the excess rainfall under investigation is 1-hour, the rainfall duration will be divided into four 15-minute rainfall events of constant intensity. This discretization approach, along with the unit hydrograph assumptions of linearity, time invariance, and superposition, enables InfoSewer to simulate storm runoff hydrograph at any location (e.g., loading manhole) throughout the collection system for any number of storm events. During generation of runoff hydrographs for locations other than the site where the unit hydrograph is originally derived, ordinates of the unit hydrographs are adjusted according to the ratio of drainage area of the two locations.

Once the storm hydrographs for every loading manhole in the collection system are known, the storm load will be added to other loading types such as sanitary loads, if any, and will be routed through the collection system using the powerful Muskingum-Cunge’s dynamic flow routing algorithm.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™
Modeling Made Easy

Steady State Modeling of Stormwater

Steady-state analysis and design of storm sewers is usually accomplished under worst (i.e., peak) runoff conditions to test transporting capacity of the network without causing major flooding or overflows. The Rational method is the most widely used technique for estimation of peak flow from urban and rural drainage basins, and is the method used by InfoSewer during steady-state analysis and design .

The Rational method is based on the premise that if rainfall occurs over a watershed at a constant intensity for a period of time that is sufficiently long to produce steady state runoff at a desired design point, such as at manhole locations, then the peak flow rate will be proportional to the product of rainfall intensity and watershed area. Peak runoff is assumed to occur when all locations in the subwatersheds draining to the point contribute flow. The rainfall intensity is considered constant over a duration commonly known as time of concentration.

Mathematically, the Rational method is expressed as:

$$Q = C\ i\ A$$

where Q = peak runoff rate (flow unit)

C = runoff coefficient (unitless)

i = rainfall intensity (intensity unit)

A = watershed area (area unit).

Time of concentration for a loading manhole is calculated as the sum of overland flow travel time and pipe flow travel time. Overland flow travel time is estimated using Kirpich's equation as shown below.

$$t_o = 0.0078\ L^{.77} S^{-.385}$$

where t_o = overland flow travel time for a subwatershed (in minutes)

L = length of flow path from the remotest spot in the subwatershed (in feet)

S = average slope of the subwatershed

Detailed description of the theoretical aspect of the Rational method is given in InfoSewer Theory section.

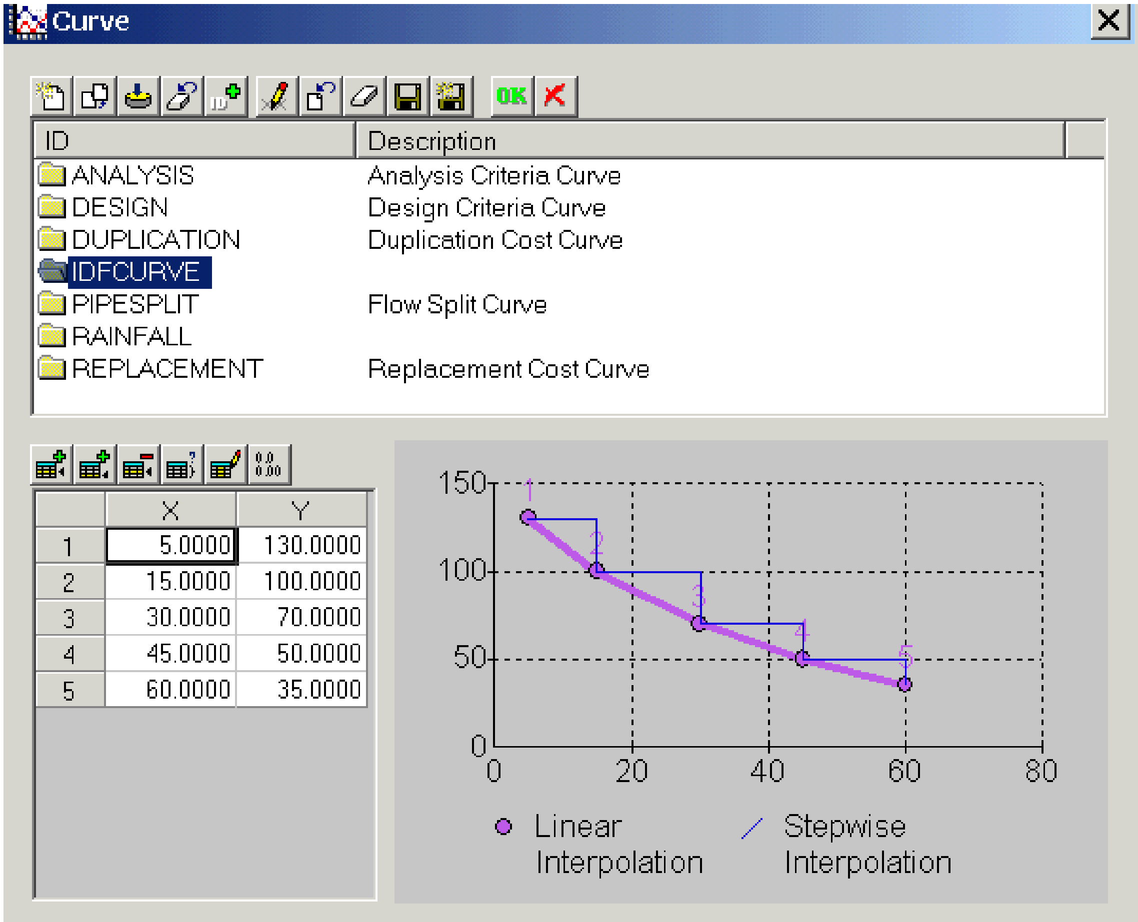
InfoSewer could be used for modeling of sanitary sewer systems, storm sewer systems, or combined sewer systems. A user interested in performing steady state simulation of storm and/or combined sewer systems needs to provide the following data for every loading manhole in addition to the data required for modeling of sanitary sewers.

- Runoff coefficient (*C*)
- Length of flow path from the remotest spot in the subwatershed (*L*)
- Average slope of the subwatershed (*S*)
- Area of the subwatershed (*A)*

If US Customary units are used, area and length should be given in acres and in feet , respectively. If SI units are used, area should be in square meters, and length should be in meters.

These manhole data could be supplied to the model either through the attribute browser or by using the manhole hydraulic/hydrologic data base table.

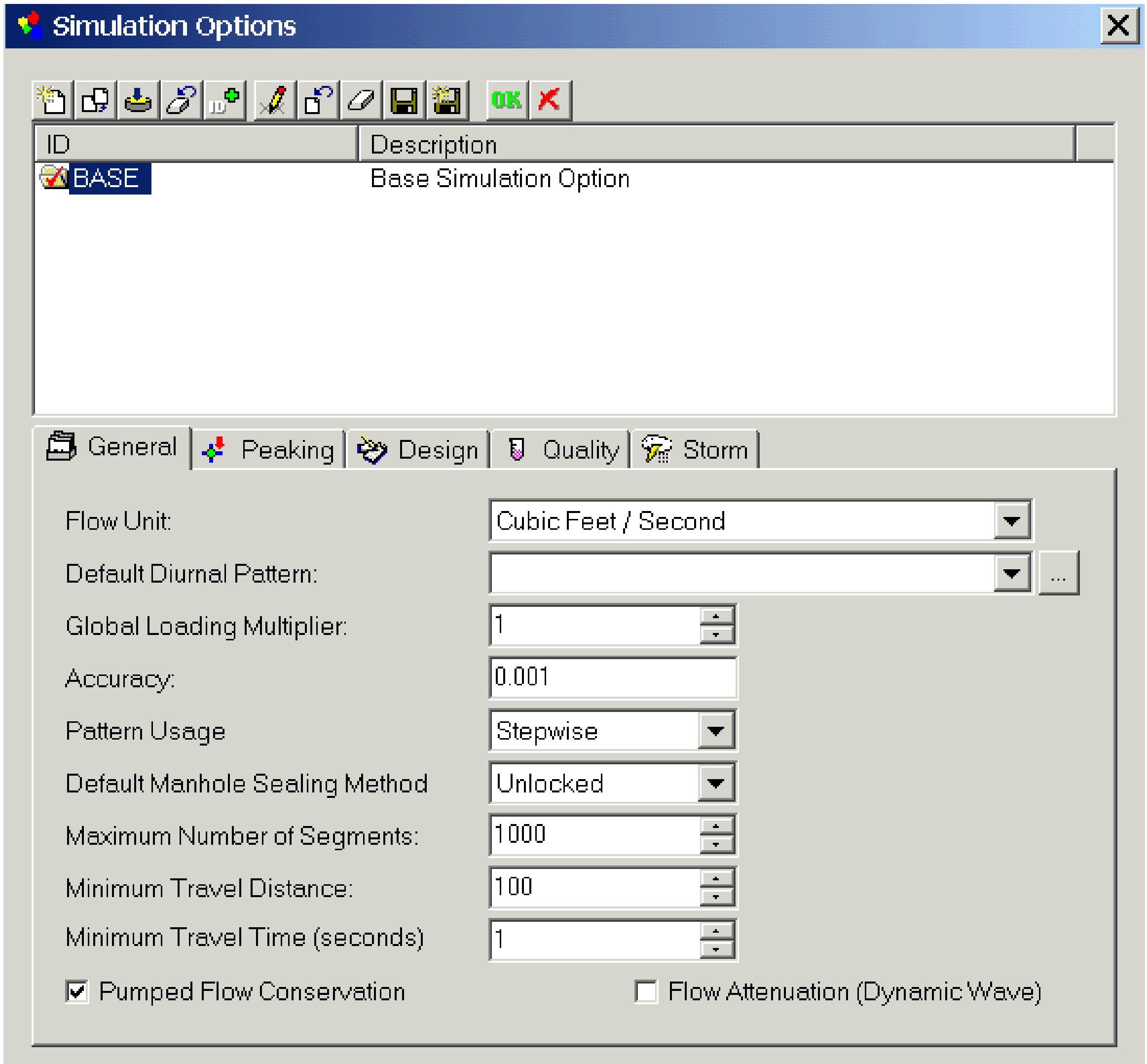
In addition to the above mentioned data, the user has to provide an Intensity-Duration-Frequency curve of the watershed for a desired return period. An IDF data is accepted in the form of a curve, where the y-axis represents rainfall intensity and the x-axis represents rainfall duration. A typical IDF curve is shown below. The model accepts rainfall intensity in any of the following units: inches/hour, mm/hour, inches/minute, or mm/minute. Rainfall duration could be in second, in minute, or in hour.



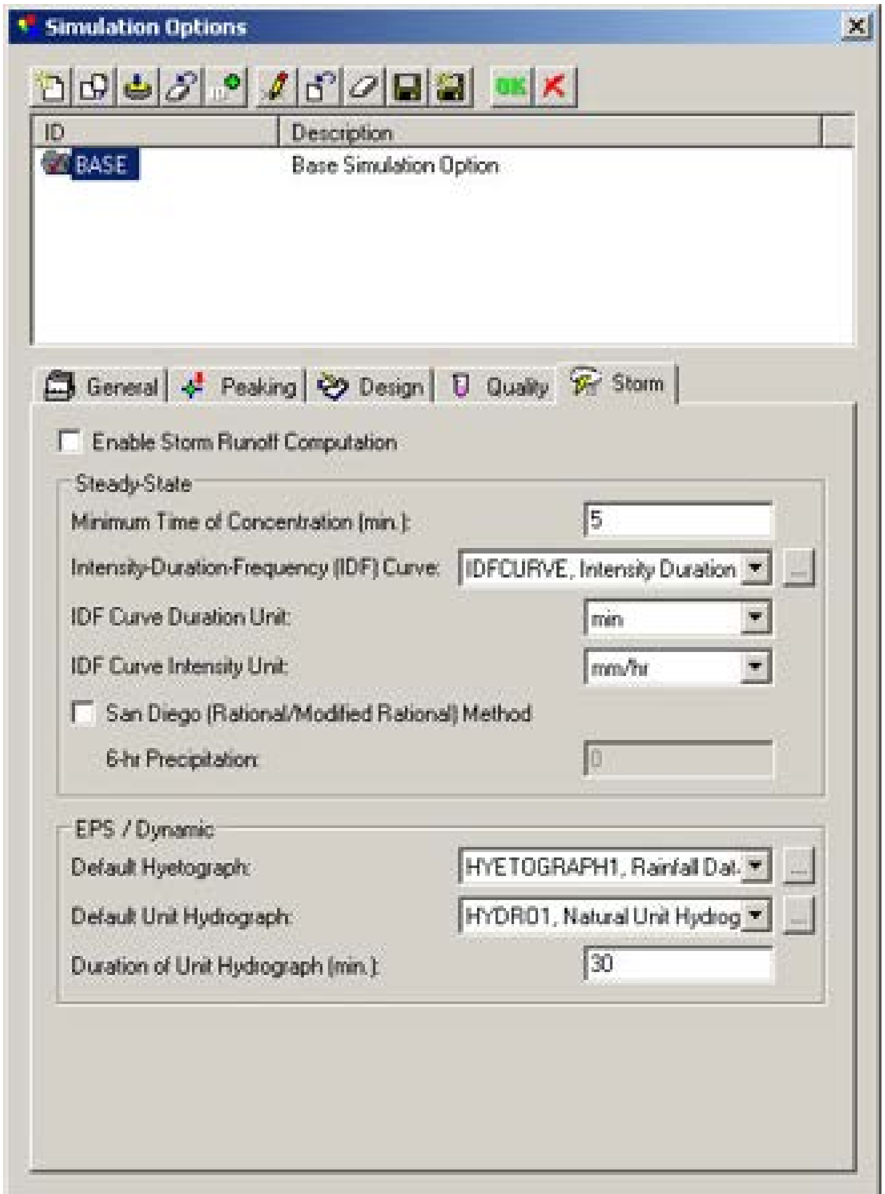
In order to avoid unreasonably low storm duration and unreasonably high rainfall intensities, InfoSewerPro allows the user to specify a minimum time of concentration to be used by the Rational method. The model uses the maximum of the user specified minimum time of concentration and an internally calculated time of concentration. The default value used by the model for the minimum time of concentration is 10 minutes.

To supply an IDF curve and its intensity and duration units, and the minimum time of concentration data, the modeler needs to follow the following procedure.

- Go to the simulation options dialog box shown below either from the control center or using the run manager.



- Click on  icon . This initiates the following storm dialog box.



- Check 

option to activate storm simulation.
- Supply data for minimum time of concentration, IDF Curve, and duration and intensity units of the IDF curve.
- San Diego Rational/Modified Rational Method uses the San Diego County rational method and modified rational method to calculate peak storm flow for storm drains.
- 6-hr Precipitation is the six hour duration rainfall depth for the design return period.
- Save the changes and close the dialog box.

- If the loading manhole data required for steady state simulation of stormwater, as described above, are already provided, the desired simulation (i.e., steady state analysis or steady state design) could be performed from the run manager.
- If the type of the simulation is steady state analysis, the model reports peak storm load for every loading manhole and pipe in the collection system. Results are available from the attribute browser.
- If the type of the simulation is steady state analysis, the model reports peak storm load for every loading manhole and pipe in the collection system. Results are available from the attribute browser.

Note The data described in this section are those pertinent to stormwater modeling. The user should also provide the data required to model sanitary sewer systems (i.e., data for the physical and the non-physical components of the collection system).

Other Related Topics - Dynamic Modeling, Hyetograph, Unit Hydrograph

InfoSewer™

Modeling Made Easy

Dynamic (EPS) Modeling of Stormwater

InfoSewer applies the unit hydrograph approach for generation of complete storm hydrographs during dynamic simulations. Unit hydrographs could be natural or synthetic. Natural unit hydrographs are derived from observed data, where as synthetic hydrographs are generated following empirical or conceptual techniques based on watershed parameters and storm characteristics.

If a natural hydrograph is available for a watershed, InfoSewer can utilize it to generate storm runoff at different locations of the collection system. In the absence of natural unit hydrograph, the model can derive a synthetic unit hydrograph using four different techniques, namely the SCS dimensionless unit hydrograph, the SCS triangular unit hydrograph, the tri-triangle method, and the Colorado Urban Hydrograph Procedure.

InfoSewer requires the following data for dynamic modeling of stormwater. Click on each of the links to know how each of these data is supplied to the model.

- Unit Hydrograph Data
- Hyetograph
- Rainfall Loss Data
- General Data

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

4.5 Stormwater Modeling

InfoSewer adds comprehensive rainfall-runoff and infiltration/inflow modeling capability on top of the sanitary sewer simulation features provided by H2OMAP Sewer, equipping the industry with a single powerful tool that can be used for planning, design, and operational management of sanitary sewer systems, storm sewer systems, and combined sewer systems. InfoSewerH2OMAP Sewer shares all the features described above in relation to H2OMAP Sewer including the methodologies used to model the physical and the non-physical components of a collection system, steady state and dynamic analysis of sanitary loads, steady state design, dynamic routing of flows through the collection system, calculation of HGLs, and water quality modeling. This section presents the theory pertinent to InfoSewer , which is stormwater modeling.

Determination of stormwater runoff resulting from rainfall event(s) is the most important component of stormwater management, and is commonly known as rainfall-runoff modeling. Similar modeling techniques are applicable to the determination of rainfall-dependent infiltration/inflow in separate sanitary sewer systems. There are various rainfall-runoff-modeling techniques. The choice of the method to use depends, among other considerations, on the type of analysis (e.g., steady state or dynamic simulation) and the information required (e.g., peak flow or complete hydrograph). Peak flow information is appropriate for steady state simulation of sewer systems whereas knowledge of the complete hydrograph is essential for dynamic simulations. For estimation of peak flow during steady state simulations, InfoSewer applies the Rational method, which is a very popular tool of choice used by many practicing engineers across the globe. For dynamic simulation, complete hydrographs are derived based on the practical and highly effective unit hydrograph theory. The unit hydrograph can be observed or synthetic. This method allows the entire hydrograph (including peak and volume) resulting from an actual or design storm event to be adequately simulated. The following sections describe the Rational method and the unit hydrograph techniques incorporated into InfoSewerH2OMAP Sewer .

RATIONAL METHOD

The Rational method is the most widely used technique for estimation of peak flows from urban and rural drainage basins. The concept of the Rational method is attractive and easy to understand. If rainfall occurs over a watershed at a constant intensity for a period of time that is sufficiently long to produce steady state runoff at a desired design point, then the peak flow rate will be proportional to the product of rainfall intensity and watershed area.

Mathematically, the Rational formula is expressed as:

$$Q = CIA \quad (32)$$

where Q = peak runoff rate (flow unit)

C = runoff coefficient (unitless)

I = rainfall intensity (intensity unit)

A = watershed area (area unit)

The units need to be consistent. For example, if intensity is in ft/s and area is in ft2, the resulting peak flow rate would be in cfs. InfoSewerH2OMAP Sewer allows several intensity unit alternatives (in/hr, mm/hr, in/min, mm/min). The area should be in acres if US Customary units are used, and in square meters for SI units. InfoSewer converts the units internally to preserve consistency.

TIME OF CONCENTRATION

The Rational method is based on the idea that peak runoff occurs at a given design point when the rainfall duration is sufficiently long so that all locations in the watershed draining to the point contribute flow, and that intensity of the rainfall is constant for that duration. Some locations of the contributing subbasins are hydraulically closer to the design point and may yield runoff to the point sooner than other locations. The Rational method uses the concept of time of concentration to ensure contribution of all locations in the upstream subwatersheds to the design point. Time of concentration, from the perspective of the Rational method and storm runoff (not to be confused with the water quality definition given in preceding sections), is defined as the time it takes for a drop of water falling on the most remote point of upstream subwatersheds to reach the design point. Remoteness refers to hydraulic travel time rather than distance.

There are a number of empirical methods for estimation of time of concentration. The techniques are formulated as a function of subwatershed characteristics such as travel distance and subwatershed slope. InfoSewer calculates time of concentration as the sum of overland flow travel time and channel (pipe) flow travel time.

$$T_c = t_o + t_c \quad (33)$$

where T_C = time of concentration for the manhole

t_O = overland flow travel time

t_C = channel flow travel time

Pipe flow travel time is calculated using the flow velocity of the pipe and the pipe length. Overland flow travel time is calculated using the method derived by Kirpich, a technique commonly used in USA. Kirpich's formula is given as:

$$t_o = 0.0078L^{0.77}S^{-0.385} \quad (34)$$

where t_O = overland flow travel time for a subwatershed (in minutes)

L = length of flow path from the remotest spot in the subwatershed (in feet)

S = average slope of the subwatershed

There could be multiple flow paths from upstream subwatershed to a given manhole leading to different flow times. The time of concentration used by InfoSewer is the longest flow time among different possible flow paths to the manhole. In order to avoid unreasonably low storm duration and unreasonably high rainfall intensities, InfoSewerH2OMAP Sewer allows the user to specify a minimum time of concentration, and uses the maximum of the user-specified minimum time of concentration and an internally calculated time of concentration. The default minimum time of concentration used by the model is 10 minutes.

RAINFALL INTENSITY

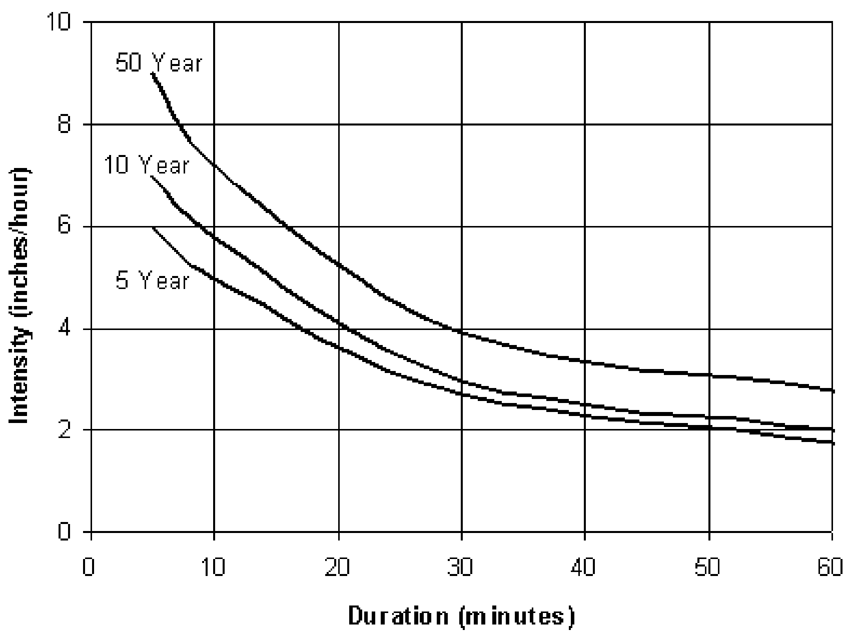
Rainfall intensity is defined as the depth of rain that falls in a given duration of time divided by that duration. It may rain harder (high rainfall intensity) for a short period of time, but the longer a rainfall event lasts, the lower its average rainfall intensity, implying that rainfall intensity is inversely related to duration of a rainfall event.

Intensity of a storm is also a function of return period or frequency of the rainfall occurrence. Intense rainfall events occur less frequently than less intense storms. Return period, also known as recurrence interval, is defined as the expected length of time (e.g., T years) between occurrences of two storm events that exceed a given intensity. Return period does not imply that multiple rainfall events exceeding a given intensity will not occur within T years, nor does it assure occurrence of a storm event exceeding the intensity within T years. All it means is that the rainfall event will occur at an average rate of once in T years.

The higher the return period (i.e., the less frequent the storm event), the higher the intensity of the rainfall event would be. As an example, a storm with 10-year return period represents a rainfall event whose magnitude is expected to be exceeded once in ten years. This implies that the frequency of the storm event is 1/10, or that there is a 10% probability for the intensity of the storm event to be exceeded in any given year.

INTENSITY-DURATION-FREQUENCY CURVE

The relationship between rainfall intensity, rainfall duration, and frequency of occurrence of storm events is often presented for a certain region in the form of an Intensity-Duration-Frequency Curve, commonly referred to as an IDF curve. A sample IDF curve is given below for return periods of 5, 10, and 50 years. InfoSewer calculates the rainfall intensity used by the Rational method based on a user-specified IDF curve. The user should provide an IDF curve for a desired return period. The model derives rainfall intensity from the IDF curve corresponding to the duration equal to time of concentration calculated for the manhole following the techniques described above.



RUNOFF COEFFICIENT

Runoff coefficient is loosely defined as the ratio of runoff to rainfall, and is a function of watershed characteristics including land use, soil type, and slope of the watershed. The value of runoff coefficient ranges between 0.0 and 1.0. A value of 0.0 means that all of the rainfall is lost in the form of abstractions such as infiltration, interception, and evaporation and none of the rainfall is converted to runoff. The value of 1.0 implies that all the rainfall is converted to runoff and is discharged from the watershed. As an example, most of the rain that falls on impervious areas such as pavement and roof would be immediately converted to runoff. A value of C for such land uses is close to 1.0. Runoff coefficient values recommended by the American Society of Civil Engineers and Water Environment Federation for return periods not exceeding 10 years are given below for various land uses, soil types, and slope conditions. For return periods that exceed 10 years, the runoff coefficient from the table should be multiplied by a frequency adjustment factor, C_f , given below.

DESCRIPTION OF AREA	RUNOFF COEFFICIENT
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	
Pavement	
Asphaltic and concrete	0.70 - 0.95
Brick	0.70 - 0.85
Roofs	0.75 - 0.95
Lawns, sandy soil	
Flat, 2%	0.05 - 0.10
Average, 2-7 %	0.10 - 0.15
Steep, 7%	0.15 - 0.20
Lawns, heavy soil	
Flat, 2%	0.13 - 0.17
Average, 2-7 %	0.18 - 0.22
Steep, 7%	0.25 - 0.35

Frequency (Years)	C_f
25	1.1
50	1.2
100	1.25

For a subwatershed composed of multiple land uses, a composite runoff coefficient should be determined by weighting C values of each of the land uses by their corresponding area according to equation 35.

$$C_w = \sum_{i=1}^{N}(C_i A_i) \Bigg/ \sum_{i=1}^{N}(A_i) \tag{35}$$

where C_i = runoff coefficient for individual land use in the subwatershed.

A_i = area of the individual land use in the subwatershed.

N = total number of land uses in the subwatershed

ASSUMPTIONS OF THE RATIONAL METHOD

For credible engineering application, the modeler needs to understand the following basic assumptions of the Rational method.

- The rainfall intensity is constant throughout the watershed over a period of time that

equals the time of concentration of the analysis site.

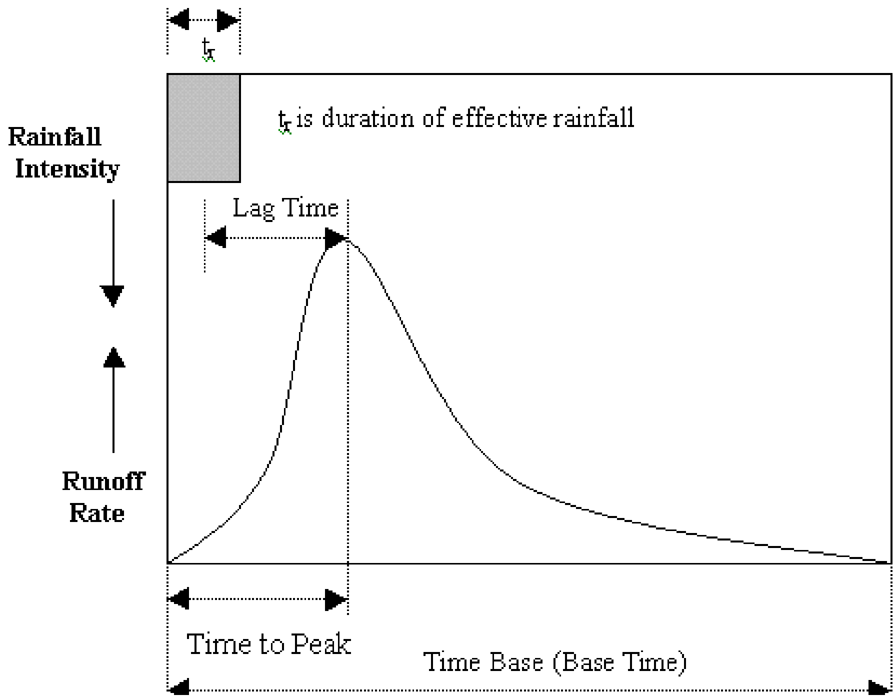
- The runoff coefficient is invariant, regardless of season of the year or intensity

of rainfall.

- Area of the contributing watershed is small (not more than 300 acres).

UNIT HYDROGRAPH

The unit hydrograph method is the most commonly used method of stormwater modeling for applications that require generation of a complete hydrograph with reasonable accuracy. A unit hydrograph is defined as the direct runoff hydrograph resulting from a unit depth of excess (effective) rainfall produced by a storm of uniform intensity and specified duration. Unit hydrograph is a specific type of hydrograph that represents the effects of the physical characteristics of the watershed on the input rainfall excess. A typical unit hydrograph is shown below.



Basic terminology and assumptions of the unit hydrograph method are presented below, followed by a description of the technique used by InfoSewer for generation of unit hydrographs and runoff hydrographs.

EXCESS RAINFALL

During a storm event, not all of the rainfall is converted to runoff. Part of the rainfall is “lost” in the form of deep infiltration (percolation), evaporation, interception, and depression storage. The amount of the precipitation (rainfall) actually reaching the outlet of the watershed or the subwatershed in the form of runoff is known as excess rainfall. Excess rainfall is sometimes called effective rainfall.

DIRECT RUNOFF

Direct runoff is a storm flow resulting from excess rainfall. It is an aggregate of surface runoff and quick subsurface runoff. Surface runoff is an overland flow that occurs when rainfall intensity exceeds infiltration capacity of the soil. Surface runoff flows on the surface of the watershed and through tributary channels to the outlet or to the point of reference such as loading manhole. Quick subsurface runoff is part of an infiltrated rainfall that travels underground and contributes runoff to the location of interest during or soon after the storm event. Unit hydrographs and subsequent runoff hydrographs derived based on unit hydrographs represent direct runoff.

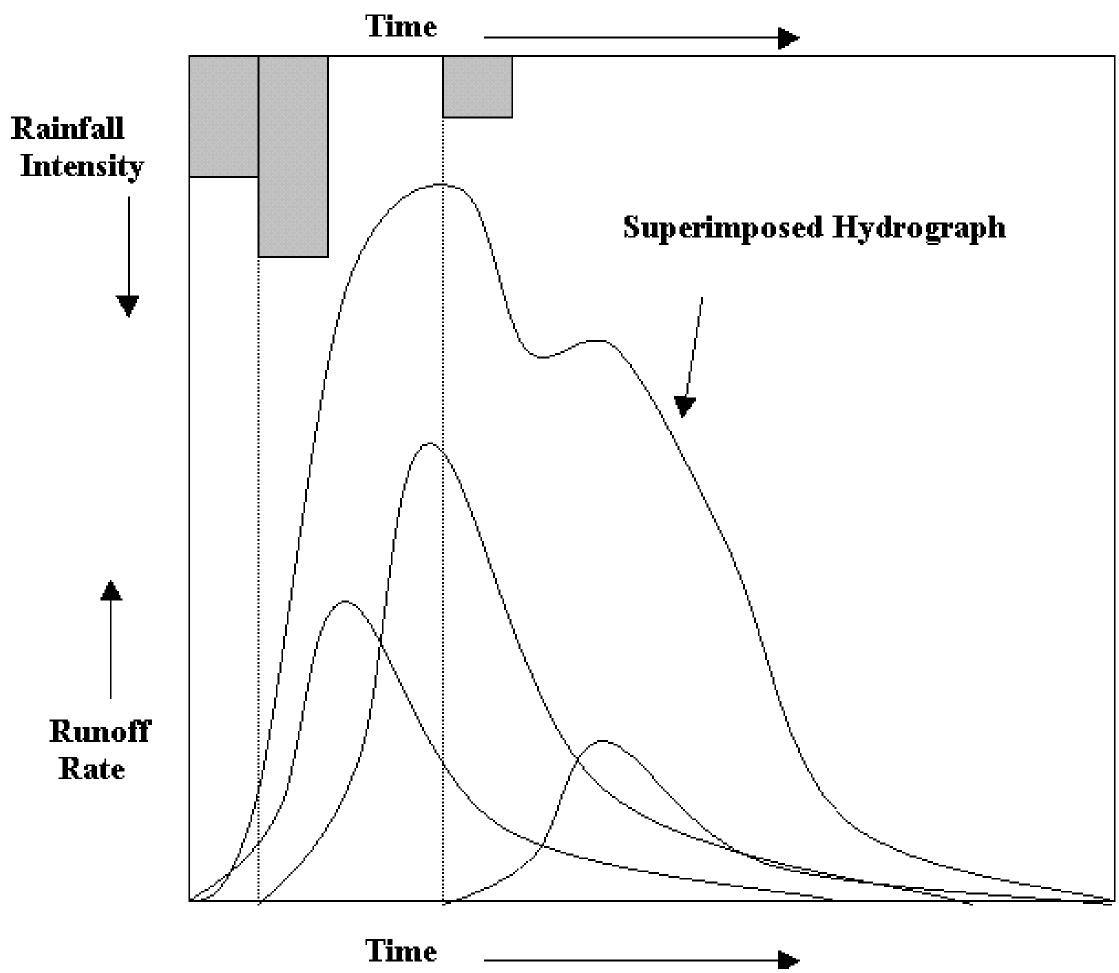
DURATION OF UNIT HYDROGRAPH

This is an important feature of a unit hydrograph, and represents duration of the rainfall excess (represented as tr in the above figure) that produced the unit hydrograph. On a given watershed or subwatershed, all storm events with a given duration of excess rainfall produce direct runoff hydrographs that have the same time to peak and same time base. If the effective duration of a storm differs from that of another on the same basin, the resulting direct runoff hydrograph will have different times to peak and different time bases. Unit hydrograph is named in terms of its duration (i.e., 15-minute unit hydrograph, 1-hour unit hydrograph). As an example, a unit hydrograph derived for a watershed from excess rainfall of a unit depth collected over 15-minute duration is known as a 15-minute unit hydrograph.

BASIC ASSUMPTIONS OF UNIT HYDROGRAPH

- Intensity of excess rainfall is constant within the duration of the excess rainfall for the entire watershed represented by the unit hydrograph.
- One of the fundamental assumptions of the unit hydrograph theory is linearity. It is assumed that an increase in depth of excess rainfall increases ordinates of the direct runoff hydrograph proportionally.
- The unit hydrograph method also assumes that for a given pattern (temporal distribution) of rainfall, the ratio of ordinates of direct runoff hydrograph to depth of effective rainfall is time invariant (e.g., seasonally and within rainfall event(s)).
- Direct runoff hydrograph resulting from a given pattern of rainfall excess can be built by superimposing the unit hydrographs resulting from the separate amounts of rainfall excess occurring in each unit period. This is known as the principle of superposition.

Application of these basic assumptions is illustrated in the figure below.


Time
Rainfall Intensity
Superimposed Hydrograph
Runoff Rate
Time

INFILTRATION LOSSES

During storm events, some of the rainfall is lost in the form of infiltration and depression/retention storage depending on soil type, land use, and topographic conditions of the modeled catchment. InfoSewer estimates part of rainfall that is lost in the form of infiltration and depression/retention storage, and uses the resulting effective (excess) rainfall to determine runoff hydrograph.

The model applies the well known and the widely used Horton's method to determine infiltration loss through pervious portions of the subwatersheds. Horton's infiltration equation is given as:

$$f_t = f_\infty + (f_i - f_\infty)e^{-\alpha t} \quad (36)$$

where, f_t = infiltration rate at time t, in inches/hour.

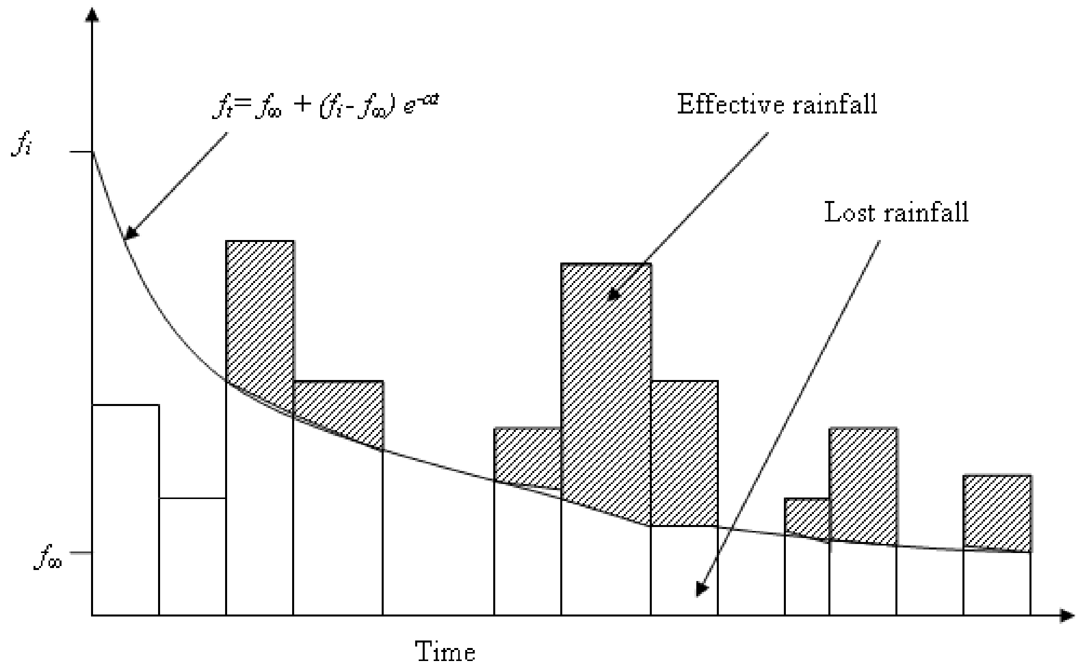
f_i = maximum or initial infiltration rate (at t = 0), in inches/hour. The default value is 3 inches/hour.

f_∞ = final infiltration rate (at t = ∞), in inches/hour. The default value is 0.5 inches/hour.

α = decay coefficient, in per second. The default value is 0.001/second.

t = time from beginning of storm, in seconds.

The actual infiltration at time *t* during the rainfall event(s) is the lesser of rainfall intensity and rate of infiltration, ft. The following figure describes Horton's infiltration model graphically. Only the shaded portion of the rainfall intensities, the region above the infiltration curve, becomes runoff. The remaining rainfall is lost in the form of infiltration.


$f_t = f_\infty + (f_i - f_\infty)\, e^{-\alpha t}$
Effective rainfall
Lost rainfall
f_i
f_∞
Time

INFILTRATION REGENERATION

Infiltration capacity can recover (regenerate) during dry periods. InfoSewer models infiltration regeneration whenever there is dry time steps (when there is no precipitation for the catchment) using the following drying curve. Modeling of infiltration regeneration enables the user to account for the effects of antecedent precipitation on rainfall abstractions, thus enhancing simulation accuracy.

$$f_t = f_i - (f_i - f_\infty)e^{-\alpha_d(t-t_w)} \quad (37)$$

where, f_t = infiltration rate at time t, in inches/hour.

f_i = maximum or initial infiltration rate (at t = 0), in inches/hour. The default value is 3 inches/hour.

f_∞ = final infiltration rate (at t = ∞), in inches/hour. The default value is 0.5 inches/hour.

α_d = decay coefficient for the recovery curve, in per second. It is generally considered to be less than
α, implying a longer drying curve than wetting curve.

t = time from beginning of storm, in seconds.

t_w = hypothetical projected time, internally computed by the model.

DEPRESSION/RETENTION STORAGE LOSS

In addition to infiltration losses, InfoSewerH2OMAP Sewer can also model losses due to depression/retention storage. The model allows the user to input separate depression storage loss values for the pervious and for the impervious areas of the catchment. The model internally derives representative depression storage loss value for the catchment by weighting the user input depression storage loss values by respective areas of the pervious and the impervious lands in the catchment.

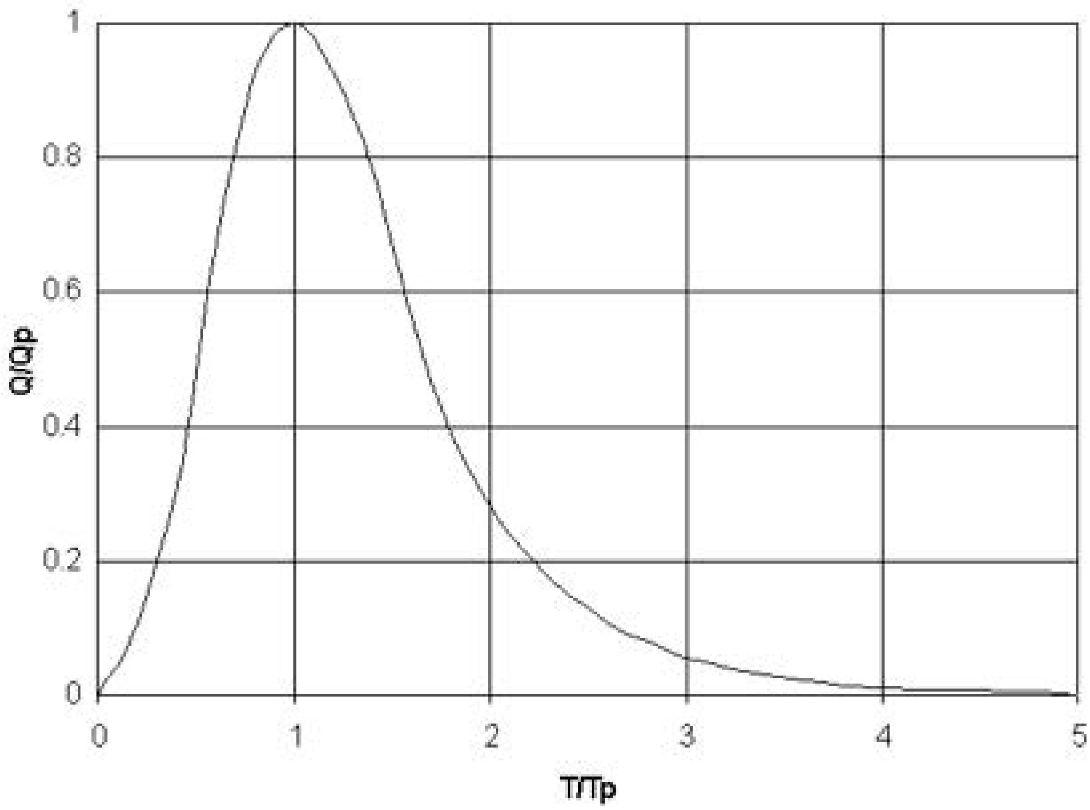
TYPES OF UNIT HYDROGRAPHS

Unit hydrographs could be natural or synthetic. Natural unit hydrographs are derived from observed data, whereas synthetic hydrographs are generated following empirical techniques based on watershed parameters and storm characteristics to simulate the natural unit hydrograph.

If a natural hydrograph is available for a watershed, InfoSewer can utilize it to generate storm runoff at different locations in the collection system. In the absence of a natural unit hydrograph, the model can derive a synthetic unit hydrograph using four different techniques. Each of the unit hydrograph synthesizing approaches is described below.

NRCS (SCS) DIMENSIONLESS UNIT HYDROGRAPH METHOD

The SCS dimensionless unit hydrograph, graphically descried below, is widely used in practice. To generate a tr-hour unit hydrograph for a watershed, time to peak (Tp) and the peak flow rate (Qp) are determined using watershed characteristics.



$$T_p = \frac{t_r}{2} + t_l \quad (38)$$

where t_r is duration of effective rainfall, and t_l is lag time of the watershed. Lag time represents the time from the center of mass of effective rainfall to the time to peak of a unit hydrograph. In other words, lag time is a delay in time, after a brief rain over a watershed, before the runoff reaches its peak. The lag time can either be specified by the user, or can be calculated by the model using the following SCS equation.

$$t_l = L^{0.8}\left(\frac{1000}{CN} - 9\right)^{0.7} \Big/ \left(19000 S^{0.5}\right) \quad (39)$$

where

t_l = lag time of the watershed in hours.

L = hydraulic length of the watershed in ft. This refers to travel distance of water from
the most upstream location of the watershed to the point where the unit hydrograph
is being derived.

CN = the SCS curve number. This is a measure of runoff generating capacity of a
watershed, and it depends on the soil, the antecedent moisture condition, the cover,
and the hydrologic conditions of the watershed. Recommended CN values for urban areas are
given below. The SCS suggests the CN values to be
within 50 and 95.

S = average slope of the watershed.

COVER DISCRIPTION		CURVE NUMBER FOR HYDROLOGIC SOIL GROUP			
Cover Type and Hydrologic Condition	Average Percent Impervious Area	A	B	C	D
Fully Developed Urban Areas (Vegetation Established)					
Open Space (lawns, parks, golf courses, cemetries, etc.):					
Poor Condition (grass cover < 50%)		68	79	86	89
Fair Condition (grass cover 50% to 75%)		49	69	79	84
Good condition (grass cover > 75%)		39	61	74	80
Impervious areas:					
Paved parking lots, roofs, driveways, etc. (excluding right-of-way)		98	98	98	98
Streets and roads					
Paved; curbs, and storm sewers (excluding right-of-way)		98	98	98	98
Paved; open ditches (including right-of-way)		83	89	92	93
Gravel (including right-of-way)		76	85	89	91
Dirt (including right-of-way)		72	82	87	89
Western desert urban areas:					
Natural desert landscaping (pervious areas only)		63	77	85	88
Artificial desert landscaping (impervious weed barrier, desert shrub with I- to 2-inch sand or gravel mulch and basin borders)		96	96	96	96
Urban districts:					
Commercial and business	85	89	92	94	95
Industrial	72	81	88	91	93
Residential districts by average lot size:					
1/8 acre or less (town house)	65	77	85	90	92
1/4 acre	38	61	75	83	87
1/3 acre	30	57	72	81	86
1/2 acre	25	54	70	80	85
1 acre	20	51	68	79	84
2 acres	12	46	65	77	82
Developing urban areas:					
Newly graded areas (pervious areas only, no vegetation)		77	86	91	94

The peak flow rate is calculated as:

$$Q_p = \frac{484A}{T_p} \quad (40)$$

where

Q_p = peak flow rate in cfs.

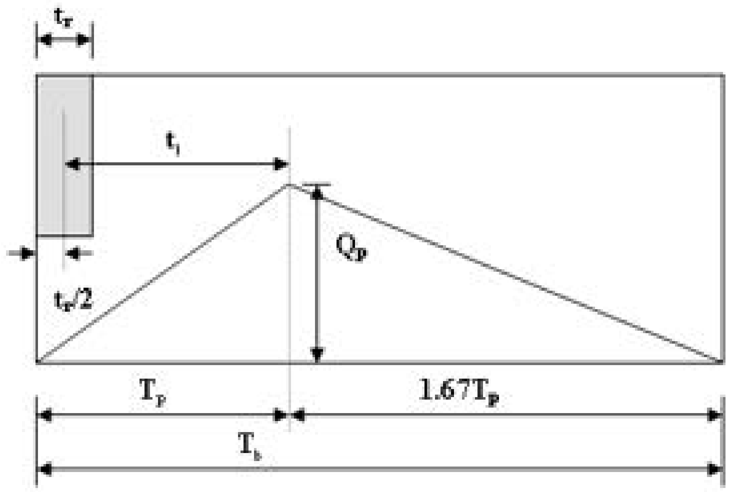
A = area of the watershed, in square miles, draining to the location of the unit hydrograph.

T_p = time to peak of the unit hydrograph in hours.

Once T_p and Q_p are known, actual time and flow rate ordinates of the t_r-hour unit hydrograph are determined by multiplying the dimensionless time (T/T_p) and the dimensionless flow rate ordinates (Q/Q_p) by T_p and Q_p , respectively.

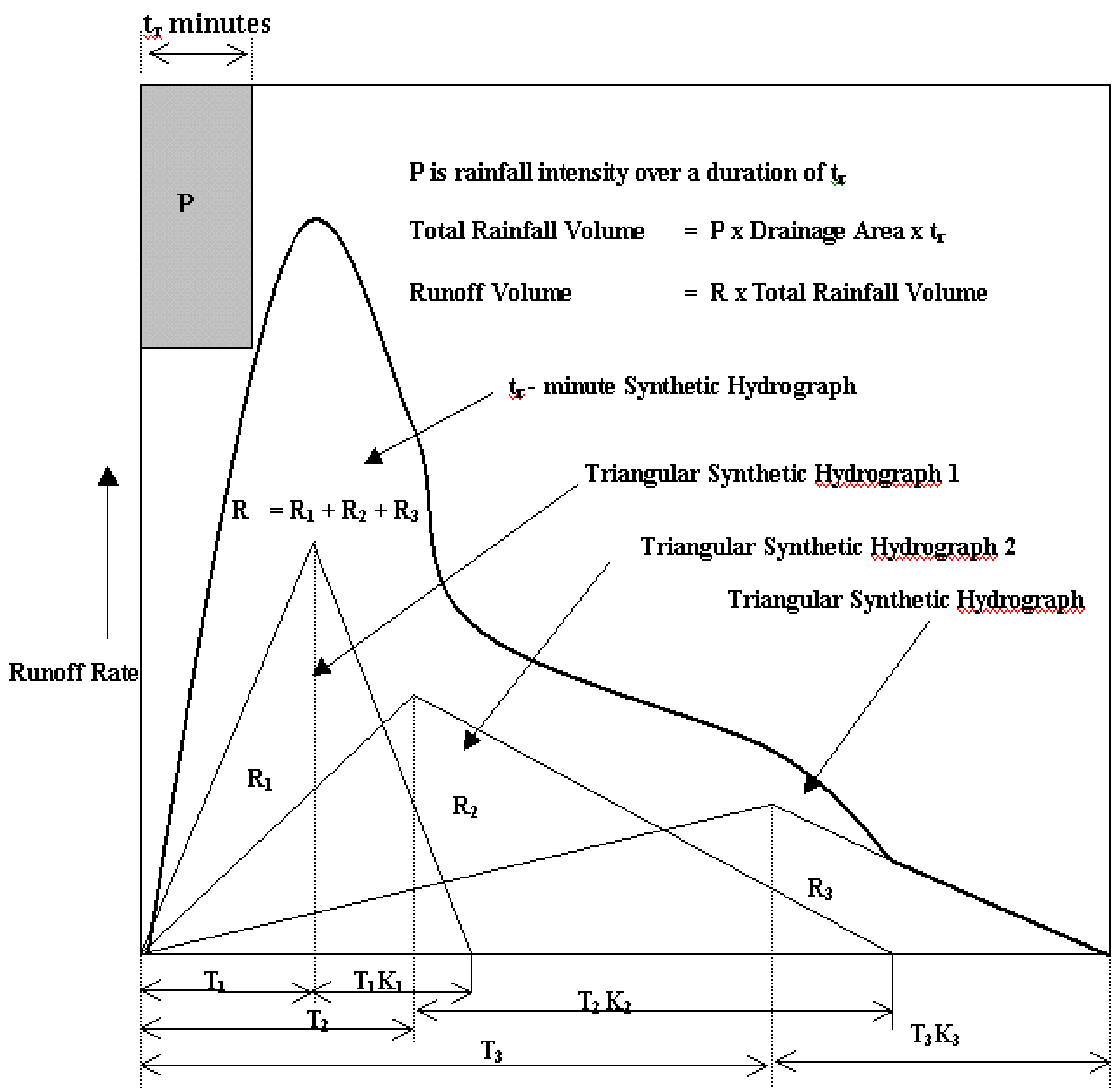
NRCS (SCS) TRIANGULAR UNIT HYDROGRAPH METHOD

The SCS has also developed a triangular unit hydrograph (see figure below) that is an approximation to the dimensionless unit hydrograph described above. The triangular unit hydrograph is entirely defined in terms of three points, Q_p, T_p, and T_b. The lag time, time to peak, and peak flow rate are calculated using the same equations as for the dimensionless unit hydrograph.



THE TRI-TRAINGLE METHOD

Tri-triangle is one of the techniques used by InfoSewer for synthesizing a hydrograph. The triangular unit hydrograph technique developed by the U.S. Soil Conservation Service (now known as Natural Resources Conservation Service NRCS) is one of the commonly used methods. As described above NRCS's triangular synthetic unit method uses a single triangle to represent a unit hydrograph. However, shape of unit hydrograph is too complex to be well captured by a single triangle. InfoSewer applies up to three triangular synthetic hydrographs, as the name implies, to simulate a hydrograph. The total synthetic hydrograph is the result of aggregating corresponding ordinates of the three triangular hydrographs. Each of these three triangular hydrographs has its own characteristic parameters, namely time to peak, recession constant, and fraction of an effective rainfall volume allocated to the triangle. The technique is graphically illustrated below.



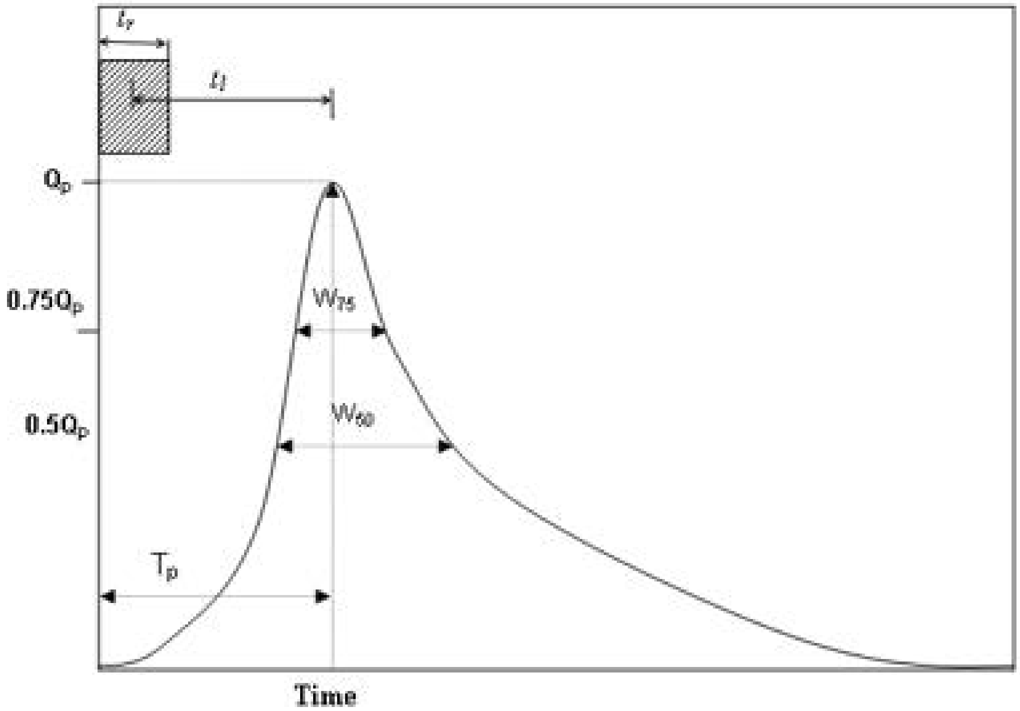
R_1, R_2, and R_3 are fractions of excess rainfall volume, R, allocated to triangular synthetic hydrographs 1, 2, and 3 respectively. T_i and K_i are time to peak and recession constants of the triangles, respectively. The three triangular synthetic hydrographs are conceptual representations of different components of direct runoff or rainfall-dependent infiltration/inflow. The first triangle represents rapidly responding (fast) components, such as contributions from pavements and rooftops, or direct inflow or rapid infiltration into separate sewer systems. The third triangle represents slow runoff components such as ground water contributions or slow infiltration into separate sewers. The second triangle represents runoff or infiltration with a medium time response.

Time to peak value of the first triangle typically varies between 1 and 2 hours, depending on the size of the tributary area in question. The second triangle takes

T values ranging from 4 to 8 hours. The third triangle parameter varies greatly depending on the infiltration characteristics of the system being modeled, and has a T value generally between 10 and 24 hours. The default values used for T_1, T_2, and T_3 are 1, 4, and 12 hours respectively. Values of K for the first triangle typically ranges between 2 and 3. The second and the third triangles take K values from 2 to 4. The default values of K are 2, 3, and 3 for the first, the second, and the third triangles, respectively.

THE COLORADO URBAN HYDROGRAPH PROCEDURE

The Colorado Urban Hydrograph Procedure (CUHP) uses the equations and procedures presented in the Urban Drainage Criteria Manual (USDCM) of the Urban Drainage and Flood Control District (UDFCD). Shape of the CUHP synthetic unit hydrograph is determined using the following equations that relate unit hydrograph parameters to catchment properties.



Lag time (t_l) of the watershed (catchment), defined as the time from the center of unit storm duration to the peak of the unit hydrograph, is determined as:

$$t_l = C_t \left(\frac{(L \cdot L_{Ca})}{\sqrt{S}} \right)^{0.48} \quad (41)$$

where t_l = lag time in hours.

L = length along the drainageway path from study point to the most upstream limits of the catchment in
miles.

L_{ca} = length along stream from study point to a point along stream adjacent to the centroid of the
catchment in miles.

S = length weighted average slope of catchment along draiangeway path to upstream limits of the
catchment.

C_t = time to peak coefficient.

Once the lag time is known, time to peak (T_p) of the unit hydrograph could be determined by adding $0.5t_r$ to the lag time in consistent units.

Peak flow rate, Q_p, of the unit hydrograph is calculated as:

$$Q_p = \frac{640 C_p A}{T_p} \quad (42)$$

where Q_p = peak flow rate of the unit hydrograph, in cfs.

A = area of the catchment, in square miles.

C_p = unit hydrograph peaking coefficient, and is determined as:

$$C_p = P \cdot C_t \cdot A^{0.15} \quad (43)$$

where P = peaking parameter.

C_t and P are defined in terms of percent impervious (I_a) of the catchment as:

$$C_t = aI_a^2 + bI_a + c \quad (44)$$

$$P = dI_a^2 + eI_a + f \quad (45)$$

The coefficients *a*, *b*, *c*, *d*, *e*, and *f* are defined in terms of I_a in the following table.

Ia	***a***	***b***	***c***	***d***	***e***	***f***
***Ia* ≤ 10**	0.0	-0.00371	0.163	0.00245	-0.012	2.16
***10* < *Ia* ≤ 40**	0.000023	-0.00224	0.146	0.00245	-0.012	2.16
***Ia* > 40**	0.0000033	-0.000801	0.120	-0.00091	0.228	-2.06

The widths of the unit hydrograph at 50% and 75% of the peak are estimated as:

$$W_{50} = \frac{500}{\left(\frac{Q_p}{A}\right)} \quad (46)$$

$$W_{75} = \frac{260}{\left(\frac{Q_p}{A}\right)} \quad (47)$$

where W_{50} = width of the unit hydrograph at 50% of the peak, in hours.

W_{75} = width of the unit hydrograph at 75% of the peak, in hours.

Q_p = peak flow rate, in cfs.

A = catchment area, in square miles.

It is recommended that a unit hydrograph duration of 5-minute be used for studies that apply the CUHP. The maximum recommended drainage area (catchment size) for any single CUHP unit hydrograph is 5 square miles. Whenever a larger watershed is studied, it needs to be subdivided into subcatchments of 5-square miles or less. For this synthetic unit hydrograph method, the minimum drainage area should be 90 acres. For catchments smaller than 90 acres, other unit hydrograph generation mechanisms should be used.

GENERATION OF RUNOFF HYDROGRAPH

InfoSewer generates a direct runoff hydrograph resulting from single or multiple storm events using a unit hydrograph synthesized according to any of the three techniques described above, or from a user-specified natural unit hydrograph. The model relies on a user input hyetograph (i.e., rainfall intensity versus time graph). The model can derive effective rainfall from user supplied total rainfall hyetographs.

An entire watershed of a collection system may not be well represented by a single unit hydrograph owing to variability in topography, land use, and soil characteristics of the subwatersheds. In the case of separate sewer systems, the magnitude and type of sources of infiltration/inflow will vary by subbasin. Accordingly, InfoSewerH2OMAP Sewer allows usage of multiple unit hydrographs, each representing part of the watershed being modeled. The unit hydrographs could be of any duration (e.g., 15-minute, 1-hr), but the duration must be the same for all the unit hydrographs involved in modeling of the watershed.

To derive the storm hydrographs, InfoSewerH2OMAP Sewer applies the basic assumptions of unit hydrograph theory described above (i.e., linearity, time invariance, and the principle of superposition). Storm events are assumed to have constant intensity over the duration of the unit hydrograph. Excess rainfall resulting from single or multiple storm events is discretized at intervals of unit hydrograph duration. For example, if a sewer collection system is being modeled using a 15-minute unit hydrograph, and if duration of the excess rainfall under investigation is 1-hour, the rainfall duration will be divided into four 15-minute rainfall events of constant intensity. This discretization approach, along with the unit hydrograph assumptions of linearity, time invariance, and superposition, enables InfoSewer to simulate storm runoff hydrograph at any location (e.g., loading manhole) throughout the collection system for any number of storm events. During generation of runoff hydrographs for locations other than the site where the unit hydrograph is originally derived, ordinates of the unit hydrographs are adjusted according to the ratio of drainage area of the two locations.

Once the storm hydrographs for every loading manhole in the collection system are known, the storm load will be added to other loading types such as sanitary loads, if any, and will be routed through the collection system using the powerful Muskingum-Cunge’s dynamic flow routing algorithm.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

Controls

InfoSewer provides comprehensive operational control schemes to accurately simulate the dynamic hydraulic behavior of a sanitary sewer collection system. The model offers five different pump control mechanisms. During an EPS, controls describe the on-off status of selected pumps. The switching mode for the pump status is activated when a control statement is satisfied based on the type of control. An example of an operational control is given below for wet-well level based controls:

> IF (level in wet-well WW1 drops below 3 feet) THEN (turn OFF pump P1)
>
> IF (level in wet-well WW1 goes above 13 feet) THEN (turn ON pump P1)

The default “ON Setting” value is the top wastewater level/volume of the wet-well while the default “OFF Setting” is the bottom wastewater level/volume of the wet-well. The default pump speed setting is one (pump speed ratio of 1). The initial pump status is overwritten by the operational controls during an extended period simulation.

InfoSewer™

Modeling Made Easy

Available Graph Types

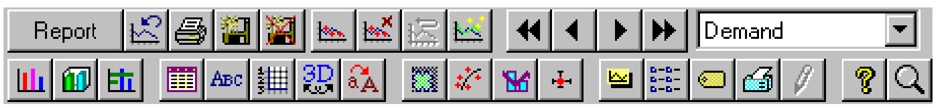
The following graph types are available from the Output Report Manager. Choose a graph for more information:

Steady State Analysis	*Steady State Design*	*EPS*
Pipe Profile Graph	**(N/A)**	**Loading Manhole Graph**
Reserve Capacity Profile Graph		**Chamber Manhole Graph**
		Outlet Graph
		Wet Well Graph
		Gravity Main Graph
		Force Main Graph
		Pump Graph
		Group Loading Manhole Graph
		Group Chamber Manhole Graph
		Group Outlet Graph
		Group Wet Well Graph
		Group Gravity Main Graph

		Group Force Main Graph
		Group Pump Graph
		EPS Pipe Profile Graph
		System Load

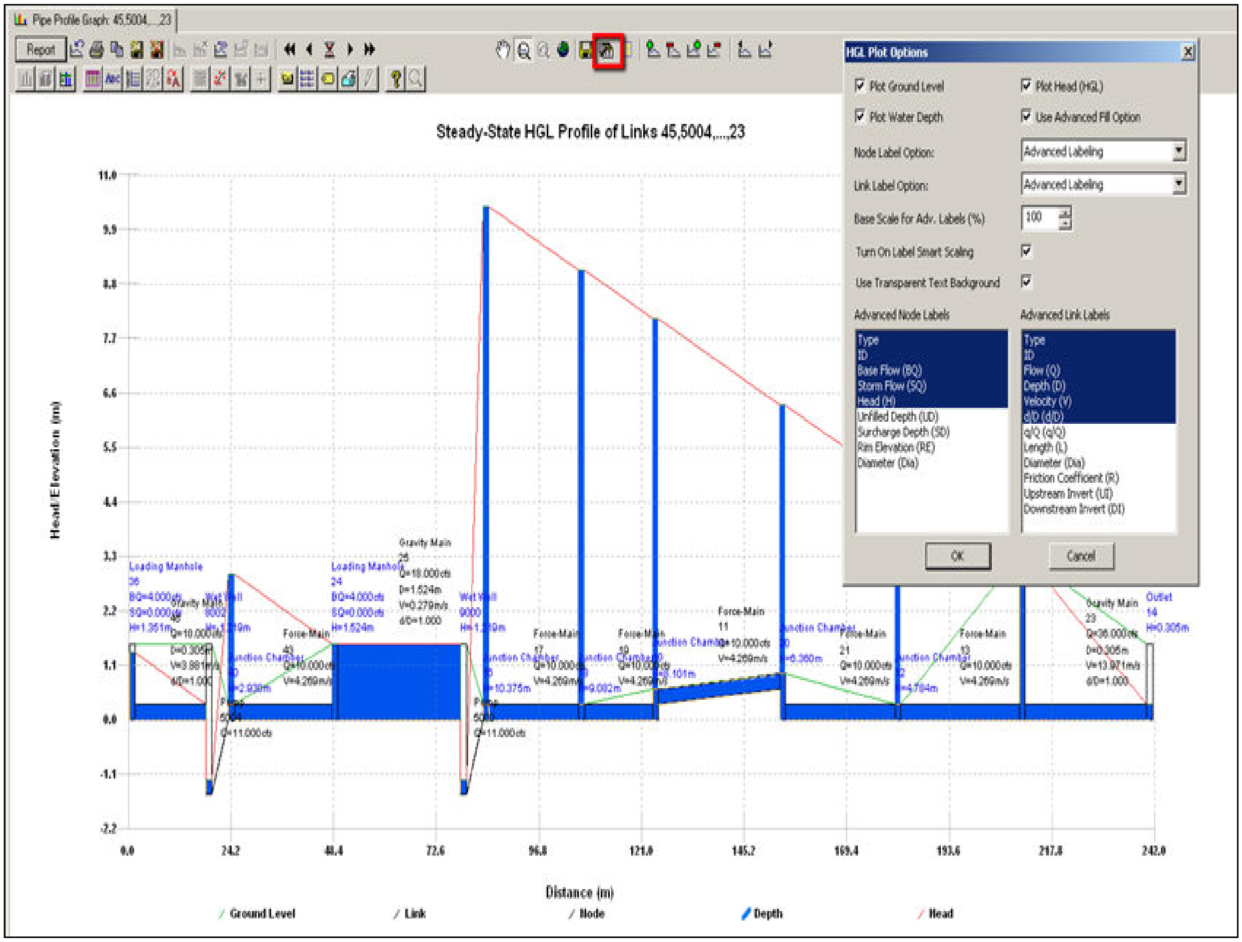
Other graphs that is included in InfoSewer is a **Frequency Graph** (Click here to learn more) and the **Data Histogram** (Click here to learn more).

Click on the Graph Modification buttons below to learn more about each icon.



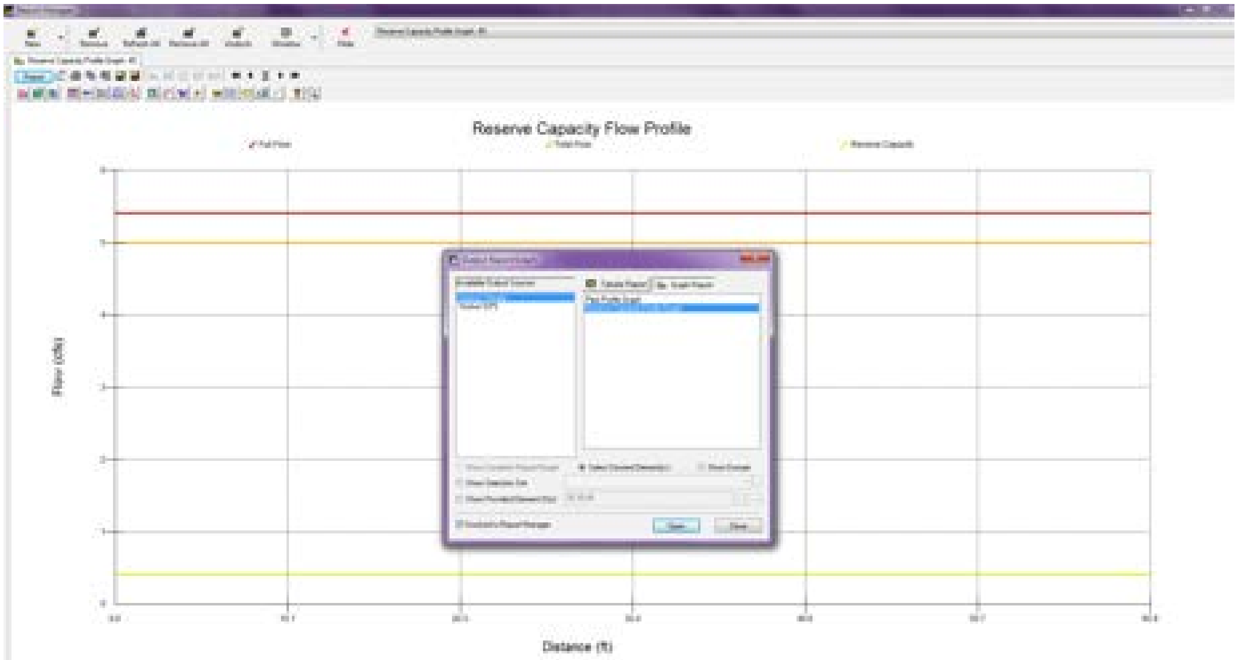
Steady State Pipe Profile Graph

Displays a pipe profile for one or more pipes during a steady state simulation. The graph X-axis displays the linear distance for the selected pipe(s) while the Y-axis displays the Elevation above datum for the pipe profile. The green lines represent the top and bottom of the pipe while the blue line represents the Actual d/D for the pipe during the steady state simulation. Upon choosing this graph type you are prompted to choose a pipe. Upon selecting a pipe, the graph appears in the Flow Profile window. Pipe graphs can display pipe flow, velocity, d/D ratio, depth, q/Q, length, diameter, friction coefficient, upstream and downstream invert if you use the Advanced Labeling option. Node plotted data now includes Base flow, Storm Flow, Head, Unfilled Depth, Surcharged Depth, Rim Elevation and Diameter.



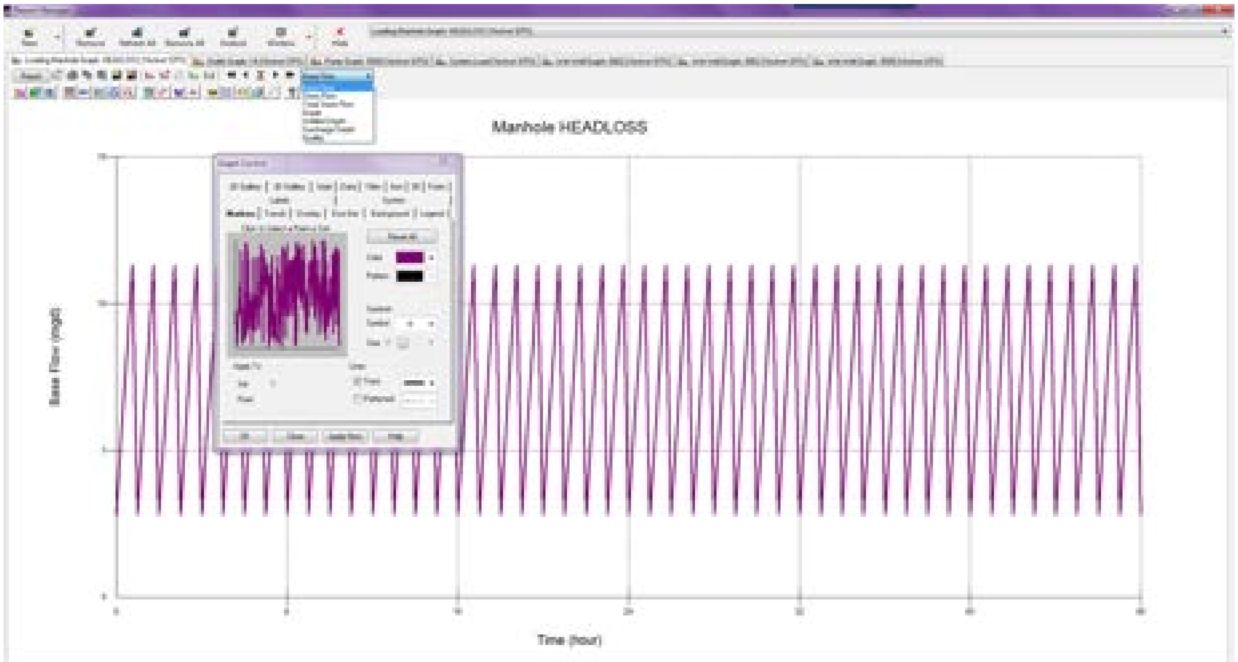
Steady State Reserve Capacity Pipe Profile Graph

Displays a graph of the full flow, total flow and reserve capacity for one or more pipes during a steady state simulation. The graph X-axis displays the linear distance for the selected pipe(s) while the Y-axis displays the three flows for all of the selected pipes. The green line represents that Reserve Capacity, which is the pipe full flow minus the total flow in the pipe. The pipe full flow in red is the full depth flow based on the pipe cross sectional area, pipe slope, pipe hydraulic radius and pipe friction loss coefficient. The total flow in yellow is the sum of all loads and storm flows.



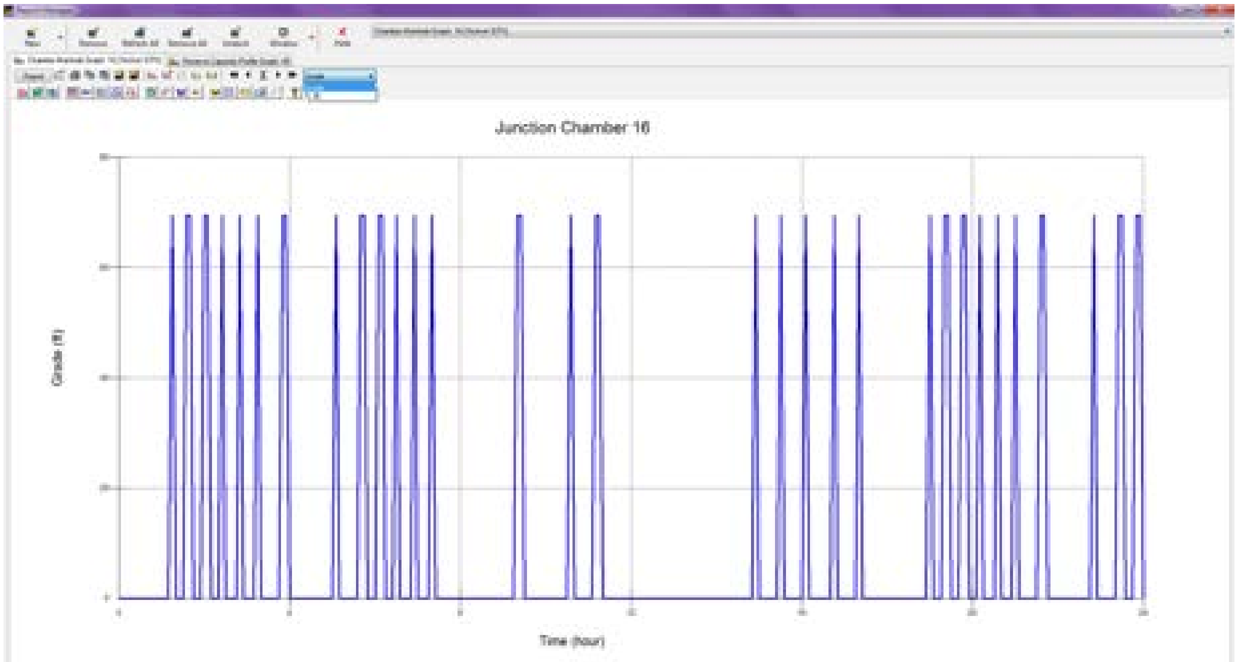
Loading Manhole Graph

Displays simulation results for a manhole. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected manhole. Once selected, the graph appears in the Output Report Manager window. Manhole graphs can display loading, Storm Flow, Total Storm Flow, Grade, Unfilled Depth, Surcharge Depth and Water Quality (if simulated).



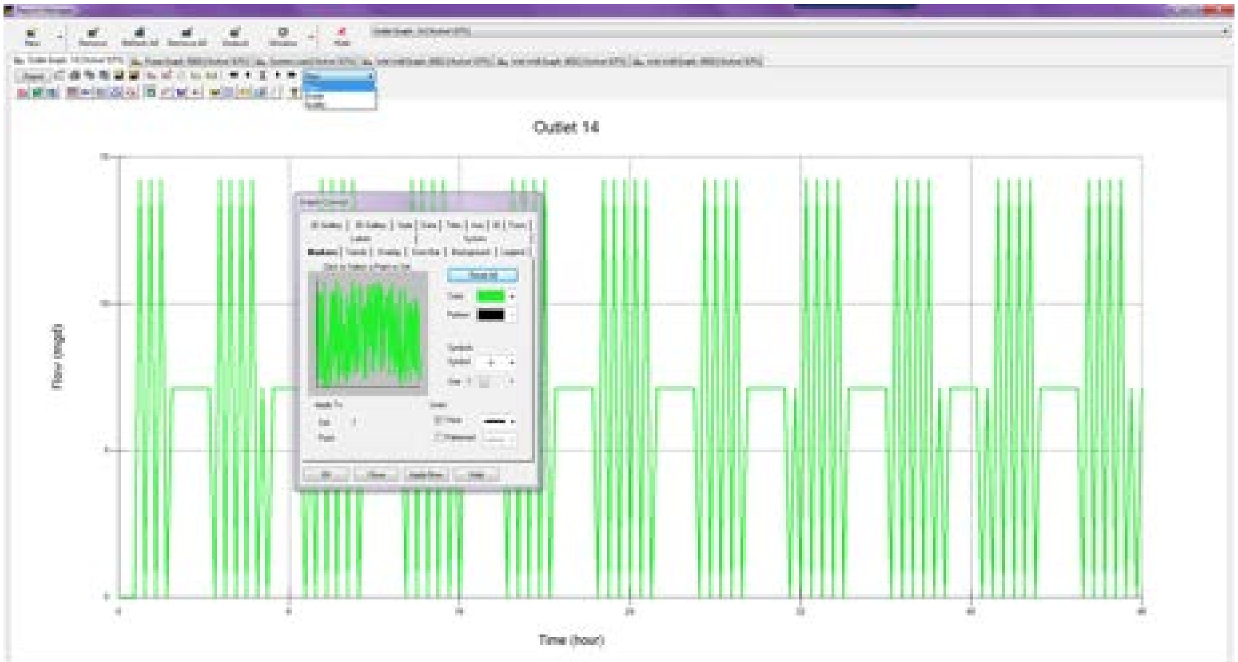
Chamber Manhole Graph

Displays simulation results for a chamber manhole. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected manhole. Upon choosing this graph type, you are prompted to choose a chamber manhole. Once selected, the graph appears in the Output Report Manager window. Chamber manhole graphs can only display the hydraulic grade and water quality concentration passing through the junction.



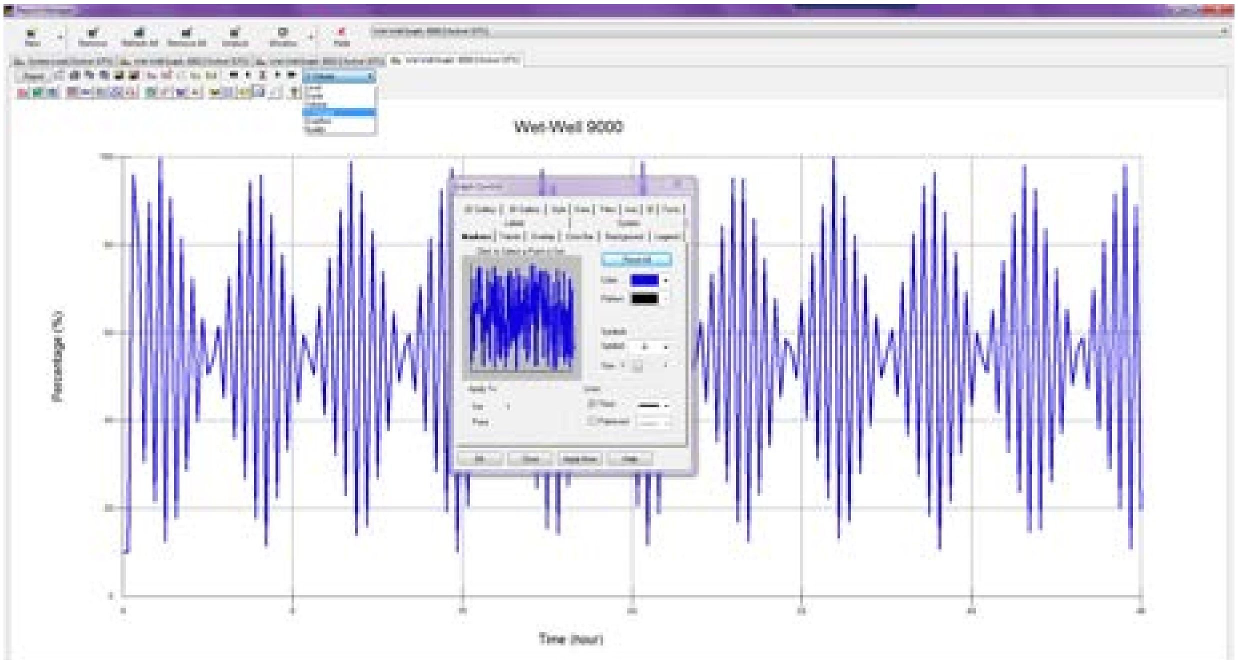
Outlet Graph

Displays simulation results for an outlet. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the outlet. Upon choosing this graph type, you are prompted to choose an an outlet. Once selected, the graph appears in the Output Report Manager window. Outlet graphs can either display the hydraulic grade, the flow passing through the outlet or the water quality concentration.



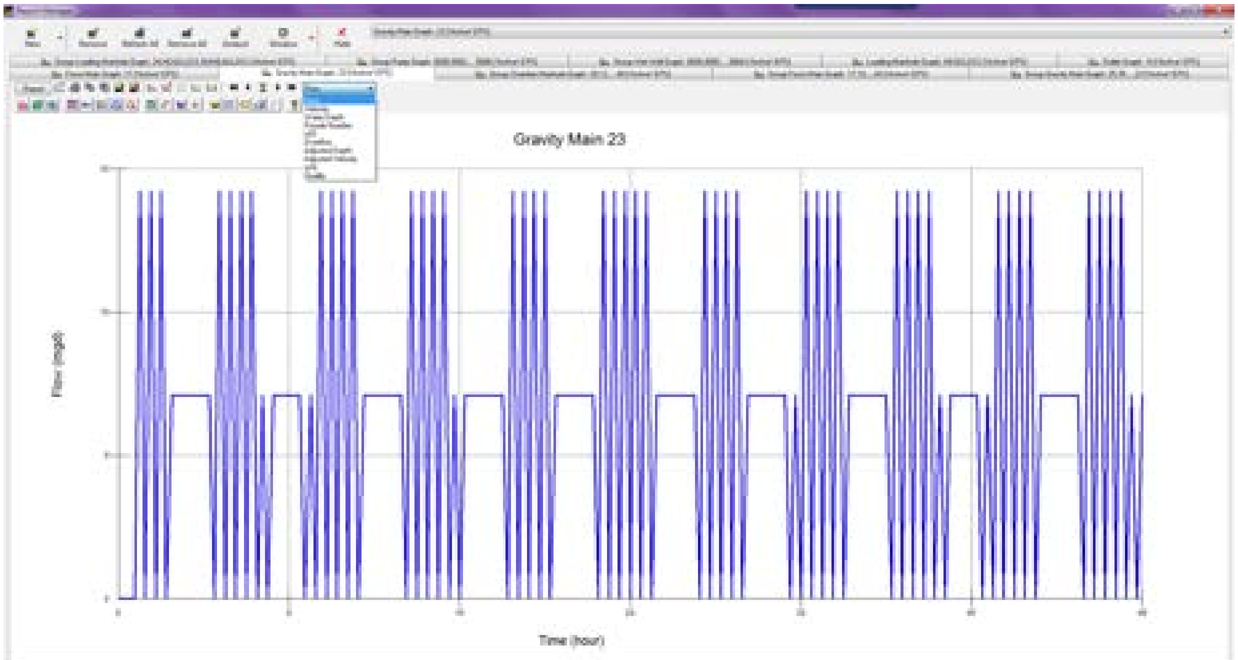
Wet Well Graph

Displays simulation results for one wet well. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected wet well. Upon choosing this graph type, you are prompted to choose a wet well. Once selected, the graph appears in the Output Report Manager window. Wet well graphs can display the hydraulic grade, water level in the wet well, volume stored, % volume stored, the overflow condition of the wet well and the water quality concentration (if simulated).



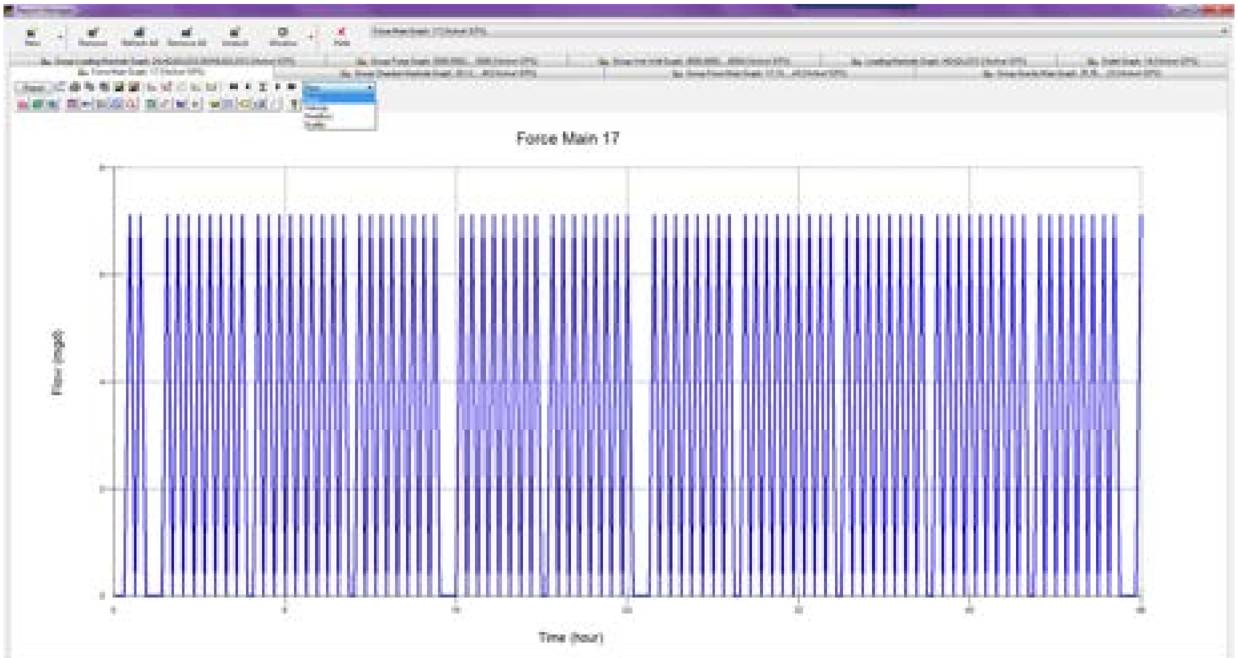
Gravity Main Graph

Displays simulation results for a gravity main. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the gravity main. Upon choosing this graph type, you are prompted to choose an gravity main pipe. Once selected, the graph appears in the Output Report Manager window. Gravity main graphs can display the flow, velocity, water depth, Froude Number, d/D ratio for the selected pipe, Overflow, Adjusted Depth, Adjusted Velocity, q/Q ratio for the selected pipe and the water quality concentration (if simulated).



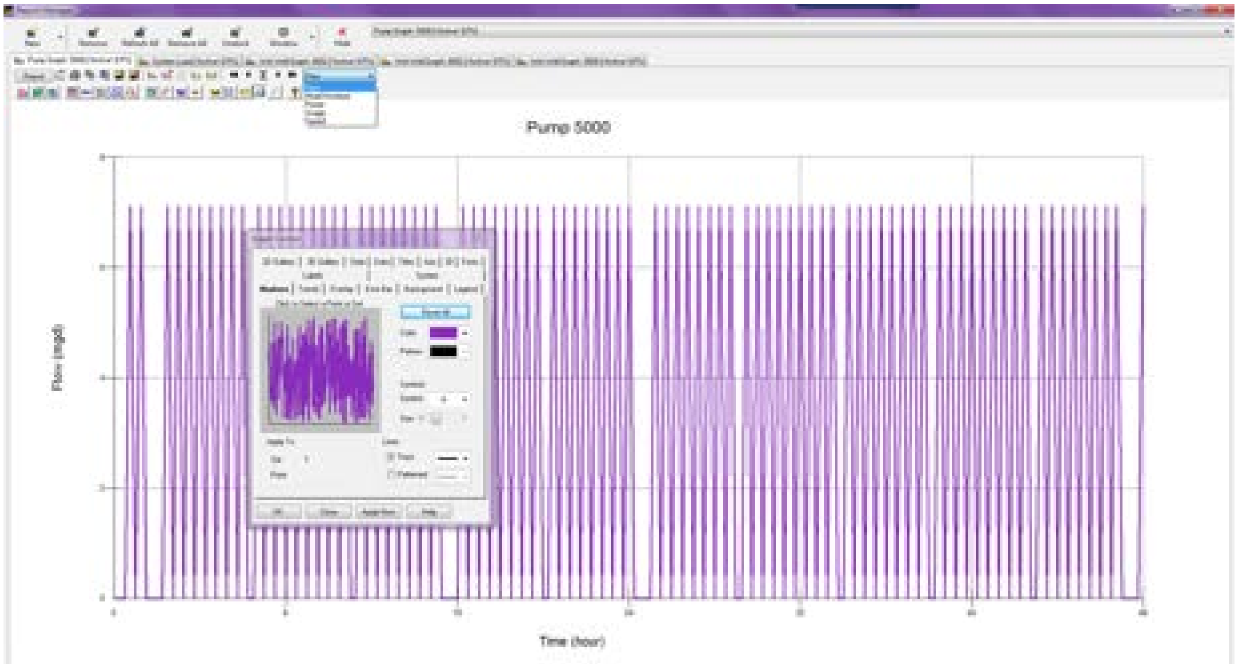
Force Main Graph

Displays simulation results for a force main. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the force main. Upon choosing this graph type, you are prompted to choose a force main. Once selected, the graph appears in the Output Report Manager window. Gravity main graphs can display the flow, velocity, headloss for the force main and the water quality concentration (if simulated).



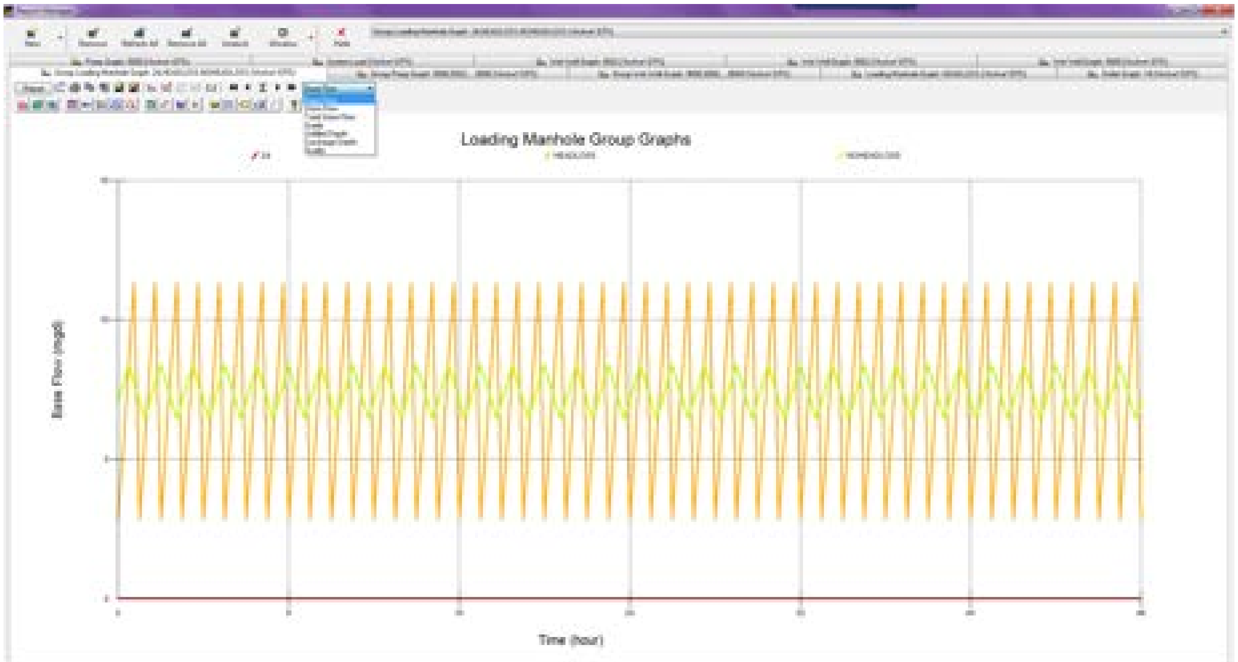
Pump Graph

Displays simulation results for a selected pump. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected pump. Upon choosing this graph type, you are prompted to choose a pump. Upon selection, the graph appears in the Output Report Manager window. Pump graphs can either display flow, head increase, power, usage and speed.



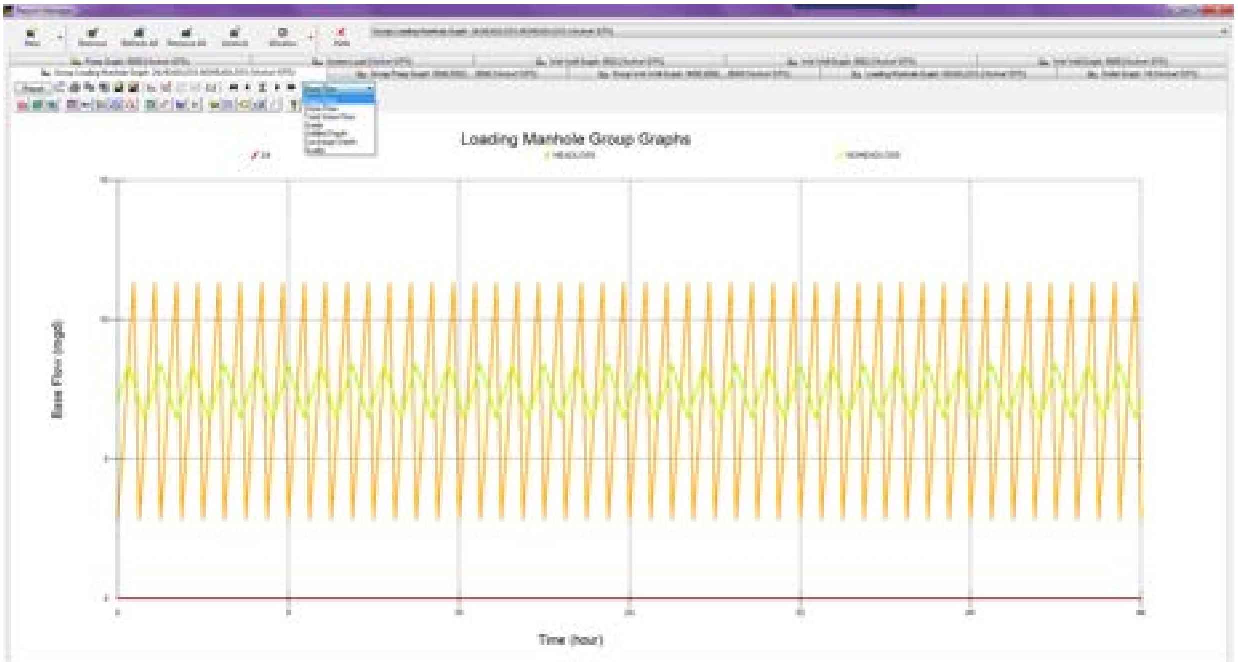
Group Loading Manhole Graph

Displays model results for two or more manholes. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected manholes. Upon choosing this graph type, you are prompted to choose two or more manholes. To do this, hold the <Shift> key down while selecting the manholes. Press the Enter key or the right mouse button to terminate the selection process. Once selected, the graph appears in the Output Report Manager window. Manhole group graphs can display load, overload and grade.



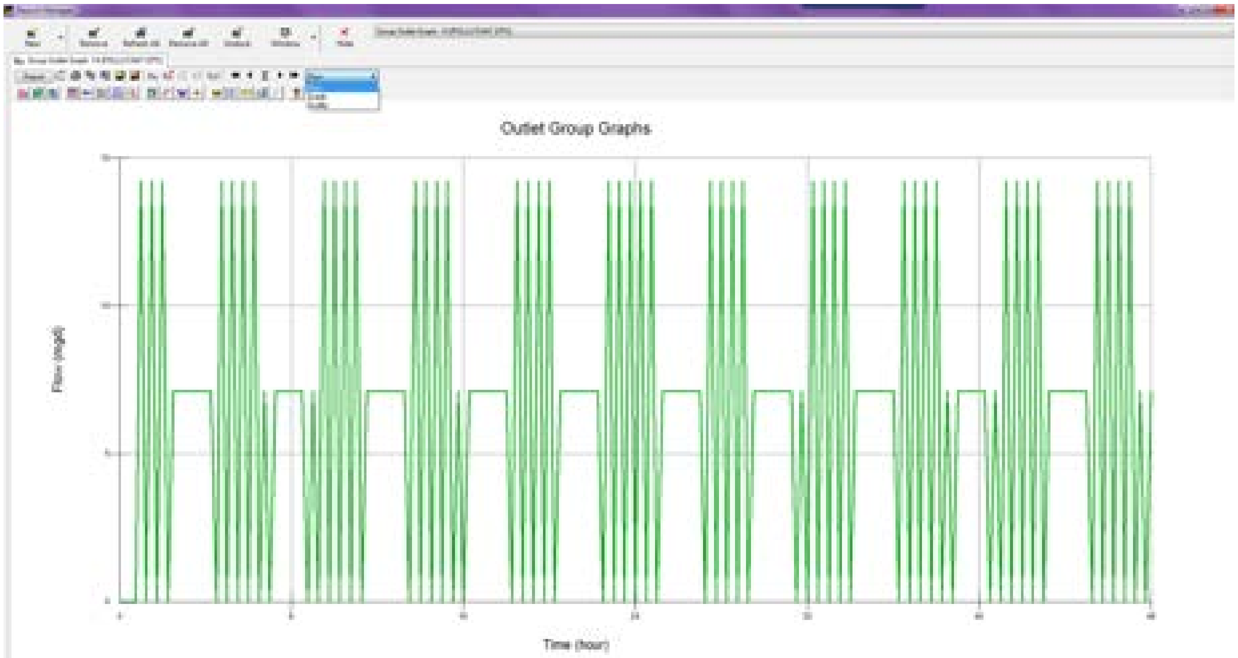
Group Chamber Manhole Graph

Displays model results for two or more chamber manholes. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected chamber manholes. Upon choosing this graph type, you are prompted to choose two or more chamber manholes. To do this, hold the <Shift> key down while selecting the manholes. Press the Enter key or the right mouse button to terminate the selection process. Once selected, the graph appears in the Output Report Manager window. Chamber manhole group graphs can only display hydraulic head and the water quality concentration (if simulated).



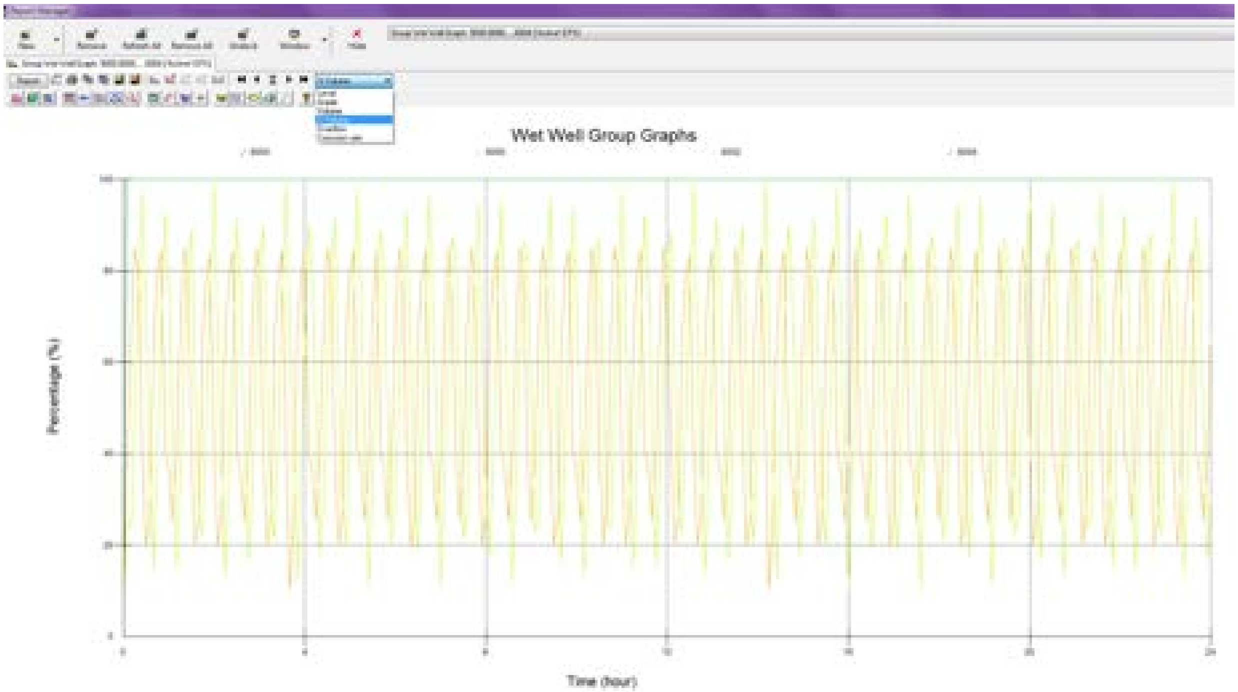
Group Outlet Graph

Displays model results for two or more outlets. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected outlets. Upon choosing this graph type, you are prompted to choose two or more outlets. To do this, hold the <Shift> key down while selecting the manholes. Press the Enter key or the right mouse button to terminate the selection process. Once selected, the graph appears in the Output Report Manager window. Outlet group graphs can display either flow, grade and the water quality concentration (if simulated).



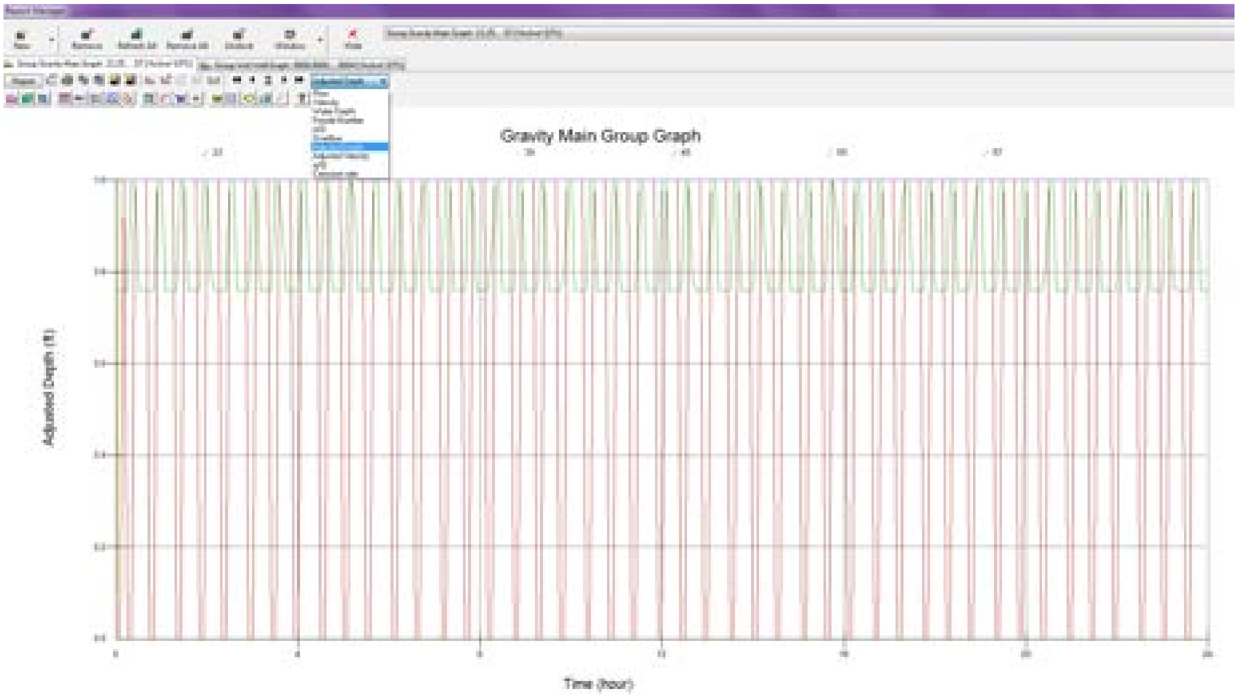
Group Wet Well Graph

Displays model results for two or more wet wells. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected wet wells. Upon choosing this graph type, you are prompted to choose two or more wet wells. To do this, hold the <Shift> key down while selecting the wet wells. Press the Enter key or the right mouse button to terminate the selection process. Once selected, the graph appears in the Output Report Manager window. Wet well group graphs can display the hydraulic grade, water level in the wet wells, volume stored, % volume stored and the overflow condition.



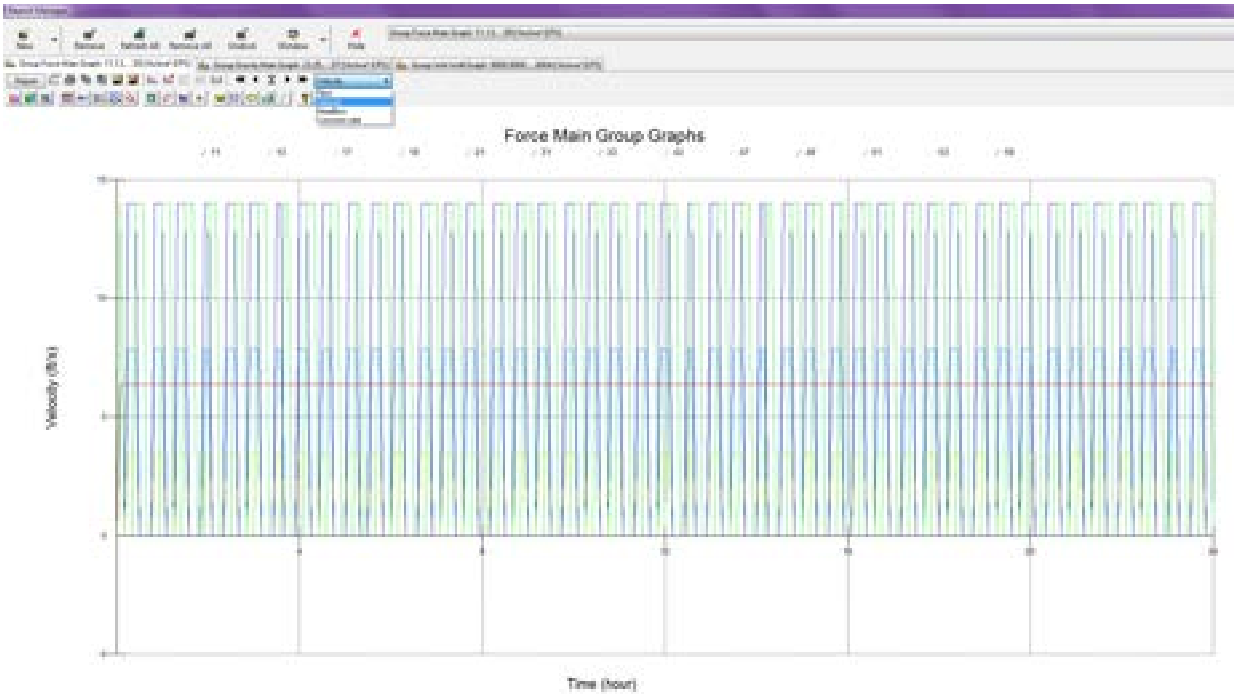
Group Gravity Main Graph

Displays model results for two or more gravity mains. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected gravity mains. Upon choosing this graph type, you are prompted to choose two or more pipes. To do this, hold the <Shift> key down while selecting the gravity mains. Press the Enter key or the right mouse button to terminate the selection process. Once selected, the graph appears in the Output Report Manager window. Gravity main group graphs appears in the Output Report Manager window. Group Gravity main graphs can display the flow, velocity, water depth, Froude Number, d/D ratio for the selected pipe, Overflow, Adjusted Depth, Adjusted Velocity, q/Q ratio for the selected pipe and the water quality concentration (if simulated).



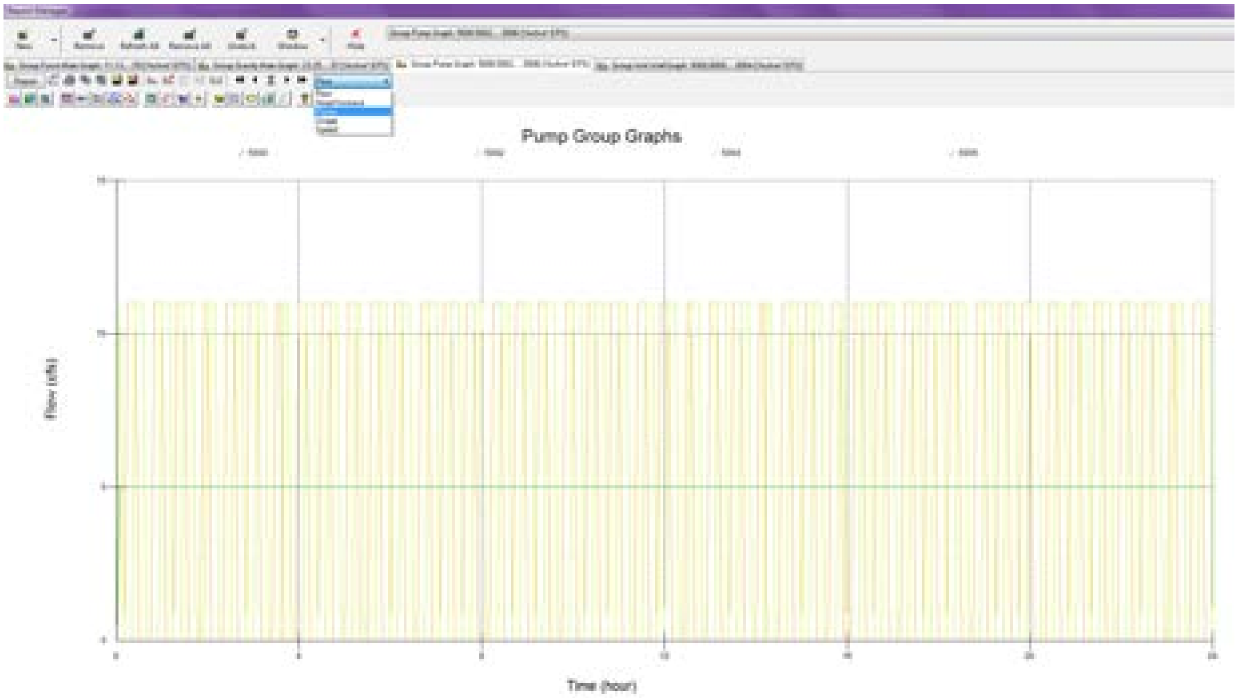
Group Force Main Graph

Displays model results for two or more force mains. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected force mains. Upon choosing this graph type, you are prompted to choose two or more force mains. To do this, hold the <Shift> key down while selecting the force mains. Press the Enter key or the right mouse button to terminate the selection process. Once selected, the graph appears in the Output Report Manager window. Force main group graphs can display the flow, velocity, headloss and water quality concentration (if simulated).



Group Pump Graph

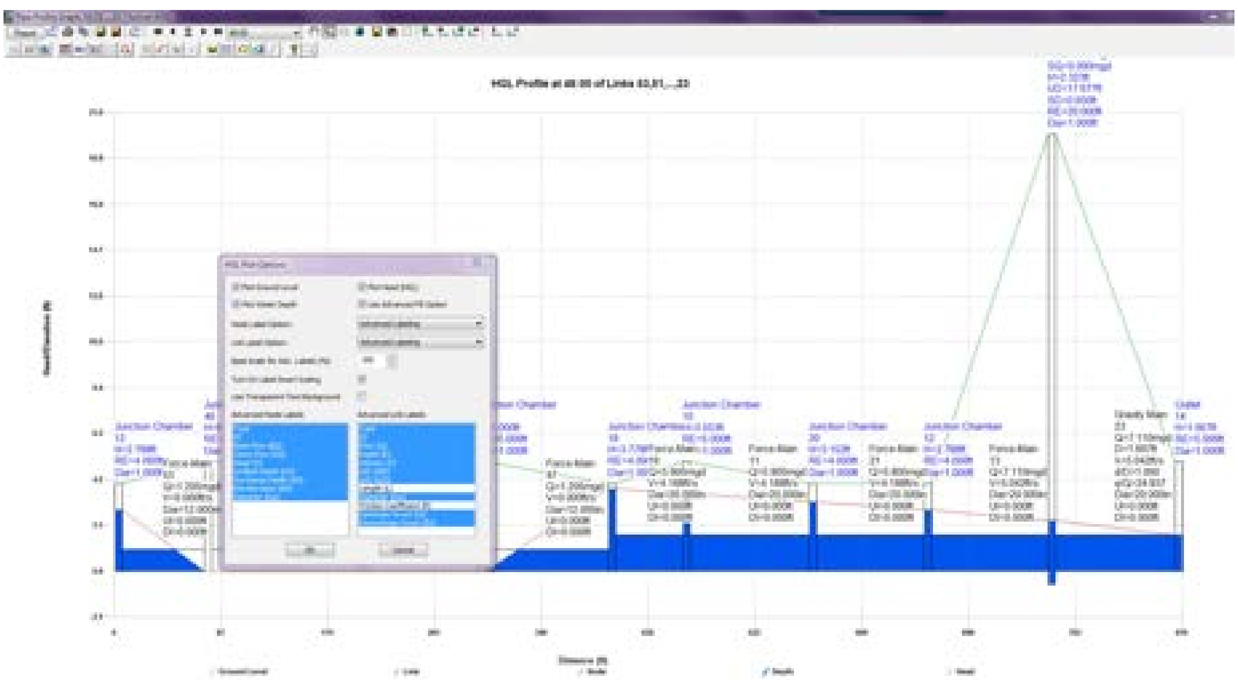
Displays model inputs for two or more pumps. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected pumps. Upon choosing this graph type, you are prompted to choose two or more pumps. To do this, hold the <Shift> key down while selecting the pumps. Press the Enter key or the right mouse button to terminate the selection process. Once selected, the graph appears in the Output Report Manager window. Pump group graphs can either display the flow, head increase, power, usage or speed of the pump.



Pump Group Graphs
Flow (cfs)
Time (hour)

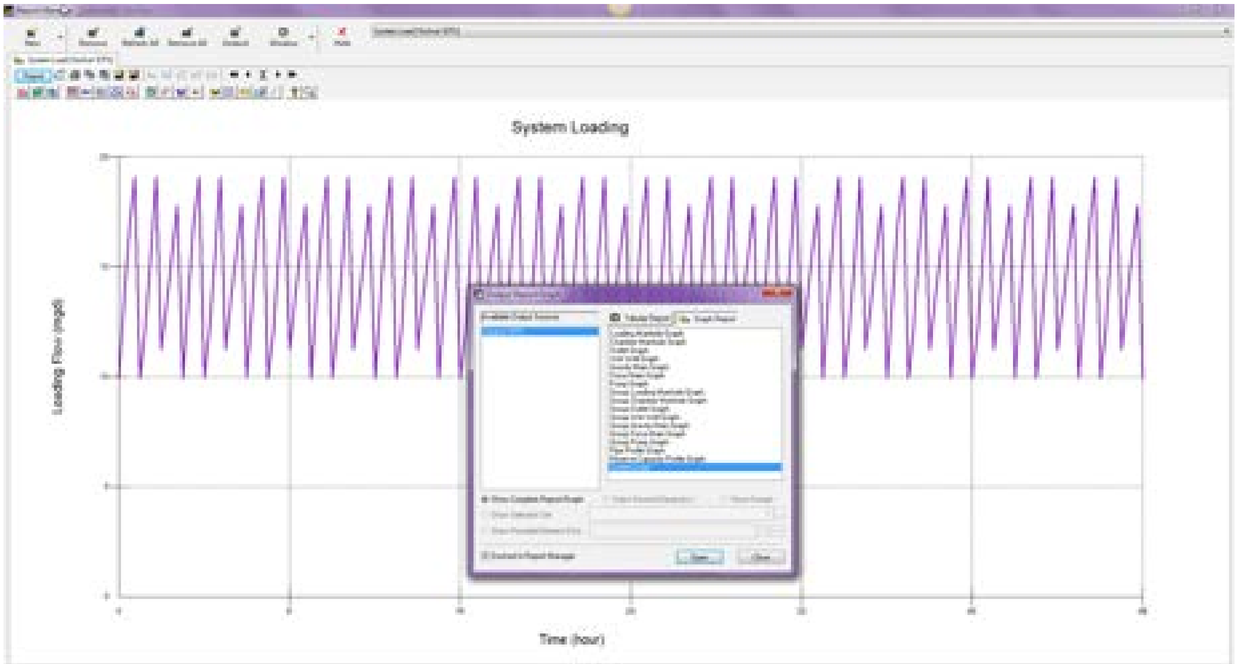
EPS Pipe Profile Graph

Displays a pipe profile for one or more pipes during an extended period simulation. The graph X-axis displays the linear distance for the selected pipe(s) while the Y-axis displays the Elevation above datum for the pipe profile. The green lines represent the top and bottom of the pipe while the blue line represents the Actual d/D for the pipe during the steady state simulation. Upon choosing this graph type you are prompted to choose a pipe. Upon selecting a pipe, the graph appears in the Flow Profile window. Pipe graphs can display pipe flow, velocity, d/D ratio, depth, q/Q, length, diameter, friction coefficient, upstream and downstream invert if you use the Advanced Labeling option. Node plotted data now includes Base flow, Storm Flow, Head, Unfilled Depth, Surcharged Depth, Rim Elevation and Diameter. The user can fast forward or step through the HGL plot at the saved time step using the time control at the top of the dialog.



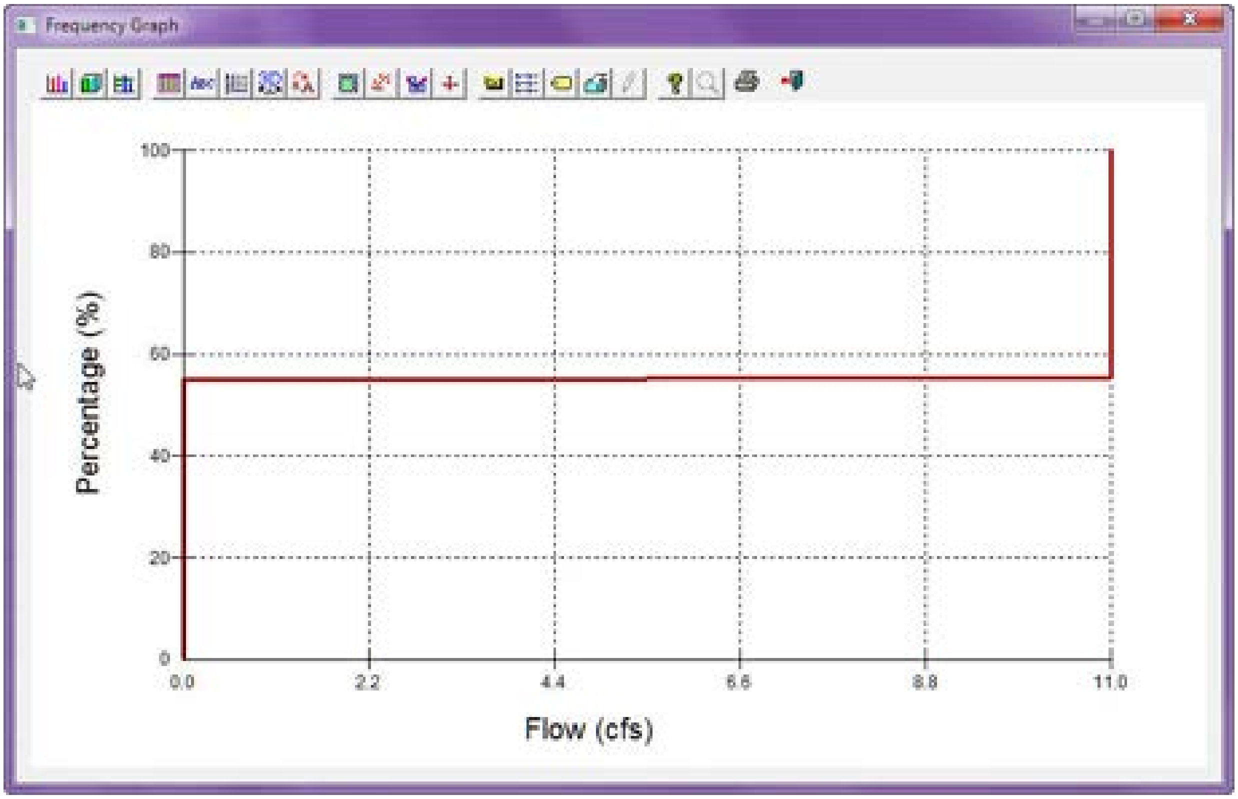
System Loading

Displays total system loading during an extended period simulation. The graph X-axis displays time in units as defined with the Simulation Times command while the Y-axis displays total sewage loading supplied to all manholes in load units.



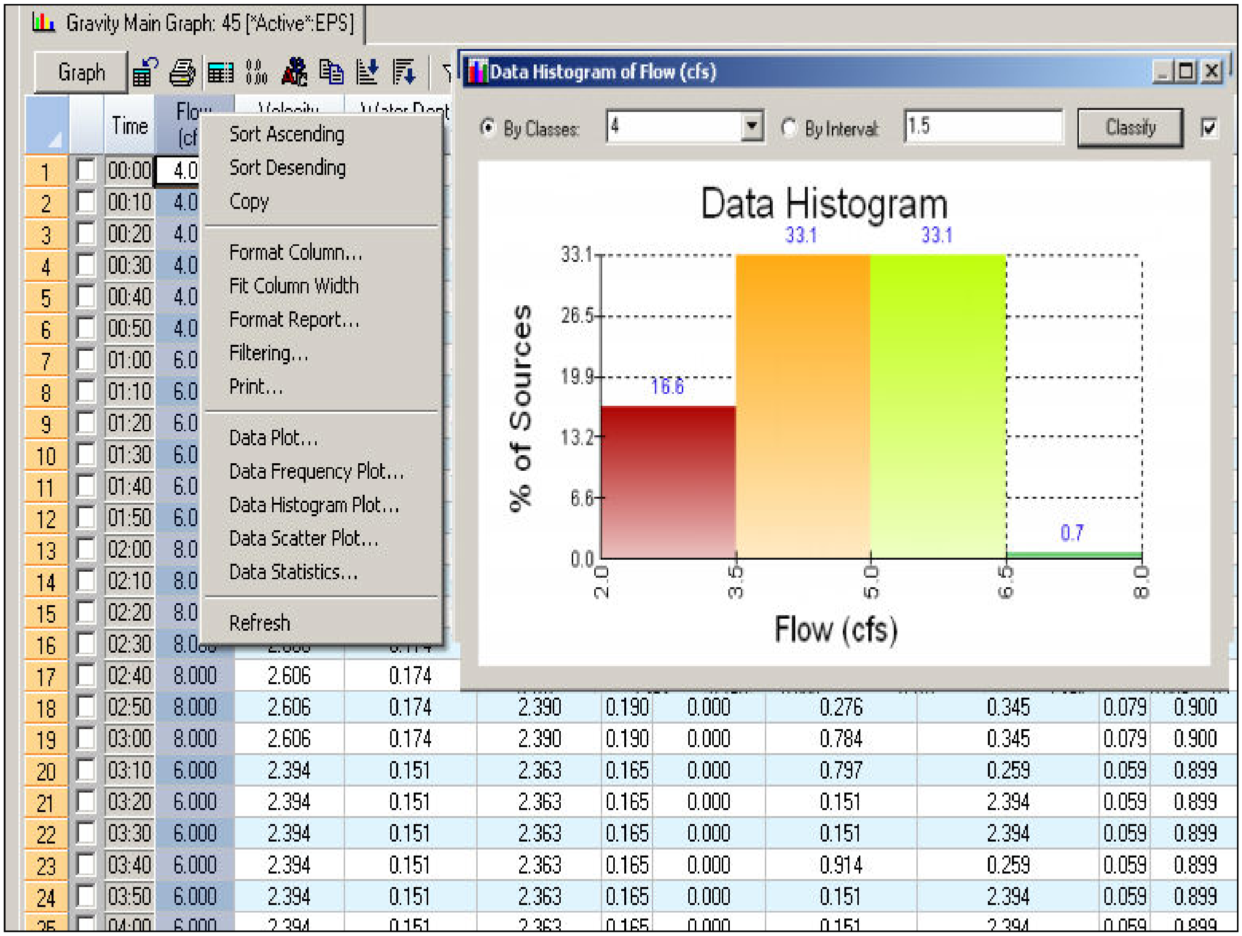
Frequency Graph

A frequency graph can only be created by viewing an output report. Once an output report is open (manhole, gravity main, etc), highlight the attribute (column) for which a frequency curve is desired and select the frequency icon at the top of the dialog box. A frequency curve will be generated as shown below.



Data Histograms

A histogram can only be created by viewing an output report. Once an output report is open (manhole, gravity main, etc), highlight the attribute (column) for which a histogram is desired and select the frequency command using a right mouse click. A histogram will be generated as shown below.



InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy

Steps to Compare Scenario Data

The Output Report Manager offers two comparison tools; a comparison report and a comparison graph. You may display a report or graph showing simulation results for the *active* (currently loaded) scenario, and then use a previously loaded output source storing results from a simulation run for a previously loaded scenario.

How Do I...

- **Compare Report Data?**
- **Compare Graph Data?**

Output Report Manager - Comparison Reports

 Compare Report Icon

In order to compare tabular data from two sources, you must first create the appropriate output source in which a comparison is to be made. Once this is done, you can click on the compare report icon to compare any output source with the current *active* report being viewed. Click here to learn a step-by-step process for creating comparison reports.

As an example of a comparison report, the illustration below shows result fields from the *active* scenario with the output source FUTURE3.Steady. Notice that the *active* scenario is not preceded with the output source name.

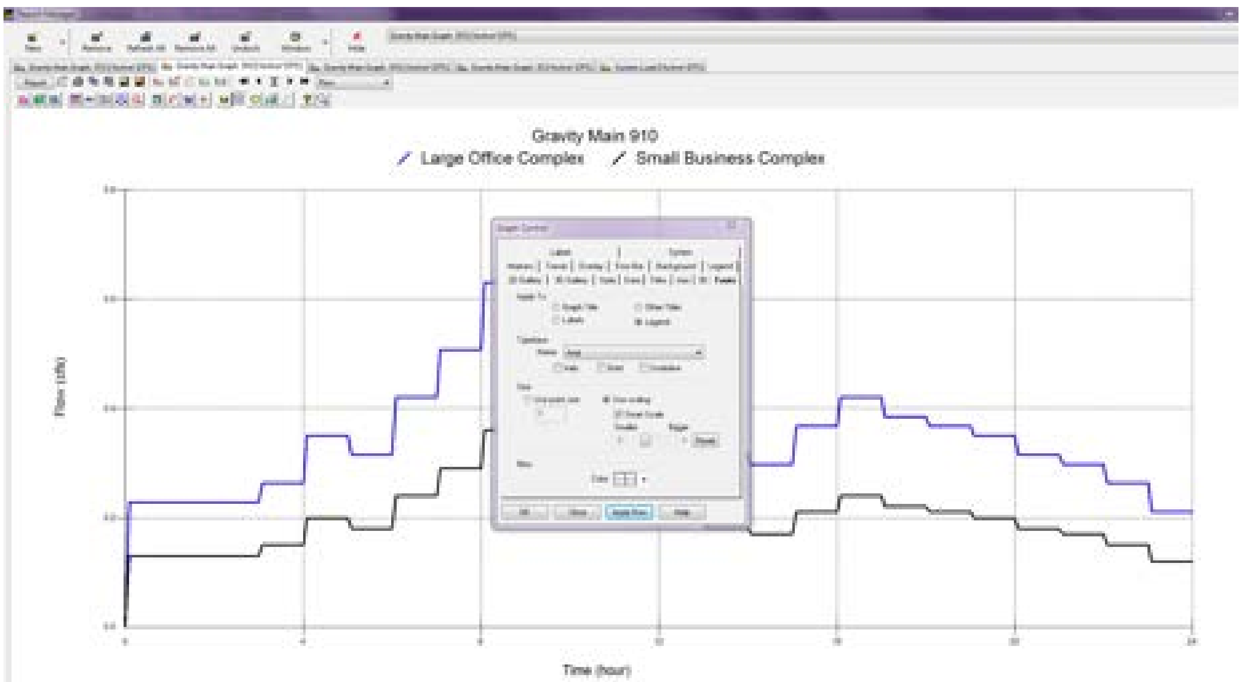
		ID	Actual d/D	Actual d/D [FUTURE3:Steady]	Analysis Flow (cfs)	Analysis Flow [FUTURE3:Steady] (cfs)
1	☐	99	1.00	1.00	0.25	0.25
2	☐	97	0.43	0.47	0.81	0.81
3	☐	95	0.76	1.00	1.10	1.10
4	☐	93	0.55	1.00	10.71	10.71
5	☐	92	0.77	0.77	3.57	3.57
6	☐	910	0.15	0.19	3.31	3.31
7	☐	90	0.47	0.47	0.39	0.39

Output Report Manager – Comparison Graphs

 Compare Graph Icon

In order to compare graphs from two sources, you must first create the appropriate output source in which a comparison is to be made. Once this is done, you can click on the compare graph icon (under the output graph manager) to compare any output source with the current *active* graph being viewed. Click here to learn a step-by-step process for creating comparison graphs.

As an example of a comparison graph, the illustration below shows values from the *active* scenario compared with the FUTURE3.EPS output source. Notice that the legend refers to the active output source as *Current*.



Steps to Develop Comparison Reports and Graphs

The following steps illustrate how to develop a comparison report or graph. These steps assume you have developed two or more scenarios and run a hydraulic simulation for each scenario. In essence, more than one output source is available.

Develop and Run Scenarios

- Develop the first scenario and then activate that scenario. For this illustration this scenario will be named “FUTURE1”, representing future conditions.
- Run an EPS simulation for the “FUTURE1” scenario using the Run Manager. Simulation results are stored in the *active* output source and are immediately available for evaluation.
- Develop the second scenario and then activate the second scenario. For this illustration, name the second scenario “EXISTING” to represent existing conditions. When you load the “EXISTING” scenario, simulation results for the “FUTURE1” scenario are moved from the *active* output source to a new output source named “FUTURE1”.
- Now run the second, recently activated EPS scenario named “EXISTING”. Simulation results are stored in the *active* output source, replacing simulation results from the previously-loaded scenario, and are now available for evaluation.

Develop a Comparison Graph

- Open the Output Report Manager (from the **Tools** menu, select **Output Report**)
- Choose the “New” button on the Output Report Manager window. The Output Report and Graph dialog box appears on the screen.
- Choose the *Active*:EPS output source (storing simulation results for the currently loaded “EXISTING” scenario), choose the Graph tab and the choose the Gravity Main Graph. Once selected, choose “OK”.
- Pick a pipe from the network map. A graph is displayed showing simulation results from the currently loaded “EXISTING” scenario for the selected pipe.
- Choose the Compare Graph icon at the top of the Output Report Manager to overlay results from the desired output source. The Select Output Source(s) to Compare dialog box appears on the screen
- Click once on the FUTURE1:EPS output source and then choose the “OK” button. The graph on the Output Report Manager should now show results for the currently-loaded scenario (“EXISTING”, referred

to as *Current* on the graph legend) and the previously-loaded "FUTURE1" scenario.
- You may now customize the graph as you desire using any of the available graph customization tools.

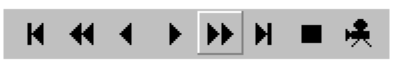
You may follow the same steps shown above to develop and customize a comparison report.

Animation Viewer

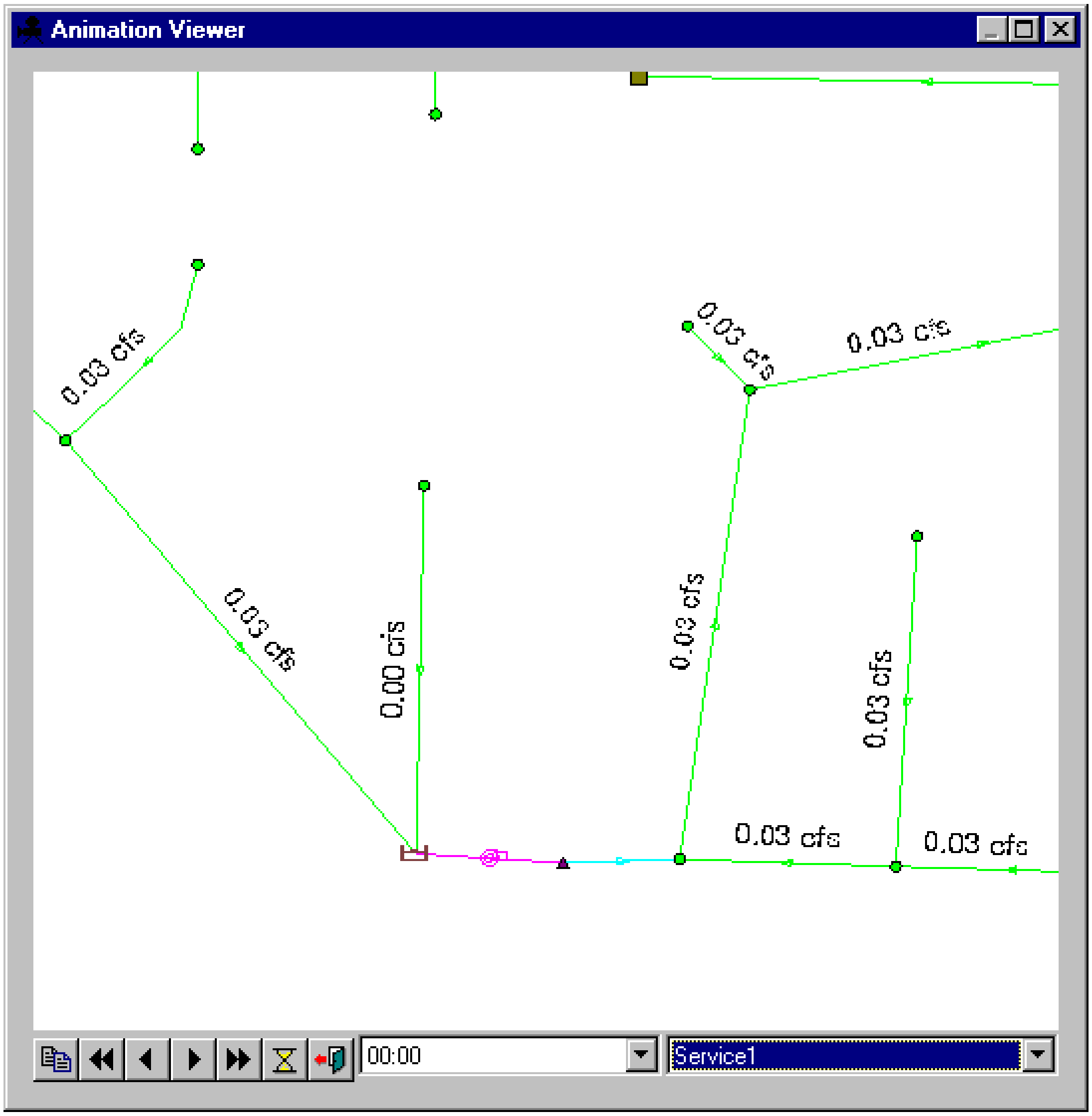
The Animation Viewer is used to retrieve and display map animations created and saved through the Animation Player found under the Map Display tab on the Control Center. Map animations display simulation results for EPS/dynamic simulation timesteps in succession, thereby creating a "movie" effect that allows the user to see and save simulation results to be viewed at any point in the future.

Creating an Animation

The graphic below shows the icons available from the Animation Player found under the Map Display tab of the Control Center. The last icon is the movie icon that allows map animations to be saved to a file that can later be played by the Animation Viewer.

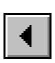


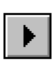
Once an animation is saved it can be viewed through the Animation Viewer dialog box shown below. To access this box, from the **Tools** menu, select **Animation Viewer**. Using the drop-down list at the bottom of the Animation Viewer dialog box, the user can select any saved movie to play a stored animation.



 **Copy Image** - Copy the image in the animation viewer to the Windows clipboard. With the image in the clipboard, the user is able to paste the image into any third party software program.

 **Fast Backword** - Play the animation in reverse at a refresh frequency as specified from the delay setting button.

 **Backward** - Go back one slide.

 **Forward** - Go forward one slide.

 **Fast Forward** - Play the animation in forward at a refresh frequency as specified from the delay setting button.

Display Setting - Specify a refresh rate for the animation (in milliseconds).

Exit - Exit the animation viewer.

test **File Name** - From the drop down box, select the desired animation viewer to show.

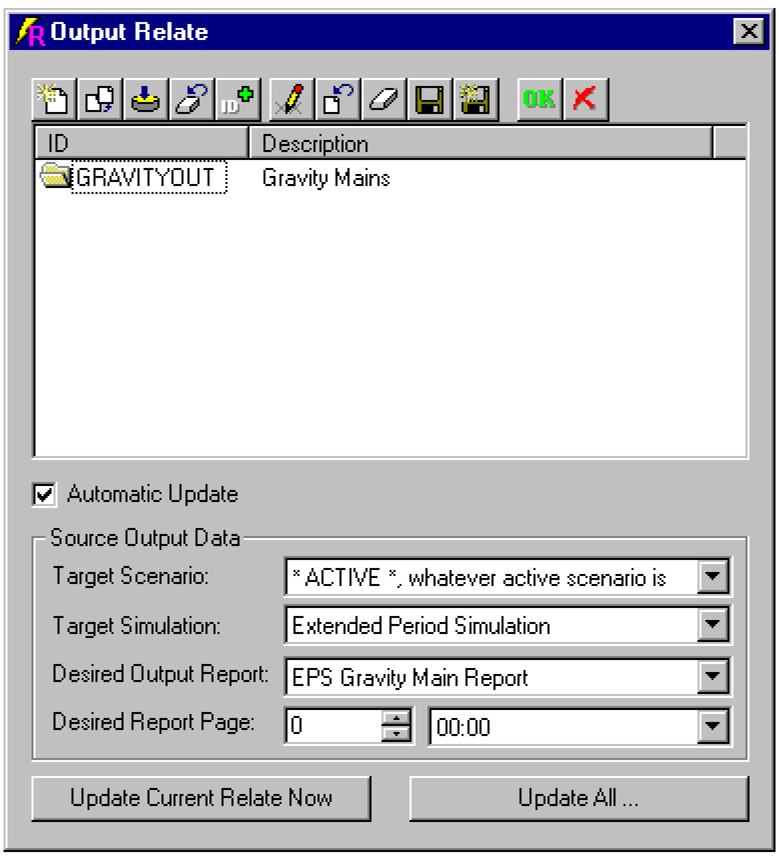
Output Relate

The Output Relate option is used to establish relationships between modeling input data and modeling results. Once a relate (link) is established, you can then apply queries against the current project based on modeling inputs and simulation results simultaneously. Click here to learn more about output relates.

Creating Output Relates

To create an Output Relate, from the Operation Data tab under the Control Center dialog box, select and highlight Output Relate. Right-mouse click on the folder and select New. At this point the user is asked to provide a name and description for the new output relate table. When created, the user is then shown the Output Relate dialog box as shown below:

Click on any portion of the box to learn more.



Automatically Update Output Relate Contents

When this option is ON (checked), the relate table will be automatically updated every time you run a simulation when the relate’s Target Scenario and Target Simulation are active.

For example, you define a relate based on the BASE EPS simulation and then set this option to ON. The next time you load the BASE scenario and run an EPS simulation, the contents of the relate table are automatically updated with the most recent simulation results.

If this option is OFF, the relate table contains simulation result data from when the relate was created or at the time you last updated the relate contents by choosing one of the buttons at the bottom of this dialog box. If you re-run the simulation (BASE EPS in the example above), the results from the previous run remain in the relate table.

Target Scenario

Select the scenario upon which the output relate is based. Data from the selected Target Simulation for the selected Target Scenario will be written to the output relate table.

You may choose from among the active (currently loaded) or any other custom-developed (output source) scenarios.

*(Note: If you have not developed any custom scenarios with the H2OMAP Sewer scenario management feature, the only available scenario will be *ACTIVE* scenario representing BASE conditions.)*

Target Simulation

Select the simulation type (steady-state, steady-state design, EPS) upon which the output relate is based. Data from the selected Target Simulation for the selected Target Scenario will be written to the output relate table.

Desired Output Report

Each output relate contains one single relate table. Choose the relate table that you would like to associate with the current output relate. The list of available tables will vary depending on which Target Simulation type you have chosen.

(Note: If you wish to store simulation results for all network component types (gravity mains, force mains, pumps, manholes, wet wells, outlets and chamber manholes) in output relates, create seven separate relates, select the same Target Scenario, Target Simulation, and Report Page for each relate, and then choose the desired output report table for each relate.)

Desired Report Page (Timestep)

Output relates store simulation results for a specific modeling condition. For an EPS model, there are two methods by which you may select a time period for which simulation results will be written to the output report table:

- **Report Page** - Choose the extended period simulation report timestep by its index (page) number. The first simulation timestep is page/index 0 (zero), the second simulation timestep is page/index 1, and so on. Use this option when there are no available simulation timesteps.
- **Available Timesteps** - Choose the extended period simulation report timestep by its clock time from the beginning of the simulation. Timesteps are not available until you have successfully run a simulation of the selected Target Simulation type for the selected Target Scenario. If your hydraulic timestep is more than 1, the available timesteps may not match the report page. For example if your hydraulic timestep is two hours, the following relationship exists between report page and available timesteps:

Report Page	Available Timestep
0	0:00 hr
1	2:00 hr
2	4:00 hr

Use the Simulation Time option under the Operation Data tab under the Control Center to specify hydraulic, report, and other extended period simulation time steps.

Update Current Relate Now

Updates the contents of the output relate table with simulation results from the most recent run of the associated Target Simulation Type for the associated Target Scenario. Simulation results are written for the associated Report Page or Report Timestep.

Update All Defined Relates

Updates the contents of all currently defined output relate table with simulation results from the most recent run of each output relates' associated Target Simulation Type for each associated Target Scenario. Simulation results are written for the associated Report Page or Report Timestep.

...More About Output Relates

Output Relates provide a method of temporarily transferring simulation result data to H2OMAP Sewer's model input database to support database operations (such as queries) using the output data. When you execute an Output Relate, simulation results stored in a standard output source you select are written to the desired output report table (e.g., Manhole Report, Pipe Report, etc.). The table contains simulation results for all network components at a single extended period simulation timestep. Refer to the Output Report Manager for more information on the format and contents of each output report.

Each output relate is identified by a name and narrative description and may be accessed by that name when developing query statements using several H2OMAP Sewer commands and tools.

The data transferred as a result of the relate is a "snapshot" of simulation results and will therefore be static regardless of subsequent simulation runs. To update the contents of an output relate with results from a more recent simulation run, use either the Update Current Relate Now or Update All buttons. The Update operation replaces the original related output data with the new output data. You must specify the Target Simulation (output source such as steady-state, steady-state design, EPS) and the Desired Output Report (Manhole Report, Pipe Report, etc.) which the output data will be mapped to. The related output will remain intact for later use (i.e., after the project is closed). Both the Query Set command and the Query Builder dialog box can access output relate data for use in developing query sets and query statements.

Results in an output relate can be made static, as a permanent snapshot of a single model run, or can be automatically updated every time you rerun a simulation for a given scenario. To make updates automatic, from the **Tools** menu, select **Preferences**. Under the Operation tab, select Auto Output Relate Update.

Output relates are stored in an external database file that can be linked to in a third party GIS software to view analysis results. Click here to learn more about viewing results in a GIS platform.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

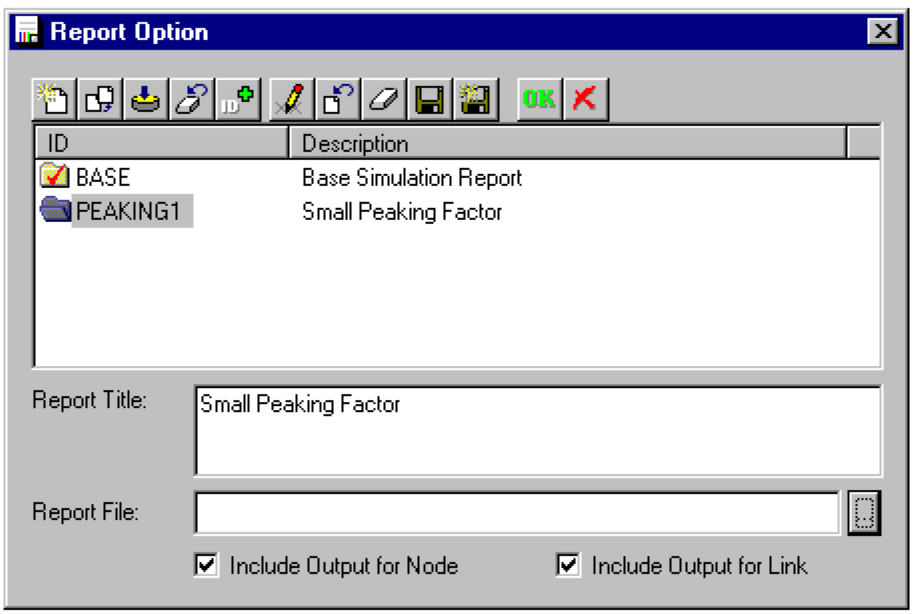
Simulation Report

A simulation report can be created as part of a scenario or simulation run and is required whenever you wish to have the native analysis model report exported to a specific ASCII text file that differs from the default file.

While you can easily copy the contents of any report, customized report or query report to the Windows Clipboard to be viewed in any third party package, a simulation report contains the results from the model run itself, which contain model specific data. Model specific data consists of items like error messages, pump status, trial runs, element actions at timesteps, connectivity problems, etc.

To create a simulation report, from the Operation Data tab in the Control Center, click on the Simulation Report and select the New icon at the top of the Control Center. Enter a name and description for the report, click OK and the following dialog box will appear.

Below the following graphic is a description of the options available.



- **Report Title** - The title header to be shown at the top of each page in the simulation report.
- **Report File** - The name and network location of the simulation report.

- **Include Output for Node** - Provides node (manhole and wet well) data to be included in the simulation report.
- **Include Output for Link** - Provides link (pipe and pump) data to be included in the simulation report.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

GIS Overview

Geographic Information Systems (GIS) is the term used for establishing a graphical link to a database and then adjusting some facet of a digital map based on the data within that database.

H2OMAP Sewer is, by its very nature, a GIS-based software program. Pipes and nodes are digitized (graphics) and a database table is kept on each element created in the H2OMAP Sewer project. By clicking on any element in H2OMAP Sewer, data related to that element can easily be seen in the Attribute Browser which is tied to an external database stored under the project directory.

- GIS Interface - Illustrates how H2OMAP Sewer data elements and contour information is viewable in other GIS software packages.
- GIS Exchange - Allows the user to transfer data between geodatabases, ArcSDE and other other GIS data formats. The GIS Exchange is also where spatial joins (such as 'Smart Topology') between an imported layer and an H2OMAP Sewer layer can be performed.
- Add SDE Layer - Allows the importation of ArcSDE and ArcINFO Geodatabase layers to the H2OMAP Sewer project.
- Hot Links - Allow the user to attach external data files, such as pictures or engineering drawings, to any H2OMAP Sewer data element.

InfoSewer™

Modeling Made Easy

GIS Interface

The full power of InfoSewer lies in the ability to view model data and model results in a third party GIS package like ESRI ArcView.

Viewing InfoSewer Data in ArcView

When data elements are created in InfoSewer, the graphic portion of the element is automatically stored under the InfoSewer project directory as an ESRI shapefile (see the Map and Contour directories under your project.DB directory). This is possible because InfoSewer is built with ESRI MapObjects software which is fully shapefile compliant.

For every data element created in InfoSewer, there is also one or more external database files kept on that element. The fields contained in these databases are viewed and edited with the Attribute Browser (ex. diameter for pipes, elevation for manholes, etc). Because each database record on each element has a unique ID supplied by you, the user - this ID field is then the common field for which data can be joined and viewed in a third party software package like ArcView. All database fields related to InfoSewerdata elements are stored under the project.DB directory. Merely "Add" any of these tables to ArcView and use the ArcView Join command to view InfoSewer data in ArcView.

Viewing InfoSewer Model Results in ArcView

Not only can you view the original link and node data in ArcView by joining the databases together by the ID field, you can also view output results by the same process. As explained in the Output Relate section, a model run, once performed, is stored in memory until the user tells InfoSewer to create a physical database of the results. This is done by creating an Output Relate and specifying which pieces of information should be written to this external file.

Once you have created an output relate and performed a hydraulic analysis, save the InfoSewer project and the data for that model run is stored in the Relate directory under the project.DB directory. By adding the link and node shapefiles under the Map directory and adding the desired output relate tables from the Relate directory, model results can be viewed, color graduated and plotted from ArcView. Simply use the Join command in ArcView to merge the two separate databases into one.

Viewing Contours in ArcView

As with model results, the same holds true for contours created with InfoSewer. Simply use the Add theme command in ArcView and add any contour layer located under the Project.DB\Contour subdirectory. All database tables used in the creation of the contour file are contained in the shapefile and are also available in ArcView.

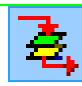
Viewing InfoSewer Data in Other GIS Packages

As stated in the Import, Export and ODBC sections, the user is able to view and manipulate InfoSewer data in any number of ways. To learn more about each of the following features, click on any of the links below.

- **Import** - Background mapping, external hydraulic models or InfoSewer data elements.
- **Export** - InfoSewer data elements or model results.
- ODBC - Download and upload data to and from InfoSewer by creating data links.

InfoSewer™

Modeling Made Easy

GIS Exchange 

The GIS Exchange allows the user to quickly import and export data between InfoSewer and other GIS formats. It also allows the user to perform spatial joins between an imported GIS layer and an InfoSewerdata table.

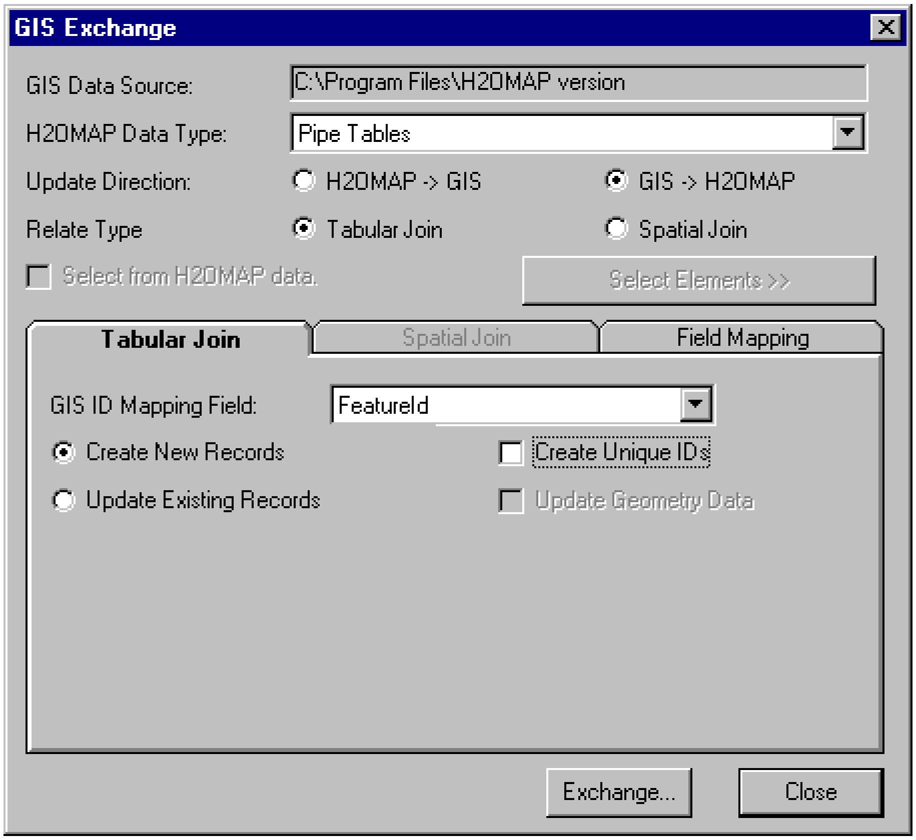
Running GIS Exchange

To run a GIS Exchange, you must first import a GIS layer for which an exchange is desired. From the **View** menu, select **Add Layer**. Once the add layer dialog box opens, point to the GIS layer on the network drive to add it to the InfoSewer session. Add the layer using your mouse and click on the GIS Exchange icon at the top of the control center. You will now see the GIS Exchange dialog box and be able to select the type of join to be performed by the GIS Exchange.

This process is also applicable to ESRI geodatabases and ArcSDE Layers. By adding an SDE layer, you are also to perform a GIS Exchange with the highlighted GIS layer.

Tabular Joins

A tabular join merely exchanges data fields between one GIS data source and another. Use the GIS ID Mapping field to denote which field in the GIS layer will be matched with the respective ID field in the selected InfoSewer data table.



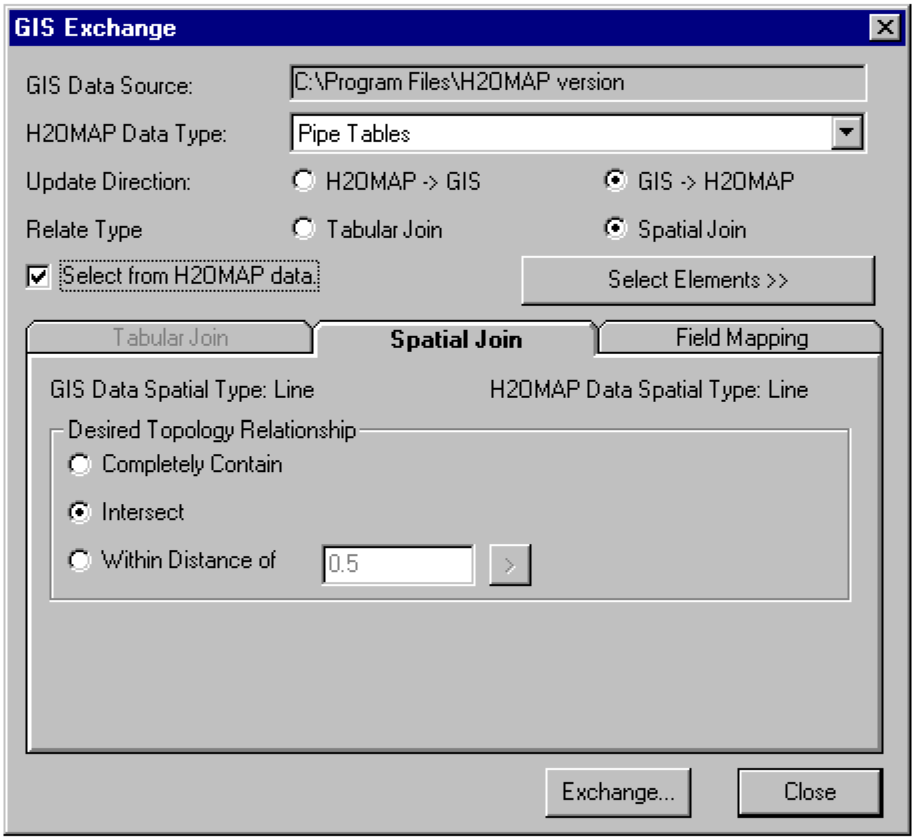
- **GIS Data Source** - The file that will be used for exchanging data.
- **H2OMAP Data Type** - The InfoSewer Element (and associated tables) that will be used in the exchange.
- **Update Direction** - Specify the direction for the GIS Exchange.
- **Relate Type** - During a tabular join, select the Tabular Join radial button.
- **GIS ID Mapping Field** - The Unique ID field that will be mapped to the corresponding ID field in InfoSewer
- **Create New Records** - When this option is selected, new records will be created. Use the Field Mapping tab to map the corresponding fields.
- **Create New ID's** - By checking this box, InfoSewer will automatically assign unique ID's to the elements being exchanged.
- **Update Existing Records** - This option will update matching fields as denoted in the GIS ID Mapping Field. Use the Field Mapping tab to map the corresponding fields.

- **Update Geometry Data** - When this box is checked, the physical geometry of the data file being exchanged will be updated to reflect recent changes.

Once the tabular join options have been specified, proceed to the Field Mapping tab to map the desired fields prior to exchanging the data.

Spatial Joins

A spatial join allows the user to perform a join between two data layers based on their physical relationship to one another. This means that the data contained in one data layer can be assigned to another table by using 'smart topography' features of InfoSewer.

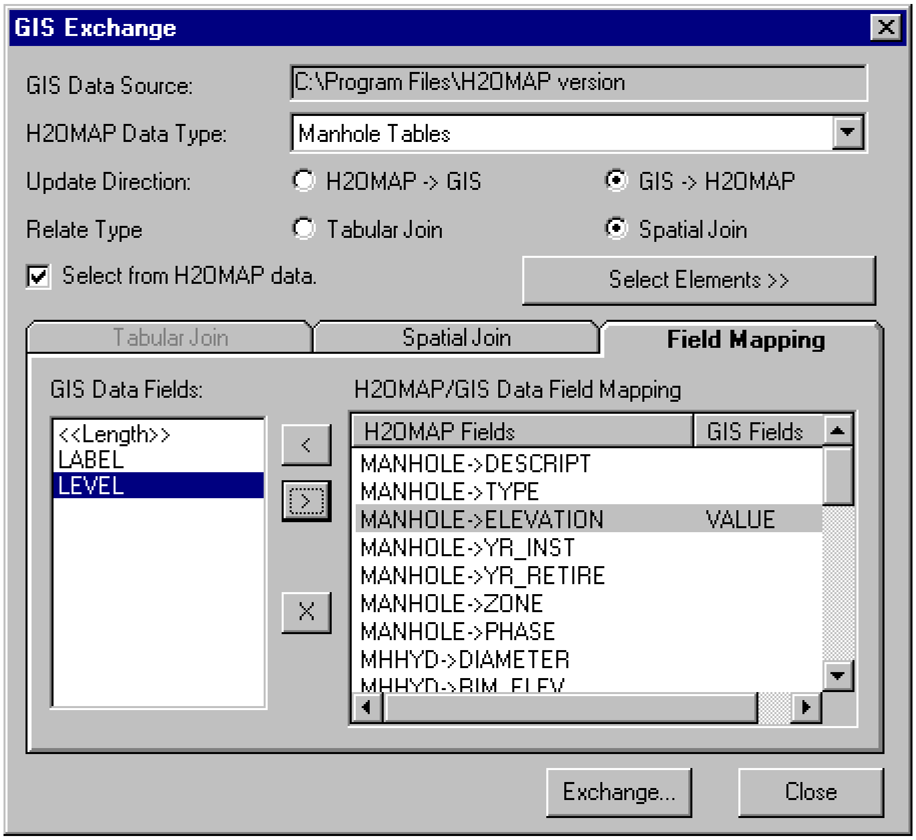


- **GIS Data Source** - The file that will be used for exchanging data.
- **InfoSewer Data Type** - The InfoSewer Element (and associated tables) that will be used in the exchange.
- **Update Direction** - Specify the direction for the GIS Exchange.
- **Relate Type** - During a spatial join, select the Spatial Join radial button.
- **Select from InfoSewer Data** - Using a graphical selection, the user can graphically select which elements are to be included in the spatial join process. When this box is checked, click the Select Elements button to make a graphical selection.

- **Desired Topology Relationship** - The user must specify the type of spatial join to be utilized during the spatial join
- ***Completely Contain*** - Used to exchange data between a point or line data layer and a polygon. Those elements that are 'completely contained' in the polygon data set will be tagged for the spatial join. Those elements not completely contained will be excluded.
- ***Intersect*** - When two data layers are intersected, those elements that are contained within and/or cross the intersected layer are tagged for the spatial join. Those elements that do not have a physical intersection are excluded.
- ***Within Distance of*** - Those elements that are within the user specified distance will be tagged for a spatial join. This option is specified to assign contour elevation data to a point element such as a manhole rim elevation. In other words, any elements of the selected layer that fall within the tolerance level specified will be tagged for a spatial join.
- Once the spatial join options have been specified, proceed to the Field Mapping tab to map the desired fields prior to exchanging the data.

Field Mapping

The field mapping tab allows the user to map corresponding fields to one another prior to performing the GIS Exchange. Merely select fields on the left hand side to be mapped to InfoSewer field on the right. Once all field mapping has been completed, click on the Exchange button to finish the GIS Exchange.



- **GIS Data Fields** - The fields found in the GIS data set that can be linked to the InfoSewer data fields. Use the arrow buttons to map one field to another.
- **InfoSewer/GIS Data Field Mapping** - TheInfoSewer fields that are available to be mapped during a GIS Exchange. By using the arrow buttons, select the desired exchange fields to map to the InfoSewer fields.

InfoSewer™

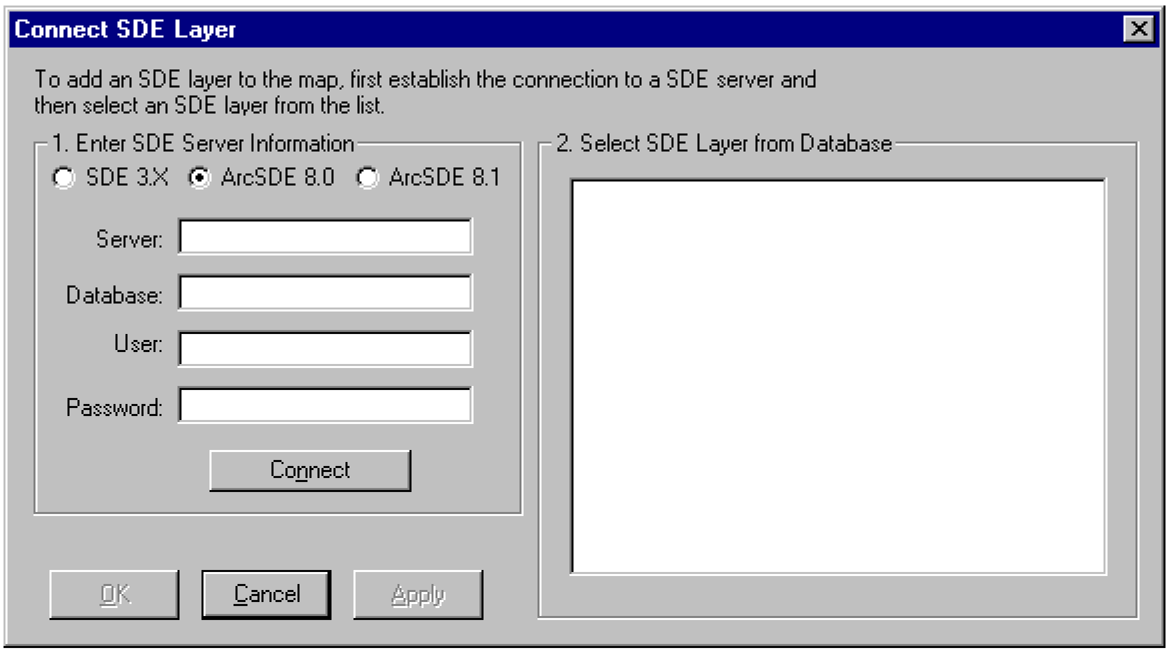
Modeling Made Easy

Home > Add SDE Layer

Add SDE Layer

Adding an SDE Layer for viewing in InfoSewer.

From the **View** menu, select **Add SDE Layer**. Once this is done, the following dialog box will be displayed. Specify the SDE Server and database for which a connection is to be made. The layer specified will now be added to the InfoSewer display.



InfoSewer is built entirely with ESRI MapObjects component technology. This means that the program is ESRI Geodatabase compliant. In order to "transfer" data between a geodatabase and InfoSewer, a connection must first be made. This is done by adding an SDE Layer to InfoSewer and then using the GIS Exchange button from the Map Legend. By adding the SDE Layer, the path of the layer will stay resident in memory so that when a GIS Exchange is performed, the path is resident. Click here to learn more about GIS Exchanges.

InfoSewer is an Extension for ArcMap 10.3 to 10.8.1

Innovyze Help File Updated March 31, 2021

More Questions? Further Help Can be Found by Emailing Support@Innovyze.com or visiting https://www.innovyze.com/en-us/support-overview

InfoSewer™

Modeling Made Easy