Innovyze®

Introduction to
InfoSewer

InfoSewer Lecture

InfoSewer

ACCREDITED IACET PROVIDER

ACCREDITED

Continuing Education & Training Provider

RCEP

REGISTERED CONTINUING EDUCATION PROGRAM

innovyze.com

Introduction to InfoSewer Training Seminar

Welcome to the Introduction to InfoSewer Training Seminar. This two-day course will introduce you to InfoSewer, the most complete wastewater collection system modeling and management software application available. Upon completion of this course, you will have complete knowledge of InfoSewer and will be able to maintain, run, and analyze stormwater and wastewater collection system models and databases.

The learning objectives for the training seminar are as follows:

1. Understanding the InfoSewer structure, organization, and terminology.
2. Understanding the InfoSewer software graphical user interface.
3. Building network models.
4. Running hydraulic water quality models.
5. Performing specialized analyses including scenario management.
6. Querying and reviewing model results.
7. Customizing the InfoSewer database.
8. Sharing InfoSewer information with other applications including spreadsheets and GIS.

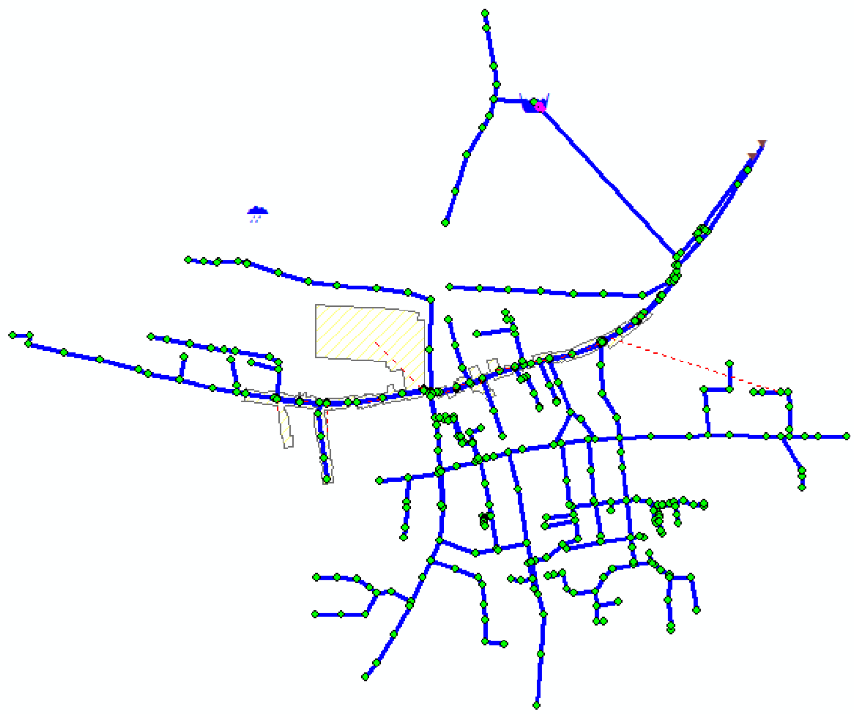
The agenda for the training seminar is as follows:

Section 1. InfoSewer Product Overview
Section 2. The InfoSewer Project
Section 3. Creating an InfoSewer Model
Section 4. Working in the InfoSewer Interface
Section 5. Building Hydraulic Model Parameters
Section 6. Running a Steady State Analysis and Viewing Results
Section 7. Running and Extended Period Simulation and Viewing Results
Section 8. Design Simulation
Section 9. Scenario Management
Section 10. Storm Modeling
Section 11. Water Quality Modeling
Section 12. Additional InfoSewer Features

In addition to each section, there will be software demonstrations and hands-on exercises. The training data for this course will include the following model:

The model

The same model will be used for all exercises.



Training Agenda

Each section will include a short lecture and demonstration period followed by one or more comprehensive hands-on exercises (except 1 and 2).

The class, although comprehensive, will be conducted in an informal manner. You are encouraged to ask questions at any time.

There will be a break for lunch and a break each morning and afternoon. A class review session will be held right before the lunch break and at the end of each day.

Chapter 1

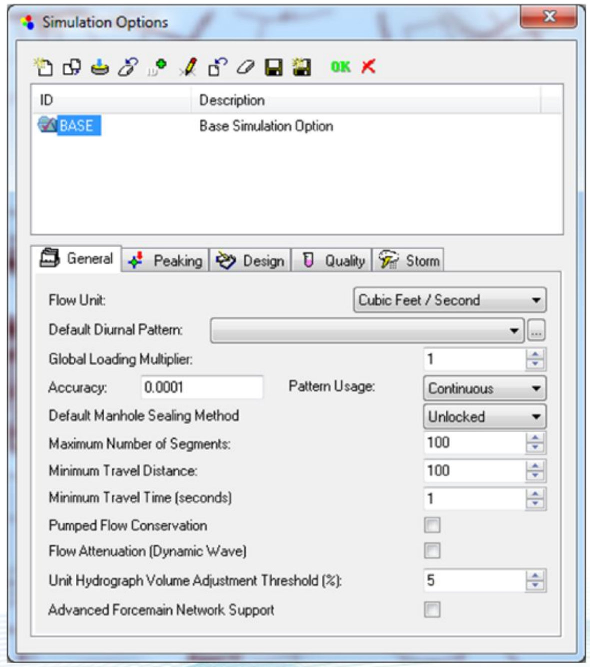
InfoSewer Product Overview

Innovyze®

Student Notes:

InfoSewer Capabilities

- Master Planning
- Flow Assessment
- System Design
- Operational Studies
- Development Assessment



Innovyze®

Page 1-2

Student Notes:

Master Planning – Identify system deficiencies and recommend system improvements.

Flow assessment – Analyze system flows for both steady state and extended period simulations.

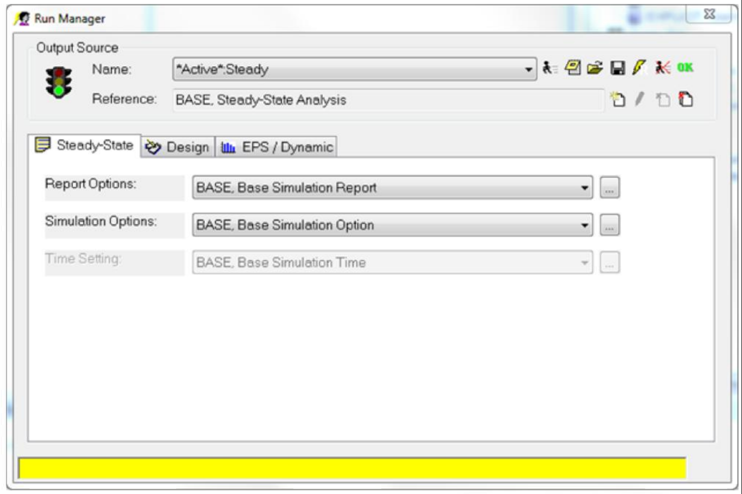
System design – Perform a steady state design simulation to determine the size and costs for parallel and/or replacement pipes for those pipes that do not satisfy a specified d/D or velocity criterion.

Operational studies – Analyze pump settings and their effect on system performance, and identify optimal wet well pump scheduling policies.

Development assessment – Assess the impacts of "what-if" scenarios on system performance.

InfoSewer Features

- Collection system modeling
- Graphical model creation
- Steady-State Modeling
  - Analysis
  - Design
- Extended Period Simulation (Dynamic)



Student Notes:

InfoSewer is a fully interactive, state of the art, multi-application software program for use in the modeling of sewer collection systems. InfoSewer combines a dynamic point and click interface for network construction and database management with a highly advanced and computationally efficient hydraulic analysis module.

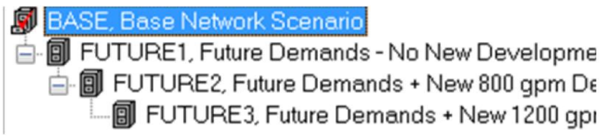
Graphical model creation – InfoSewer uses ArcGIS 10.0 or above tools to develop sewer or storm water collection system network models.

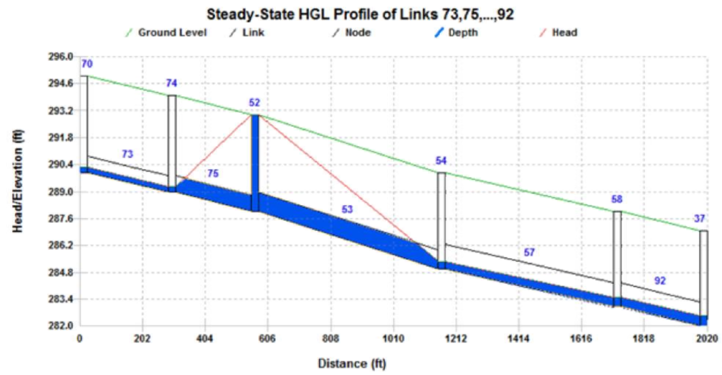
Steady state modeling – InfoSewer provides for both steady state analysis and steady state design simulations. Both simulations utilize either the Federov or Harman and Babbit peaking equations. Either method results in sewer flows being routed and peaked for pipe design and analysis.

Extended period simulation – InfoSewer is also unique in that it provides comprehensive extended period simulation analysis capabilities by routing and attenuating sewer flows. Pipe profiles can be generated to illustrate how flows vary across a pipe or range of pipes throughout a simulation.

InfoSewer Features

- Scenario Management
  - Comprehensive parent-child relationships: Inheritance
- Pump Controls
- Model Result Analysis
- Integration with other information systems





Innovyze®

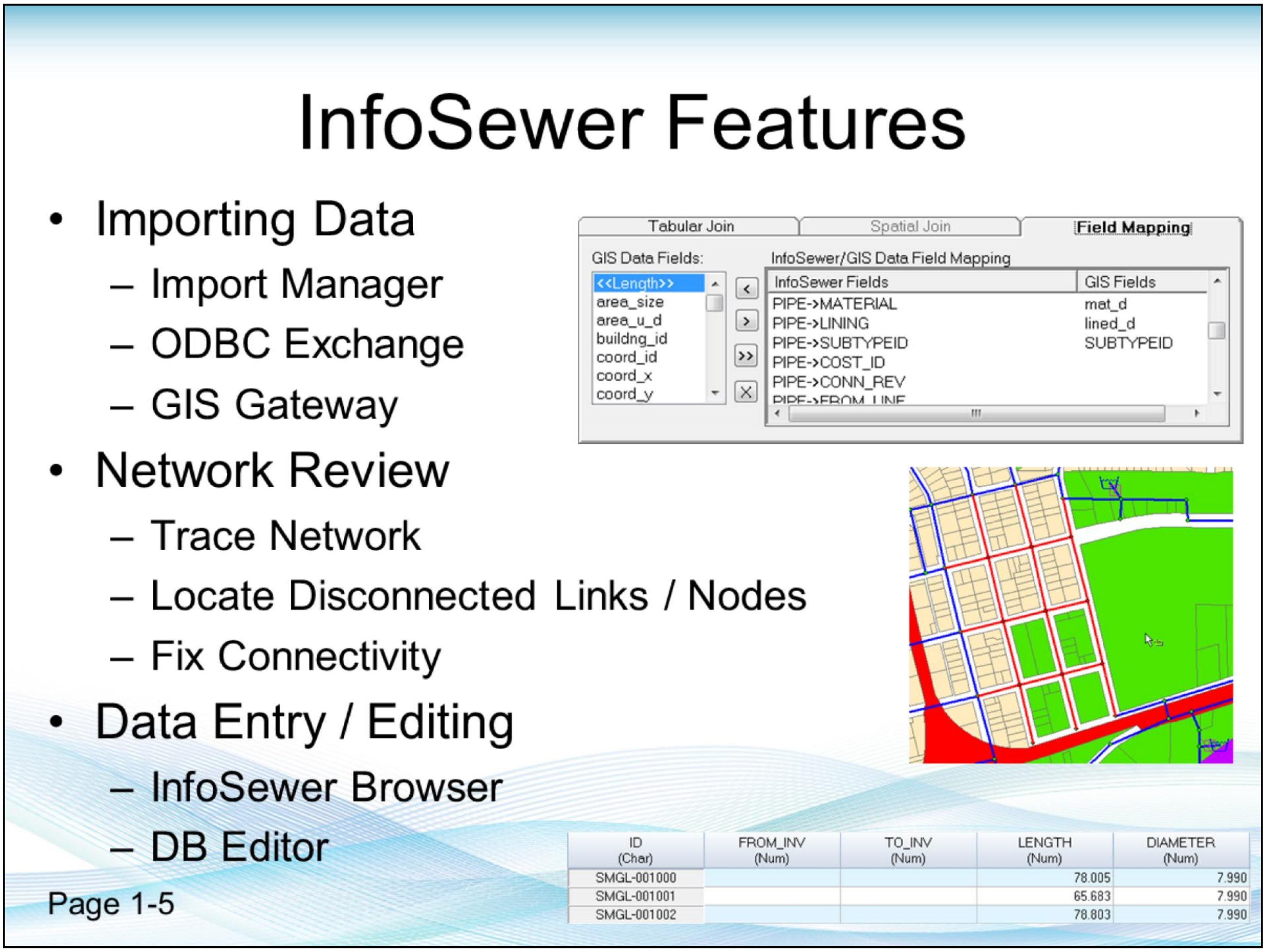
Student Notes:

Scenario management – With InfoSewer’s scenario manager, you can develop multiple specific modeling scenarios for a given collection system model. The user can formulate and analyze numerous modeling alternatives based on differing network facilities, loadings, and operating conditions.

Pump controls – Using simple control statements, vary pump operation according to either water levels or stored volume in adjacent wet wells. Place multiple pumps in parallel to increase pumping capacity.

Model results – Steady state analysis, steady state design or extended period simulation results can be reviewed in graph form, tabular form, TIN-based contours, map symbols, colors, and/or labels.

Integration with other information systems – InfoSewer provides linkages to other utility management and operation tools including GIS, Facilities Management, and any Database Management System (DBMS).



Student Notes:

Model input capabilities

Model input data can be imported into InfoSewer from existing data sources or entered directly into InfoSewer.

The user can import data from other information systems such as facility management databases or a GIS directly into InfoSewer. The InfoSewer database can be fully customized so that any additional information from external databases can be directly imported into the program.

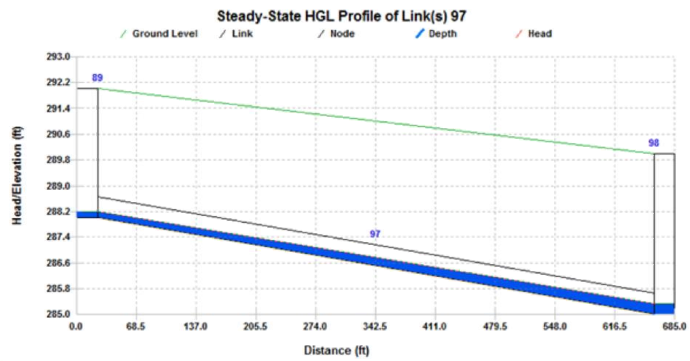
InfoSewer provides a full suite of graphical network creation tools while ensuring proper network connectivity throughout the network creation process. As an alternative to creating the network in InfoSewer, the network map can be imported directly from a GIS or can be built using location and connectivity information provided in a facility management database.

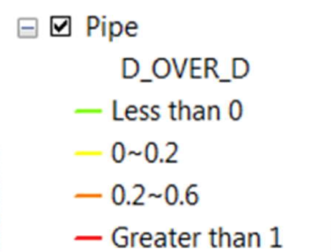
Information associated with each network facility can be entered and modified using the InfoSewer Browser or can be entered and modified in a spreadsheet-like format using the InfoSewer Database Editor.

InfoSewer Features

- Model Review
  - Graphs
  - Reports
  - Thematic Mapping
  - Animation

ID	Total Flow (cfs)	Unpeakable Flow (cfs)	Peakable Flow (cfs)
1	0.0500	0.0000	0.0500





Innovyze®

Student Notes:

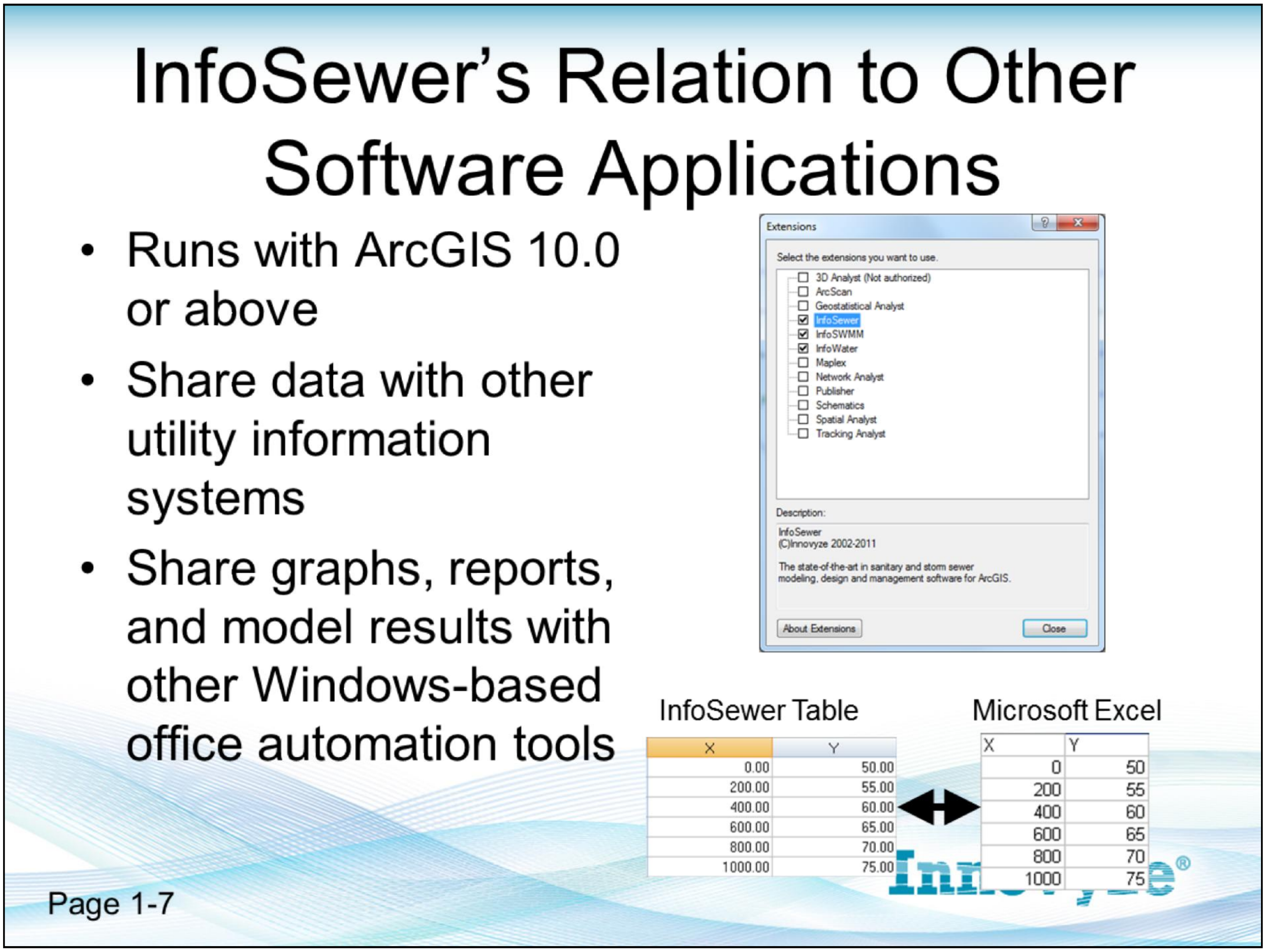
Reviewing model results - Map Display

InfoSewer provides the ability to view model inputs and model results directly on the network map display.

InfoSewer can display element identifiers and color-code elements based on model inputs or model results at any model simulation timestep. You can also label the entire collection system map or any user-selected portion of the map.

Contours can be generated for any nodal numeric model input or simulation result value. You can control contour interval, colors, labels, and generate contours for all, or selected portions, of the sewer collection system.

Map colors, symbols, and text sizes can be easily customized at any time. Range breaks and the colors associated with each range are fully controlled by the user.



Student Notes:

Runs within ArcView – InfoSewer runs entirely within ArcGIS 10.0 or above. While running InfoWater, you have simultaneous access to InfoSewer commands, ArcView commands, and other Windows applications.

Share databases with other utility information systems – InfoSewer's databases are stored in an industry-standard dBASE format. You can import and export data with other databases and spreadsheets (Oracle, Access, Foxpro, Lotus, Excel, and others) via the ODBC module.

Share graphs and reports with other Windows-based office automation tools –

Using Windows copy and paste tools, you can export InfoSewer graph and report displays to any other Windows-based program via the Windows clipboard. You may copy and paste report data into spreadsheets as delimited text.

Chapter 2

The InfoSewer Project

Student Notes:

Project Overview

- Each project contains:
  - Network Map
  - Network Component Attributes
  - Modeling Parameters
  - Model Solution and Results
  - Network Scenario

Page 2-2

Innovyze®

Student Notes:

The Project is the basic working entity within InfoSewer. One project contains the model data for a single collection system model. When running a model, InfoSewer simulates open channel hydraulics for the entire active network.

Network scenarios – Each model contains one or more modeling scenarios. Each scenario is a unique combination of facility sets (entire network or subset of the network), data sets (facility attributes, loadings), simulation options, and simulation results (option sets).

Network map – The graphical representation of the network.

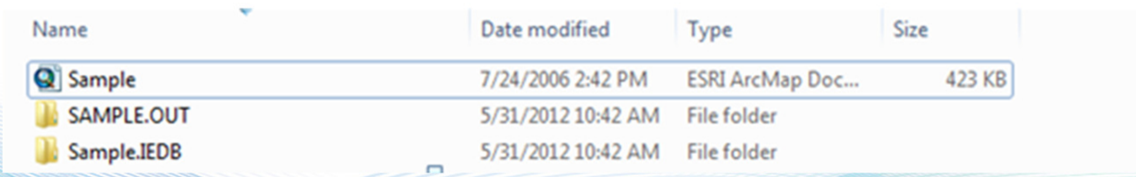
Network component attributes – Model inputs associated with each network component. There is one attribute record for each network component.

Modeling parameters – Parameters defining the current model run. These include units, model duration, time steps, loadings, initial operating status, and operational controls.

Model solution and results – The mathematical solution to the model, stored in binary format. Each component activated at the time of the simulation contains a result. Results are stored as Output Sources (a set of results for a single simulation run).

Project Storage Structure

- Network schematic
  - Stored as *.MXD file
- Database tables and modeling files
  - *.IEDB
- One directory containing model results
  - *.OUT folder created after a successful simulation run



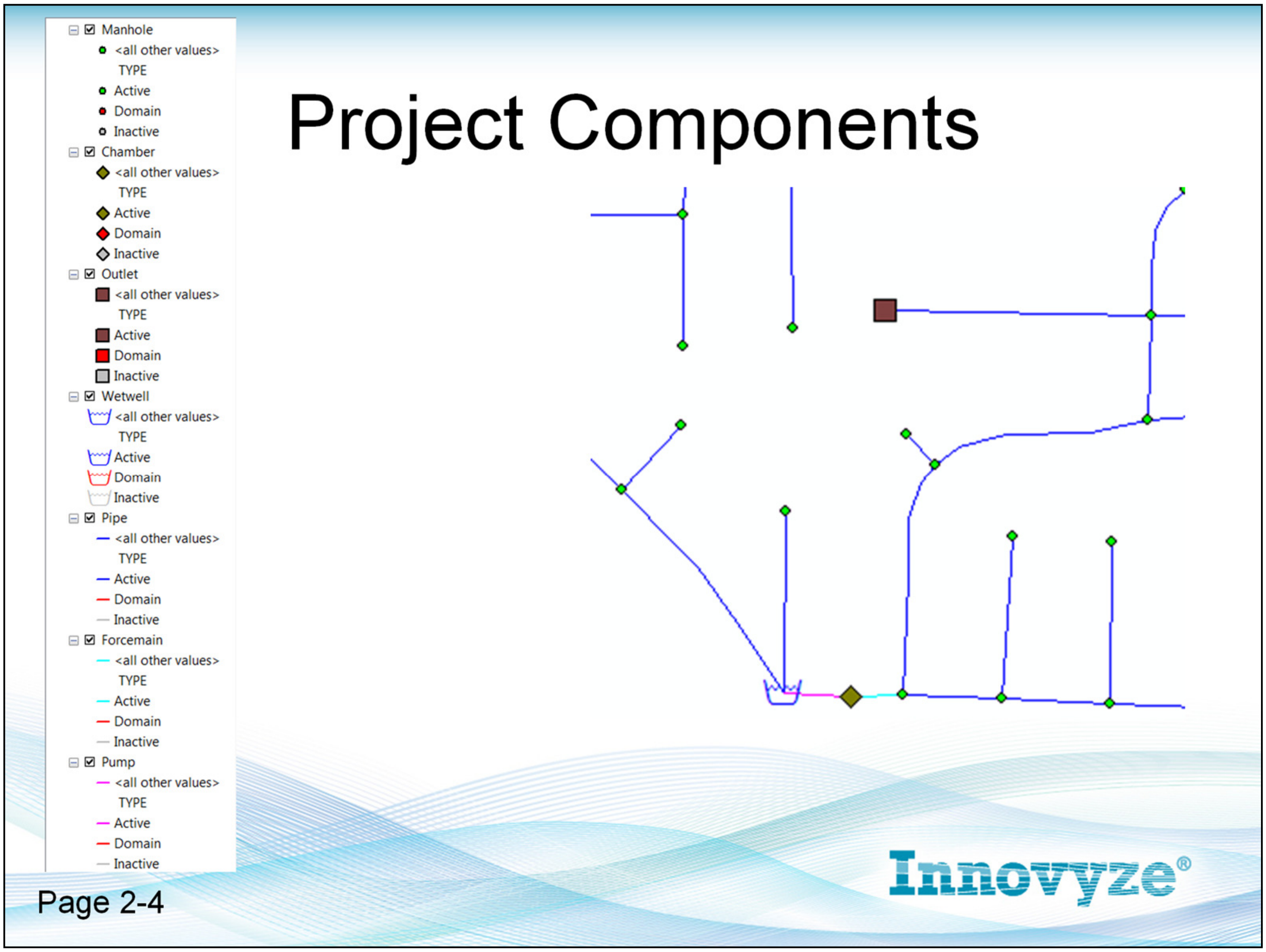
Student Notes:

One network schematic file – Named [PROJNAME.MXD]

One directory containing project database files – The directory is named [PROJNAME.IEDB] and contains all database tables storing component attributes and model parameters. Data for the default scenario and all custom scenarios are stored in subdirectories in the PROJNAME.IEDB folder.

One directory containing simulation results – The directory is named [PROJNAME.OUT] and contains simulation results (output sources). Simulation results for the default scenario and all custom scenarios are stored in subdirectories in the PROJNAME.OUT folder. **NOTE:** This folder does not exist until you complete the first simulation for a given model.

One directory (each) containing contours, annotation and network topology – The directories are named [CONTOURS], [ANNOTATION] and [MAP] respectively and are located under the [PROJNAME.IEDB] directory. Once a graphic of any sort appears in the InfoSewer display, it is stored as a shapefile under one of these directories.



Student Notes:

Collection systems in InfoSewer are represented by following model component types:

Normal/Chamber/Outlet Manholes – Normal manholes represent locations where pipes intersect and where loads are allocated to the collection network. Chamber manholes connect pumps to force mains. Outlet manholes are required to model outfalls or other points of discharge from the system.

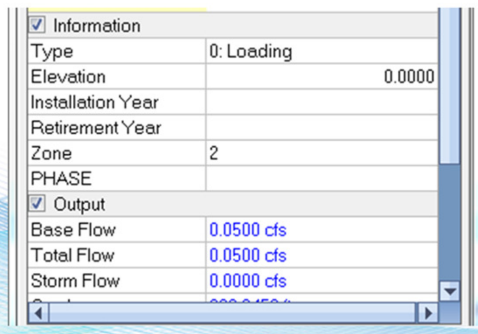
Wet Wells - Represent locations of wet wells that comprise the storage element of a lift station.

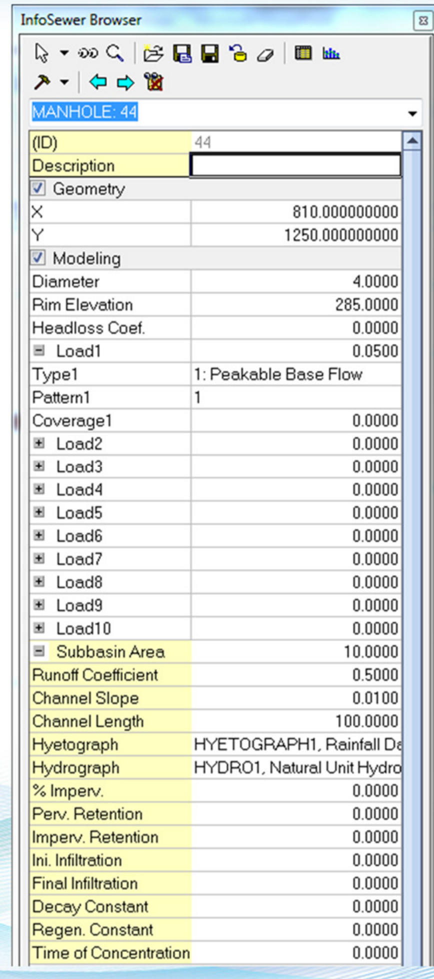
Pipes - Links conveying flow from one node to another. They can either be gravity mains or force mains.

Pumps - Unidirectional links adding energy to sewage for the purpose of discharging the water out of a localized low spot.

Manholes

- Manholes – Normal, Chamber, Outlet
- Represent flow into the system (out of the system for an outlet)
- Placed where links intersect
- Multiple loadings and patters
- Serve as storage devices during surcharge conditions
- Divide pumps and force mains (chamber)





Student Notes:

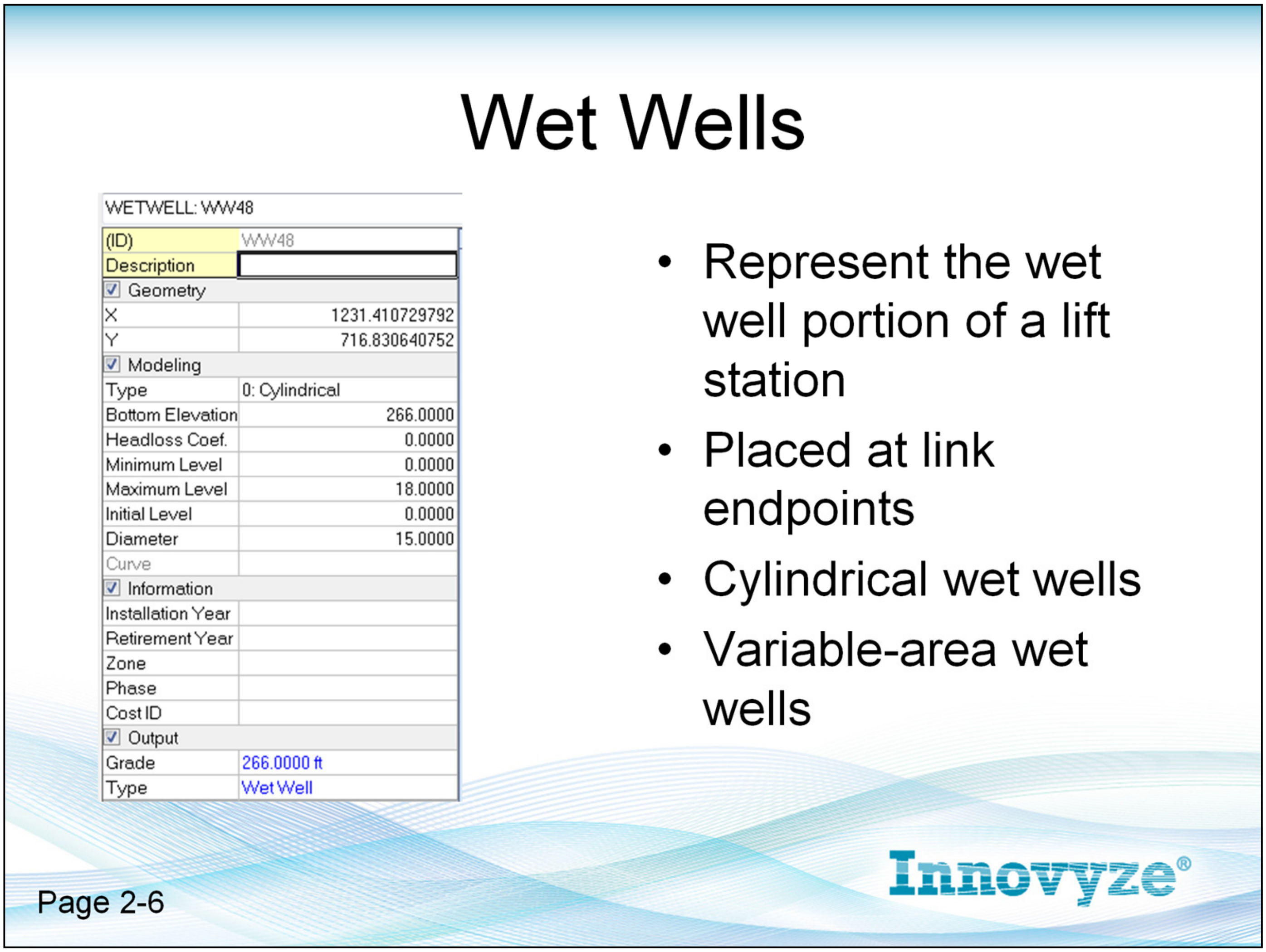
Normal manholes serve as storage devices during surcharged conditions of an EPS simulation. Chamber manholes always divide pumps and force mains. At least one outlet manhole is required in a collection system.

Multiple Loadings and Loading Patterns

You may assign up to ten individual loadings and ten related patterns for each load. For example, the first load (and related pattern) could represent residential loading at the node, the second could represent commercial loading, and so on. When running a simulation, InfoSewer will combine all of the loadings associated with a manhole node in an additive fashion (at each timestep in an extended period simulation) to accurately model the total loading assigned at each manhole in the system.

Extra Loadings

In addition to the ten base loadings, a loading manhole can also be assigned unlimited additional loads and patterns. These extra loads work in a similar fashion as the base loads as they are combined in additive fashion prior to the simulation being run. Extra loads are treated as a separate data set, but are analyzed in the same manner as base loadings.



Student Notes:

Wet wells represent those facilities in the collection system that serve as points of detention for a sewage lift station. Wet wells Storage nodes are those where a free water surface exists and the hydraulic head is equal to the free water surface. The following storage node representations are available:

Cylindrical wet wells

By default, wet wells are assumed to be cylindrical in shape. To define a cylindrical wet well, merely enter the bottom elevation (above datum), a minimum and maximum water level, an initial water level, and the diameter of the wet well.

Variable-area wet wells

By default, InfoSewer assumes all wet wells are cylindrical in shape. Alternately, the user can specify variable cross-sectional area wet wells by assigning a volume vs. depth *curve* that defines the changes of water volume in the wet well at different water depths.

Pipes (Gravity or Force)

PIPE: 28

(ID)	28
Description	
☑ Geometry	Reverse
Start Node	25
End Node	29
☑ Modeling	
From Invert	287.0000
To Invert	286.0000
Length	280.0000
Diameter	12.0000
Coefficient	0.0140
Parallel	0
☑ Information	
Type	0: Gravity Main
Installation Year	
Retirement Year	
Zone	
Phase	
Material	
Lining	
COST_ID	

- Convey water between nodes
- May be drawn in parallel
- Direction is critical
- May cross without connecting
- Must have manhole, wet well, or outlet at its end points
- Coefficient for force mains is expressed as Hazen-Williams C Factor

Student Notes:

Pipes are links that convey water from one node to another. Sewage flows only in the downstream direction of the pipes. Flow direction within the pipe is determined by InfoSewer as the direction for which the pipe was digitized.

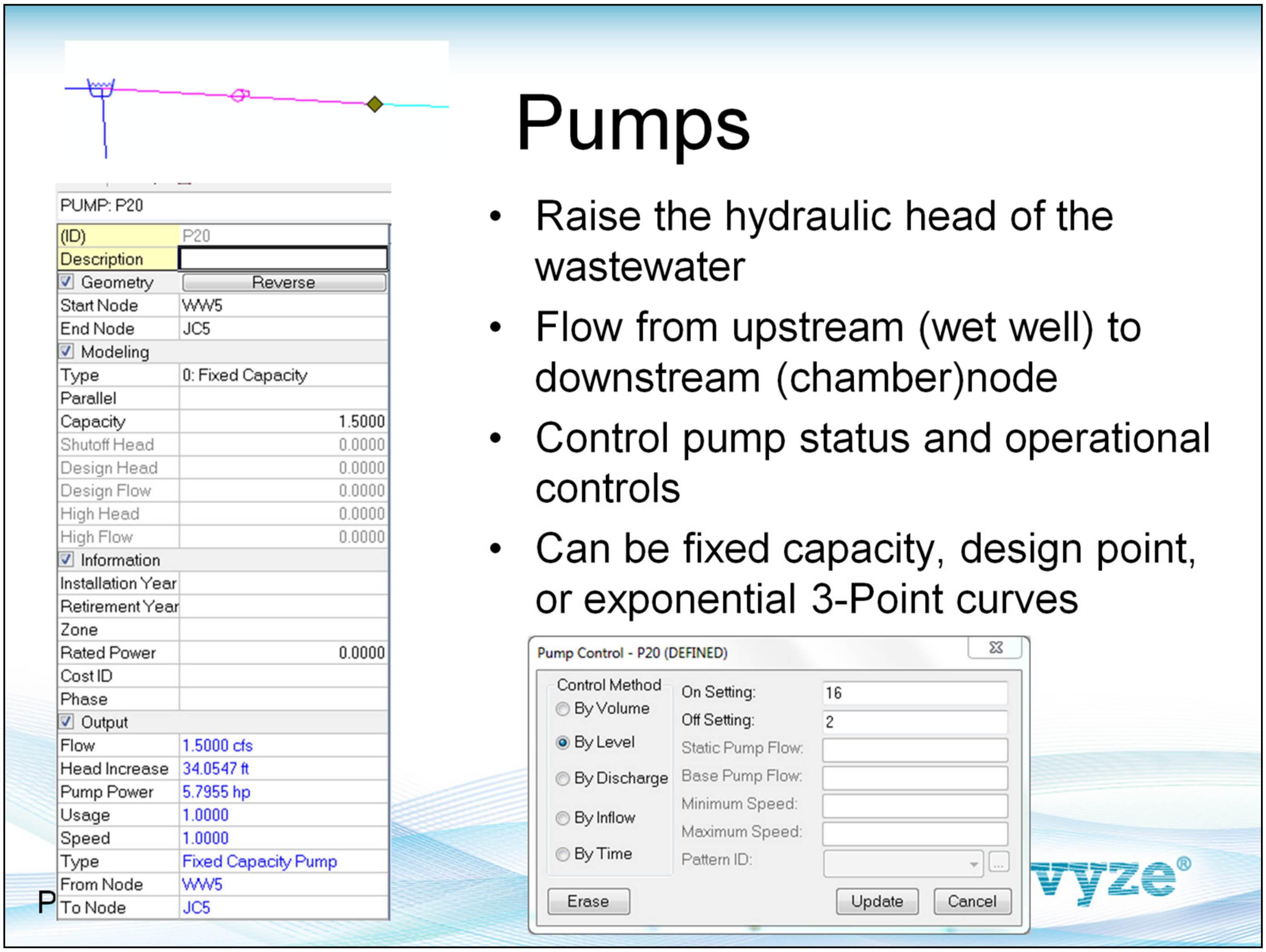
The “UPSTREAM” (“FROM”) node is defined as the first coordinate entered during pipe creation. Likewise, the “DOWNSTREAM” (“TO”) node is defined as the last coordinate entered during pipe creation.

Pipes may have any number of intermediate shape-defining points (vertices) between the upstream and downstream nodes. InfoSewer can automatically calculate the length of curvilinear pipes.

Any pipe digitized leaving a chamber manhole is automatically converted into a force main. Roughness factors for force mains are expressed as Hazen-Williams roughness coefficients.

Parallel pipes – Parallel pipes (two or more pipes with the same from and to nodes) may be entered in InfoSewer.

Non-interconnecting pipes – Pipes may cross graphically without connecting. Digitize pipes crossing one another by not connecting them to a manhole node.



Student Notes:

Pumps are links that raise the hydraulic head of sewage as it moves from a wet well to a chamber manhole. Flow in pumps is unidirectional, and always flow from the “UPSTREAM” node to the “DOWNSTREAM” node. Pumps may have any number of intermediate shape-defining points (vertices) between the upstream and downstream nodes. InfoSewer does not store length as an attribute of pumps. The following list represents the types of pumps available in InfoSewer:

Fixed Capacity – The pump curve is unknown and a constant capacity output is assumed.

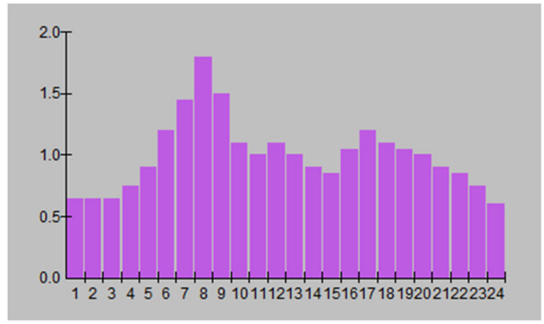
Design-point – A standard pump curve with a single operating point for the known head. The remainder of the curve is calculated by InfoSewer.

Exponential 3-Point Curve – A three-point pump curve with an optional extended range flow; some pumps exhibit a different type of characteristic curve beyond their normal flow range.

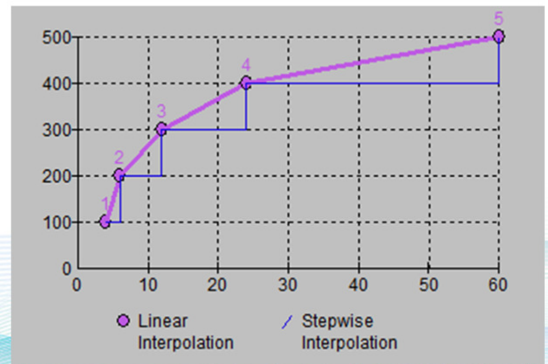
NOTE: Pumps can only be digitized between a wet well and a chamber manhole.

Modeling Parameters

- Hydraulic Analysis
  - Duration and timesteps
  - Time-Series Patterns



- Curves



Page 2-9

Student Notes:

Duration and timesteps – Duration and timesteps may be defined in any time units including days, hours, minutes, and seconds. They are used to define the duration of the simulation, at what interval network hydraulics are calculated and when reporting will take place during the simulation. InfoSewer can model both steady-state (SS) and extended period simulations (EPS).

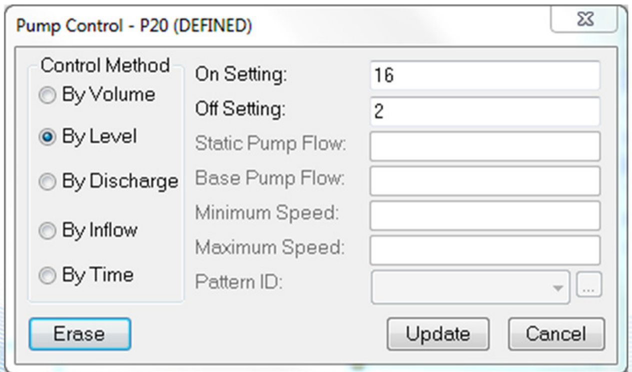
Time-series patterns – Patterns represent variations in model inputs over time. Patterns can represent time-based variations in the following:

Loading for manholes – Each pattern value represents a multiplier against which baseline loadings are applied. Up to ten patterns can be used to define water usage at manholes (one for each base load) and an additional unlimited patterns can be applied as an extra load set.

Curves – Curves represent non-temporal relationships including depth vs. volume relationships in variable-area wet wells, flow relationships at split flow manholes, head vs. flow relationships for user-defined pump curves, cost curves for replacement and parallel pipes and d/D criteria for existing and future pipes.

Modeling Parameters

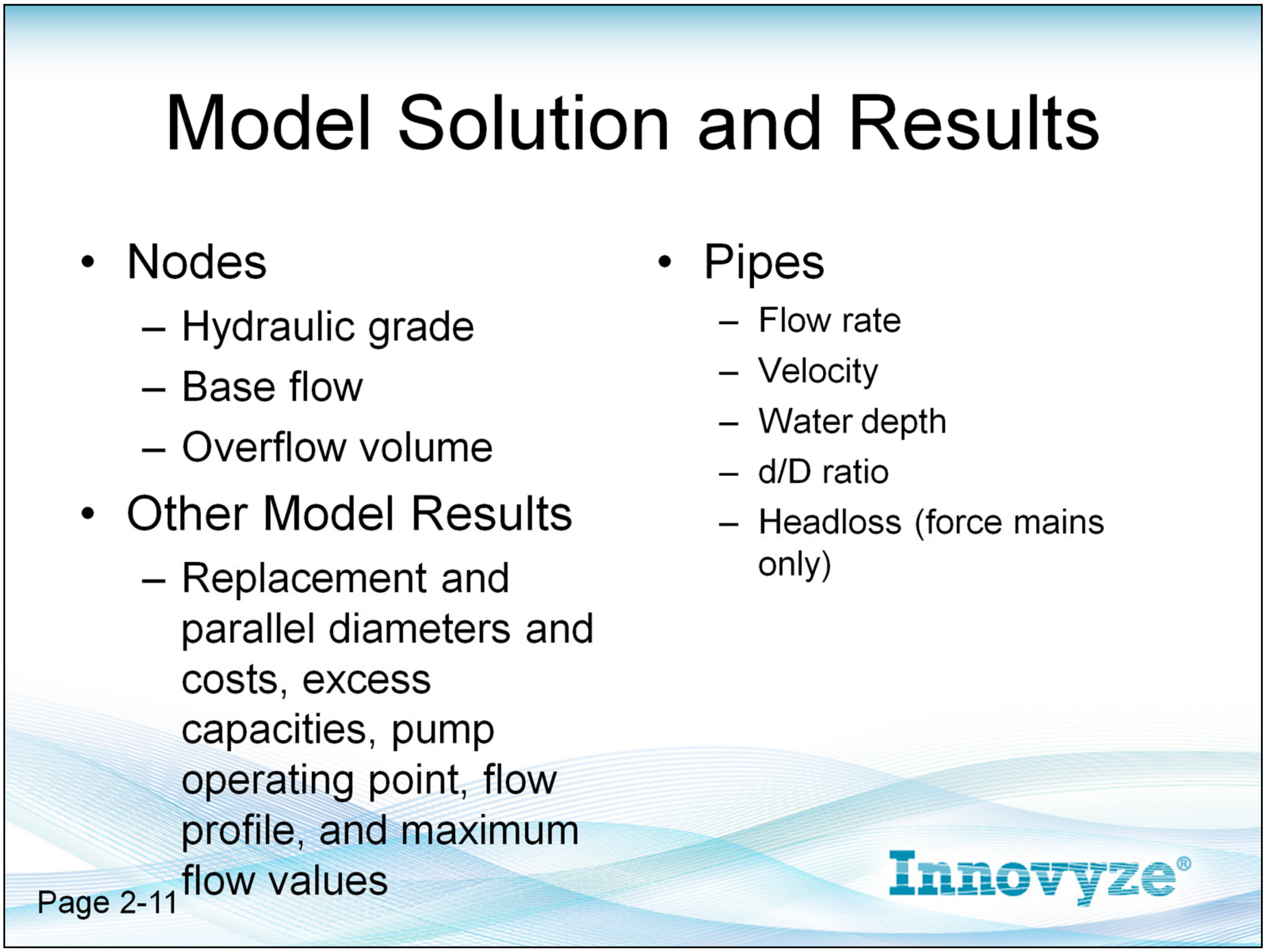
- Hydraulic Analysis
  – Initial operating status
  – Operational controls



Student Notes:

Initial operating status – Initial operating status defines whether pumps are open or closed at the beginning of a simulation. By default, pumps are assumed to be on at the beginning of a simulation.

Operational controls – You can change pump status (open or closed) and speed ratio during a simulation by specifying conditions that are tested at each hydraulic timestep during an extended period simulation.



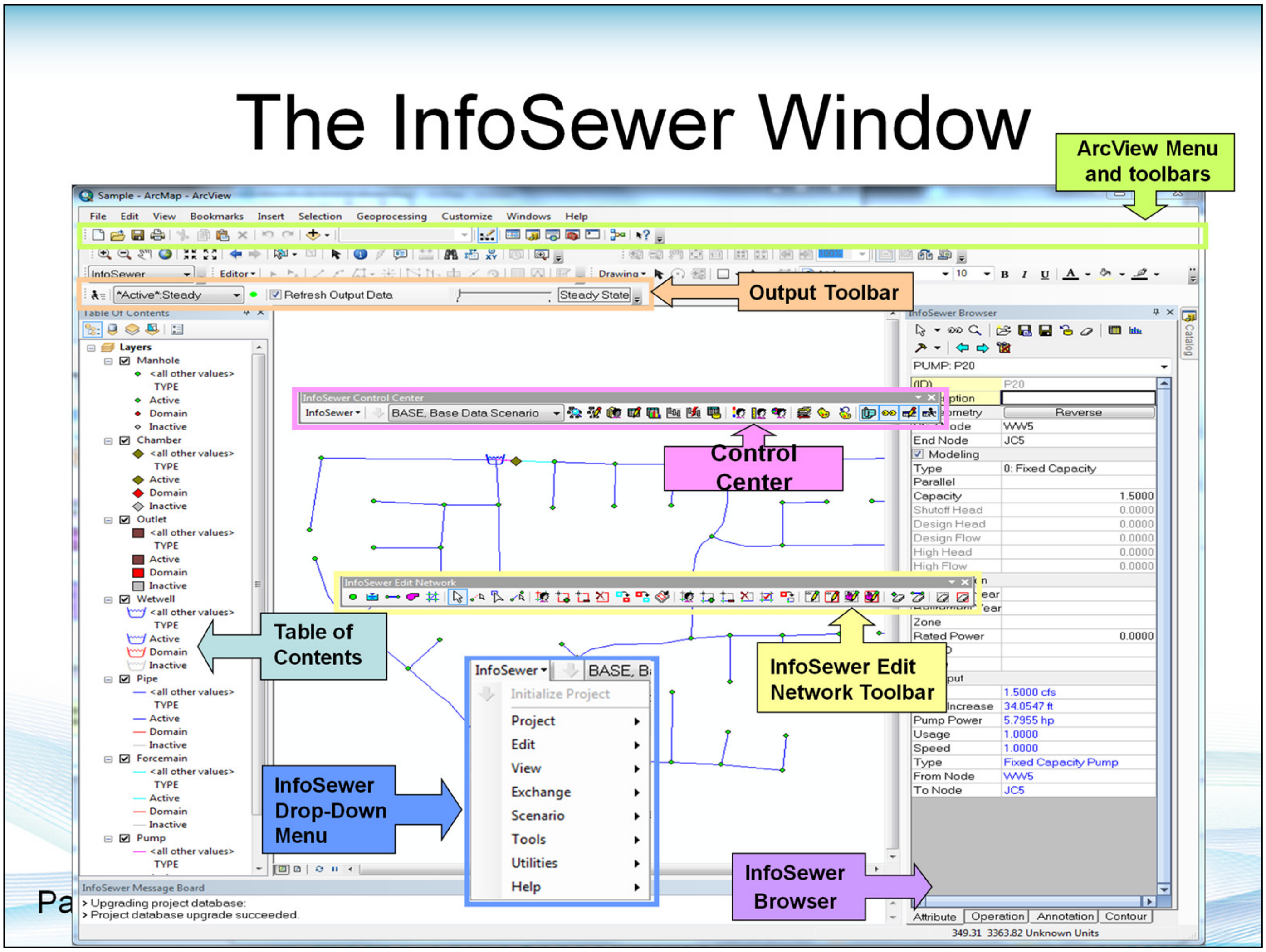
Student Notes:

The mathematical solution for a model is stored in a binary format and can be saved by the user for later review and analysis. This file can also be saved to any database, spreadsheet, or text format for use in other software packages.

Model results for nodes:

- **Hydraulic grade** - Hydraulic grade (HGL) is calculated for each network node.
- **Base flow** – Base flow is calculated by multiplying the base loading by the loading pattern at a specific time interval.
- **Overload volume** – The amount of water that is being held at the manhole during a surcharged condition.
- **Model results for Pipes**
- **Flow rate** – The flow rate is calculated for each link at a specific time interval.
- **Velocity** – The velocity is calculated for each link at a specific time interval.
- **Water depth** – The water depth is calculated for each link at a specific time interval.
- **d/D** – The depth to diameter ratio is calculated for each link.
- **Headloss (Force mains only)** - Average headloss is calculated for each link.

Additional analytical capabilities of InfoSewer include calculating excess capacity, design flow properties and evaluating replacement facilities during a steady state design simulation.



Student Notes:

InfoSewer Menu's – Contains all InfoSewer menus. Individual menus are organized functionally within this main menu.

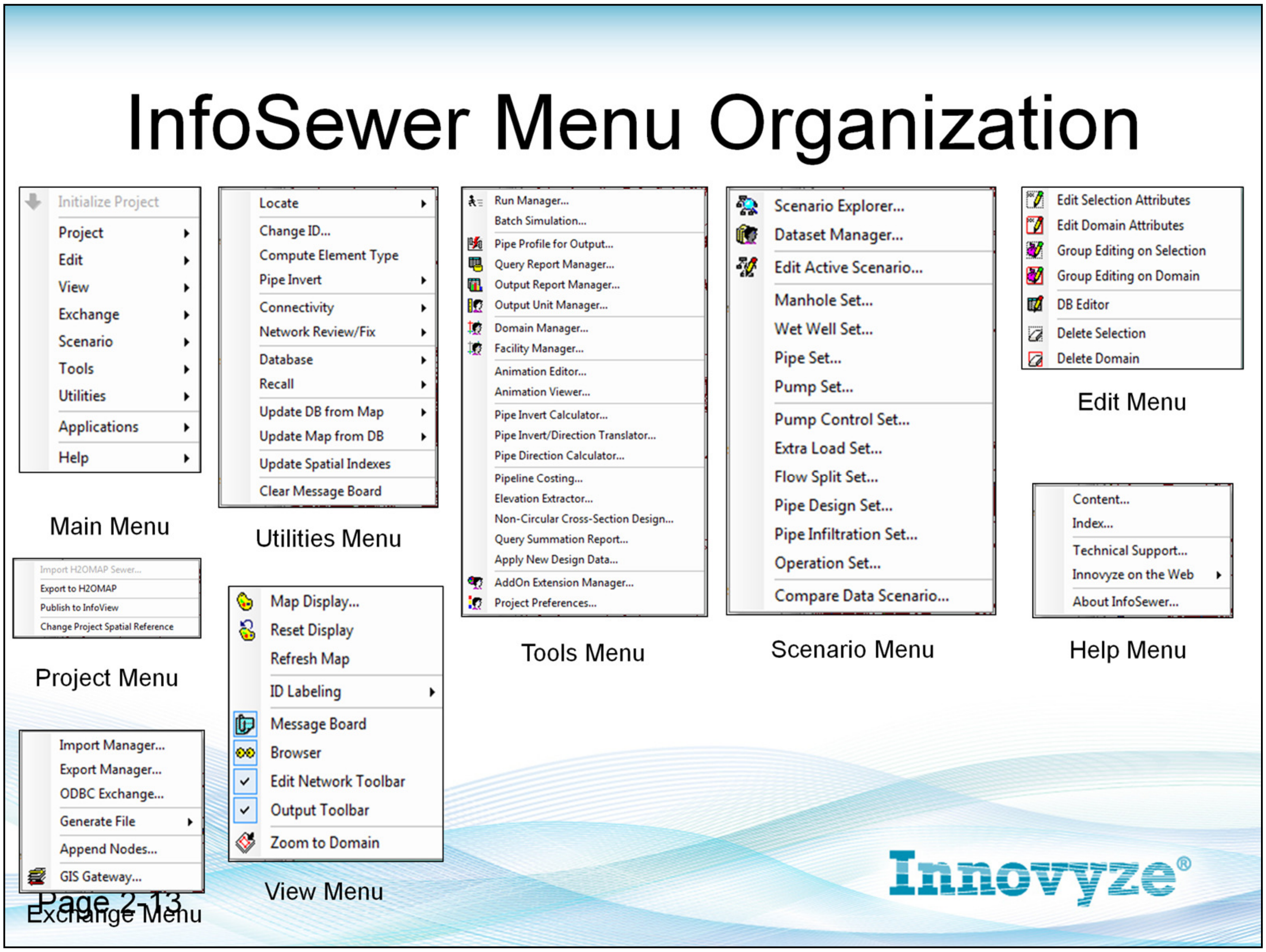
Table of Contents – Controls the display of all layers in the map document. Refer to ESRI documentation for more details.

Attribute Browser - The Attribute Browser allows the user to view and edit the database information of a selected link or node facility. The attribute browser provides access to operational features as well as annotation layers, contour layers.

InfoSewer Control Center - The Control Center provides access to the most commonly used features in InfoSewer including the Scenario Manager, Dataset Manager, DB Editor, and Output Report Manager.

InfoSewer Edit Network Toolbar - The Edit Network Toolbar contains all commands to create, edit, or delete network components. The domain manager and facility manager are also found in this toolbar.

InfoSewer Output Toolbar - The Output Toolbar controls the output data source and the time / date of the currently displayed output. Access to the run manager is provided.



Student Notes:

Project – Use to open existing projects, set the project's spatial reference, and define calibration data.

Edit – Use to edit InfoSewer model input data including attributes for multiple components in a single operation and attributes in a spreadsheet (DB) editor.

View – Use to change the map display and add/remove InfoSewer toolbars.

Exchange – Use to import and export data including comma-delimited text, shapefiles, generate files, and ESRI feature classes.

Scenario – Use to define and edit scenarios and their components (facility sets, data sets, and simulation options).

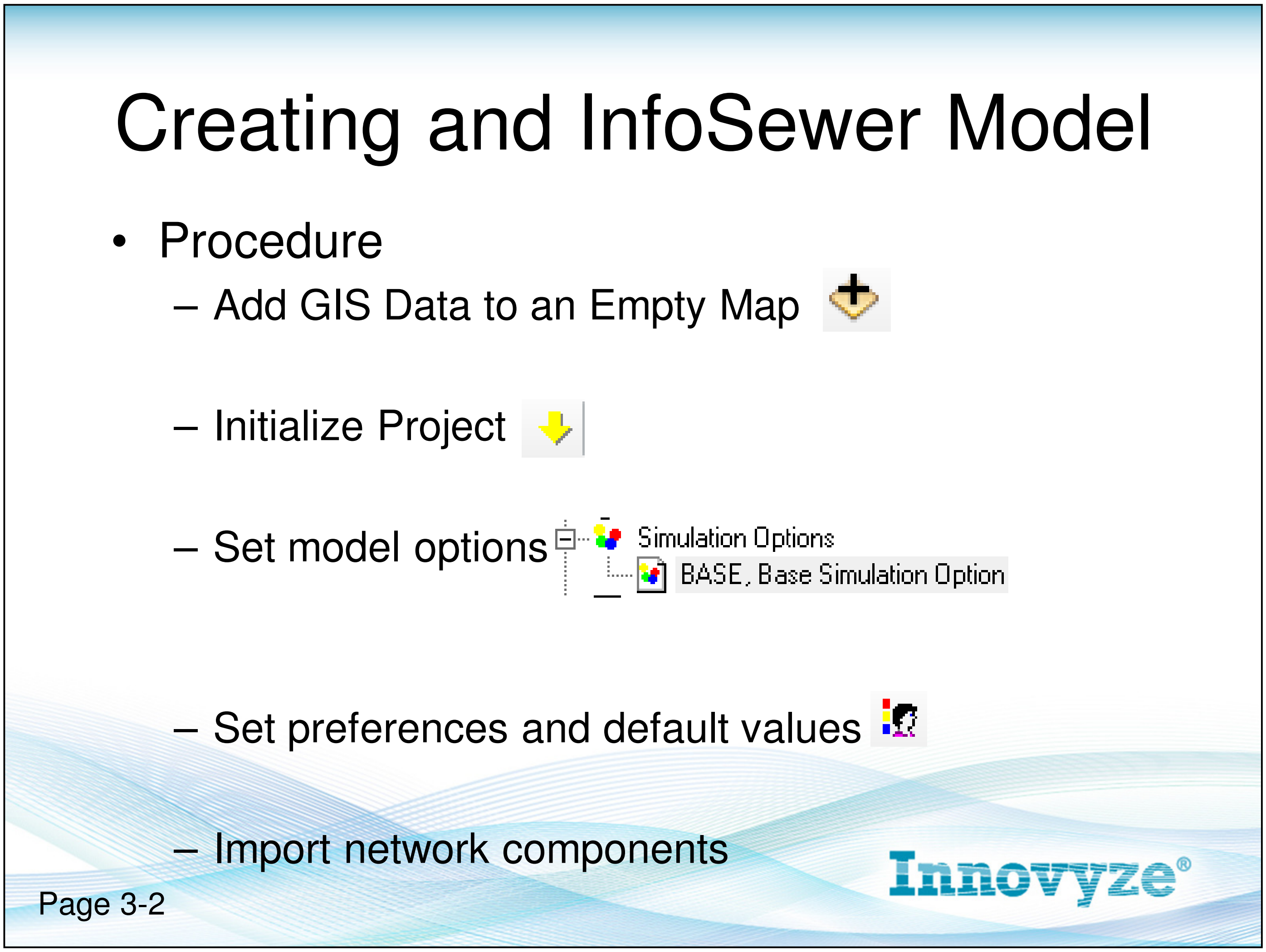
Tools – Use to run a simulation, view reports/graphs with the Output Report Manager, create animations, and open the domain and facility managers.

Utilities - Contains specialized editing tools including Locate network components, Change Element ID's, Review network connectivity and fix features without topology, Perform database and graphical transfers.

Chapter 3

Creating an InfoSewer Model

Student Notes:



Student Notes:

Add GIS Data to an Empty map - Add datasets to be imported as well as any desired background layers.

Initialize the InfoSewer Project- The databases and commands used by InfoSewer are not accessible until the InfoSewer project has been initialized. This is true for both existing and new projects.

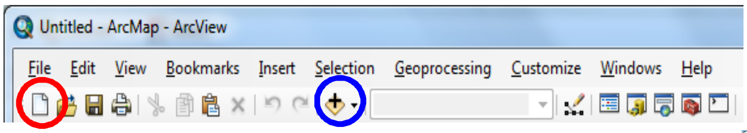
Set model options - Set global options including units of measurement, simulation duration, and the desired method for infiltration, routing and process modeling. These options should be set before creating or importing network components.

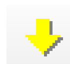
Set preferences and defaults - Set InfoSewer system preferences for symbology, ID labeling, and default values for new network features.

Import network components - Import network components using one or more InfoSewer's import methods.

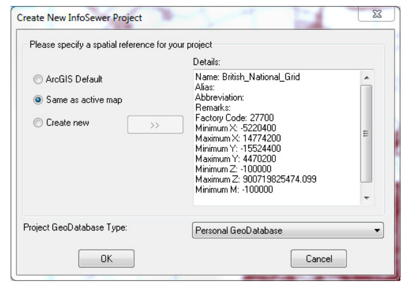
Creating an InfoSewer Project

- Create a new ESRI ArcMap Document file then Add Data



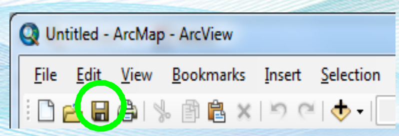
  – Initialize the Project 

  – Set the spatial reference when prompted



  – The modeling database and feature classes are created

- Save the project



Innovyze®

Student Notes:

Create a new ArcMap file - Start ArcMap and choose *A New Empty Map* on the ArcMap window.

Initialize the InfoSewer Project - Press the Initialize icon. This creates the databases necessary to store model information, and establishes links between the ArcMap Document file and the InfoSewer database tables. In addition, feature classes are created for each element in InfoSewer. The spatial reference must be set, i.e. the coordinate system and spatial extent must be specified.

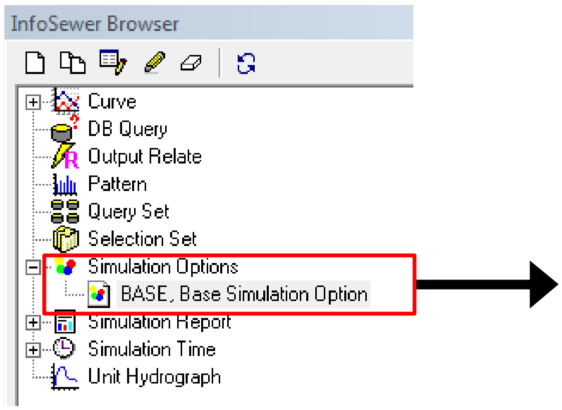
Save the project -Save the project to the file location with any desired name. InfoSewer Project names are Windows compliant. You may assign a name of any length. The name may contain embedded blanks.

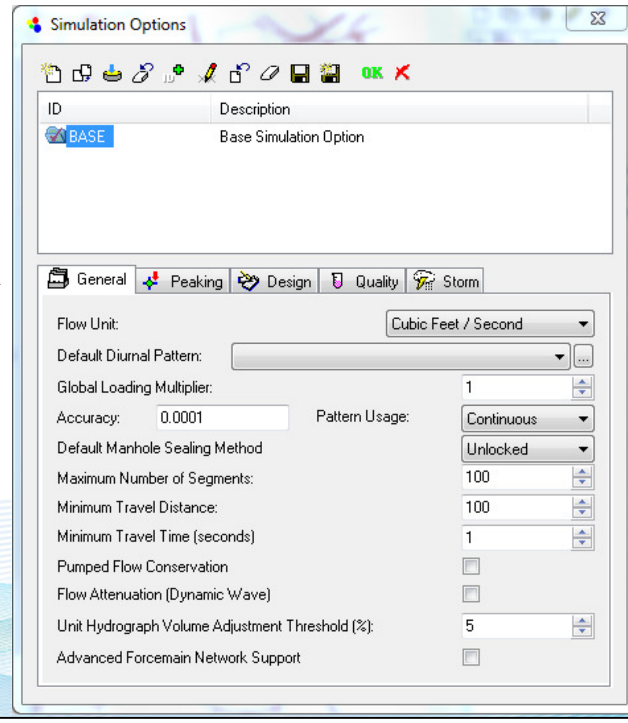
NOTE: Features cannot be created outside the spatial reference. For more information on spatial references, please refer to the ArcMap Help file.

NOTE: InfoSewer uses a temporary folder to store all changes made while working in ArcMap \ InfoSewer. The temporary files replace the permanent files when the ArcMap document is saved.

Setting Basic Model Options

- Open the BASE Model Options





- General Model Options

Student Notes:

Set units of measurement for flow – (gpm, cfs, mdg, imgd, afd, lps, lpm, mld, cmh, or cmd).

Set minimum travel distance – To determine the minimum travel distance for an EPS model run. The value provided here is utilized in determining routed flows between manholes and determining flow attenuation.

Set accuracy – Sets the convergence criterion that is used to signal the analysis engine that a solution has been found to the nonlinear equations that govern network hydraulics. A value of 0.001 is recommended.

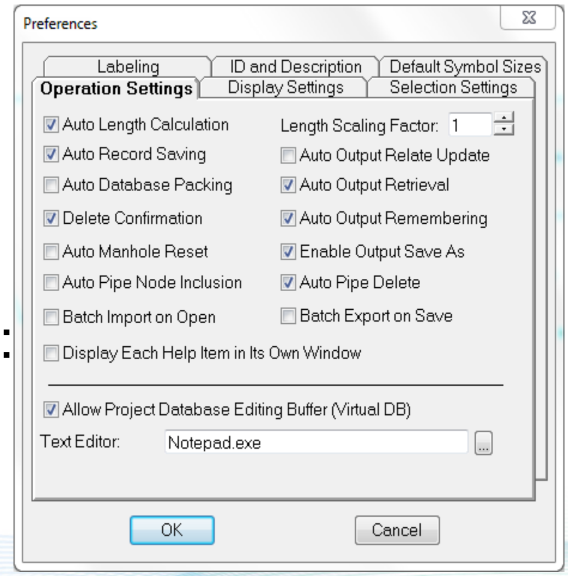
Set loading parameters – Global Loading Multiplier - Used to assign a multiplier in which global loadings to be multiplied. For example, a MAXDAY model scenario may have a global loading multiplier of 2.5.

Set peaking equation parameters (“Peaking” tab) – Choose the “Peaking” tab of the “Simulation Options” dialog box. Peaking equations only apply to steady state analysis and steady state design simulations and do not apply to EPS simulations.

Specify peaking factors according to the Federov (base load) or Harman Babbit (population coverage) peaking equations. Which equation is employed will depend upon the peaking “Type” associated with the manhole (see InfoSewer Browser – Attribute tab).

Preferences and Default Values

- Select “Preferences” from the “InfoSewer Tools” menu
  - – Auto-length calculation
  - – Delete confirmation
  - – Auto-database packing
  - – Auto-record saving
- Setting default symbol sizes:
  - – Default Symbol Sizes
- Setting new identifiers
  - – ID and Description



Student Notes:

Auto Length Calculation - When on, InfoSewer calculates the length of new links in map units. **NOTE:** Be mindful of the state of this setting (ON or OFF) while you are working.

Delete Confirmation - When ON, you must verify the deletion of network components.

Auto-Database Packing - When OFF, you are able to recall previously deleted components. When ON, you are unable to recall deleted components, but you are immediately able to re-use any previously deleted IDs.

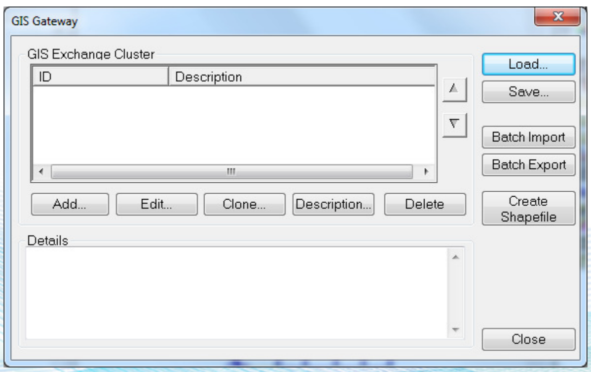
Auto-Record Saving - Controls if modifications to records on various dialog boxes (such as the Edit Patterns, Edit Curves, Edit Data Set, Edit Manhole, etc.) are immediately saved when you pick another record for editing on the same dialog box.

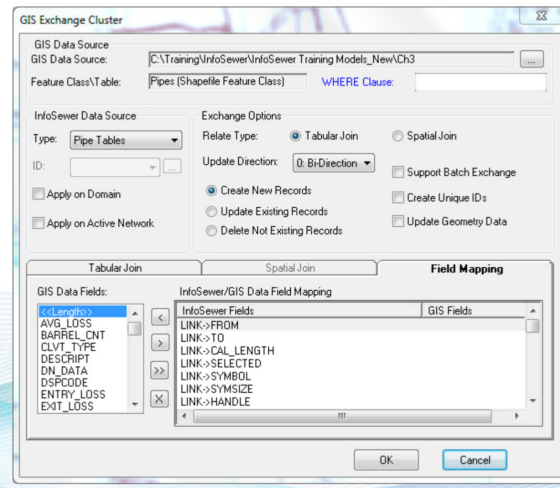
Set symbol sizes - Choose the “ Default Symbol Sizes” tab to specify the size of symbols representing new network components.

Set new component identifiers - Choose the “ID and Description” tab to specify the ID value assigned (suggested) for newly added network components. You may set an alpha-numeric prefix and specify the increment between ID values for new components.

InfoSewer – Importing Features from GIS – GIS Gateway

- Exchange Information with GIS Data Sources
  - Shapefiles
  - Feature Classes
  - Enterprise Geodatabases (via SDE)
- Refresh Map after Import





Student Notes:

The GIS gateway is designed for seamless and manual data transfer between InfoSewer and GIS data sources. The data may be transferred through unique ID's (tabular join) or by spatial proximity (spatial join). Both methods require field mapping. GIS exchange clusters hold all options for data transfer. Data exchange can be limited to the active network or a domain. Clusters may be saved and subsequently used with the push of one button.

Features of the GIS Gateway are described below:

Load - Load information into InfoSewer per the specified cluster

Save- Save data to a GIS data source per the specified cluster

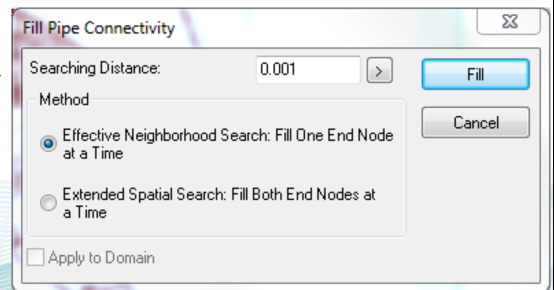
Update Direction - Specifies if the cluster will be used to load, save, or both

Create Unique ID's - Create ID's when data loading - GIS ID field not needed

Where Clause - Write a SQL statement to restrict data exchange to records whose attributes satisfy the SQL statement

Automated Start/End Node Assignment

- Searches for junctions near pipe end points
- The closest junction within the specified tolerance is assigned as the start or end node
- If importing from a geometric network, use the XY tolerance for the searching distance
- Use prior to Network Review
- Apply to Domain if necessary

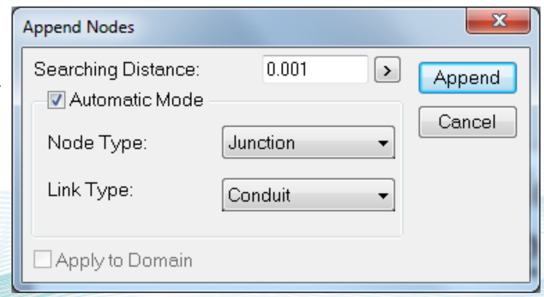


Innovyze®

Student Notes:

Automated Node Creation

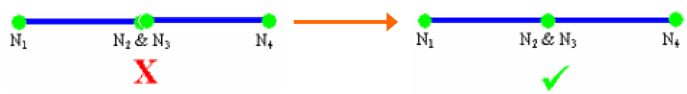
- Creates junctions at all pipe end points
- Only 1 junction is created for multiple pipe end points within the specified tolerance
- If importing from a geometric network, use the XY tolerance for the searching distance
- Use prior to Network Review
- Apply to Domain if necessary



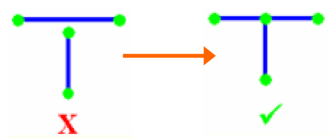
Student Notes:

Network Review

- Examine the geometry of imported data sources
  - Locate / Fix Nodes in close Proximity



  - Orphan Nodes / Links
    - Use Domain Manager
  - Trace Network – Utilities Menu
  - Locate / Fix Conduit-Split Candidates



  - Locate Parallel Conduits
    - Consider Re-digitizing
  - Locate / Fix Crossing / Intersecting Conduits

Student Notes:

Locate / Fix Nodes in close Proximity – Find nodes with a specified tolerance. Merge all nodes or a domain that are within the tolerance.

Locate / Fix Conduit-Split Candidates – Find nodes near pipes that are not connected to the pipe. Split the pipe and connect each to the node if desired.

Locate / Fix Crossing / Intersecting Conduits – Find all pipes that cross but are not connected. Add a node at the intersection as needed.

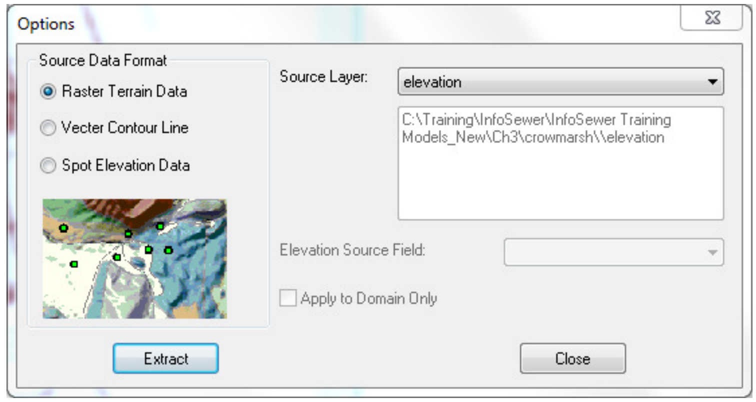
Orphan Nodes / Links – Find disconnected nodes or pipes missing a connection.

Trace Network – Highlights all elements that are connected to the network of a user selected feature.

Locate Parallel Conduits – Finds all pipes with the same to and from nodes

Elevation Extraction

- Use Point, Contour, or Raster elevation data
  - Rasters are fastest
- Calculate 3D pipe lengths if desired
- Apply to Domain Only
- When finished, block edit to simulate buried depth if desired



Student Notes:

Chapter 4

Working in the InfoSewer Interface

Student Notes:

Digitizing Features

- Edit Network Toolbar – Creates Network Components
  - Manhole
    - Normal
    - Chamber
    - Outlet
  - Wet well
  - Pump
  - Pipe
    - Gravity
    - Force
  - Digitize Network
    - Creates Links and Nodes continuously

Student Notes:

To create network components, use one of the commands from the “Edit Network” Toolbar.

Creating individual components

To create individual network components, choose the “Add/Insert Manhole”, “Add Wet Well”, “Add Pipe”, or “Add Pump” icons from the “Edit Network” toolbar

Shortcut digitizing options

To digitize multiple network components in a single operation, choose the “Digitize Network” icon. This command is used to add nodes and links in series.

NOTE: The “Digitize Network” command should be used as the primary mode of graphic data entry in InfoSewer when adding large numbers of network components. This creation technique will not prompt you for necessary data input, but will allow you to assign graphical locations to facilities and populate the database tables in a spreadsheet-like fashion at a later point.

Create Network Components

- Hints:
  - Pipes and pumps always begin and end at a manhole or wet well
  - Pumps can only be connected to wet wells and chamber manholes
  - Pipe and pump direction is essential
  - Enter components as accurately as possible
  - Downstream of a chamber force main is required.

Student Notes:

Network connectivity – InfoSewer forces links to be snapped to existing nodes; you must digitize the node(s) first, then create the links connected to those nodes.

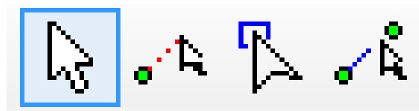
Link orientation – Be sure to create all links (pipes, force mains, pumps) in the direction of flow. Adverse gravity pipes can exist in the system.

Accurate digitizing – To ensure an accurate model representation, enter the correct link length in the InfoSewer database tables if you are not using a topologically correct base map.

Although InfoSewer does not require an accurate graphic representation of the network, you should digitize links as accurately as necessary for plotting and presentation purposes. Having a good base map and using the Auto Length Calculation simplifies model creation.

Modifying Features

- Move Nodes
- Redraw Links
- Edit Link Vertex



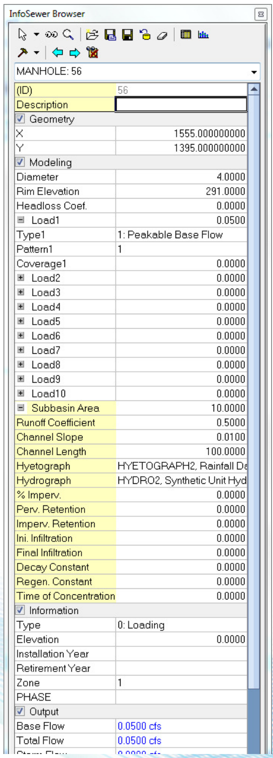
Student Notes:

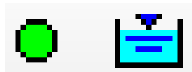
Moving Nodes - Drag the node with the cursor to the desired location. All links attached to that node are automatically re-shaped (stretched or shortened) to maintain connectivity with the node. Pipe lengths are automatically recalculated if the auto-length calculation option is enabled.

Redraw Link - The user is prompted to re-digitize the link. Select the desired link then redraw it. The length of the link is automatically updated if the auto-length calculation option is enabled.

Edit Link Vertex - Used to change the location of intermediate vertices along a link. Select the desired link and intermediate vertices will appear as boxes. Click on the desired vertex and it turns blue. Drag the vertex to the desired location. The length of the link is automatically updated as long as the auto-length calculation option is enabled.

The Attribute Tab - Nodes



- ID 
  - A unique value identifying the node
- Geometry
  - X and Y coordinates
- Modeling Data
  - Hydraulic Parameters
  - Load, type, patter, coverage (normal)
  - Type and associated modeling data (wet well)
- Information
  - General information
  - Not used when running the modeling
  - Add attributes as necessary
- Output
  - Only available after running a simulation

Student Notes:

The Attribute tab of the InfoSewer Browser dialog box allows the user to view and edit the database information of a selected element. The dialog box contains most information related to nodes. Any of the records under the input section headers (Geometry, Modeling, and Information) are editable at any time unless "grayed" out.

Node ID – Contains the unique InfoSewer identification of the node and a user defined description of the node.

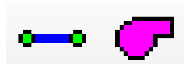
Geometry - The x-coordinate and y-coordinate of the node.

Modeling - Modeling attributes are used for network simulations. All fields in this section must be populated before running a simulation unless “grayed” out.

Information - Information tables are fully customizable. They contain useful information that is not necessary to carry out the simulation.

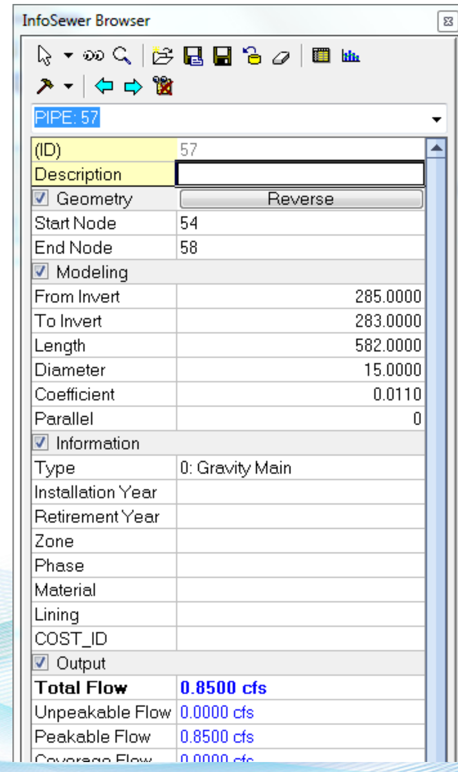
Output - Output tables contain simulation results commonly accessed through the output report manager and map display functionality. These tables are created during a simulation and will not appear before running a simulation.

The Attribute Tab - Links



- ID
  - A unique value identifying the link
- Geometry
  - Change link connectivity
- Modeling Data
  - Hydraulic Parameters
- Information
  - General information
  - Not used when running the modeling
  - Add attributes as necessary
- Output
  - Only available after running a simulation

Attribute Tab - Pipe



Student Notes:

Use the Attribute tab of the InfoSewer Browser dialog box to view and edit the database information of a selected pipe. The dialog box contains most information related to pipes. Any of the records under the input section headers (Geometry, Modeling, and Information) are editable at any time unless "grayed" out.

Link ID - Contains the unique InfoSewer identification of the pipe and a user defined description of the pipe.

Geometry - The upstream and downstream node of the pipe.

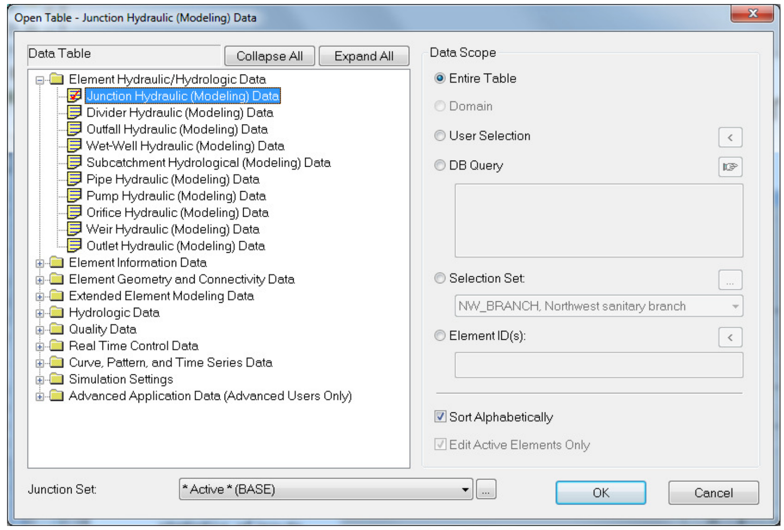
Modeling - Modeling attributes are used for network simulations. All fields in this section must be populated before running a simulation unless “grayed” out.

Information - Information tables are fully customizable. They contain useful information that is not necessary to carry out the simulation.

Output - Output tables contain simulation results commonly accessed through the output report manager and map display functionality (See Ch. 6). These tables are created during a simulation and will not appear before running a simulation.

Introduction to the Database Editor

- Opens the DB Editor
- DB Editor Uses:
  - Editing a selected set of features
  - Copy and paste into / from Excel
  - Viewing summary statistics of inputs



Student Notes:

The Database Editor provides access to database tables in the InfoSewer project. DB table are separated by feature or operation type, and are grouped in the following categories:

Element Hydraulic Data – All data contained in the modeling section of the attribute browser.

Element Information Data – All data contained in the information section of the attribute browser. This table can be customized from the DB Editor.

Element Geometry and Connectivity Data – All data contained in the geometry section of the attribute browser. Also includes transect tables.

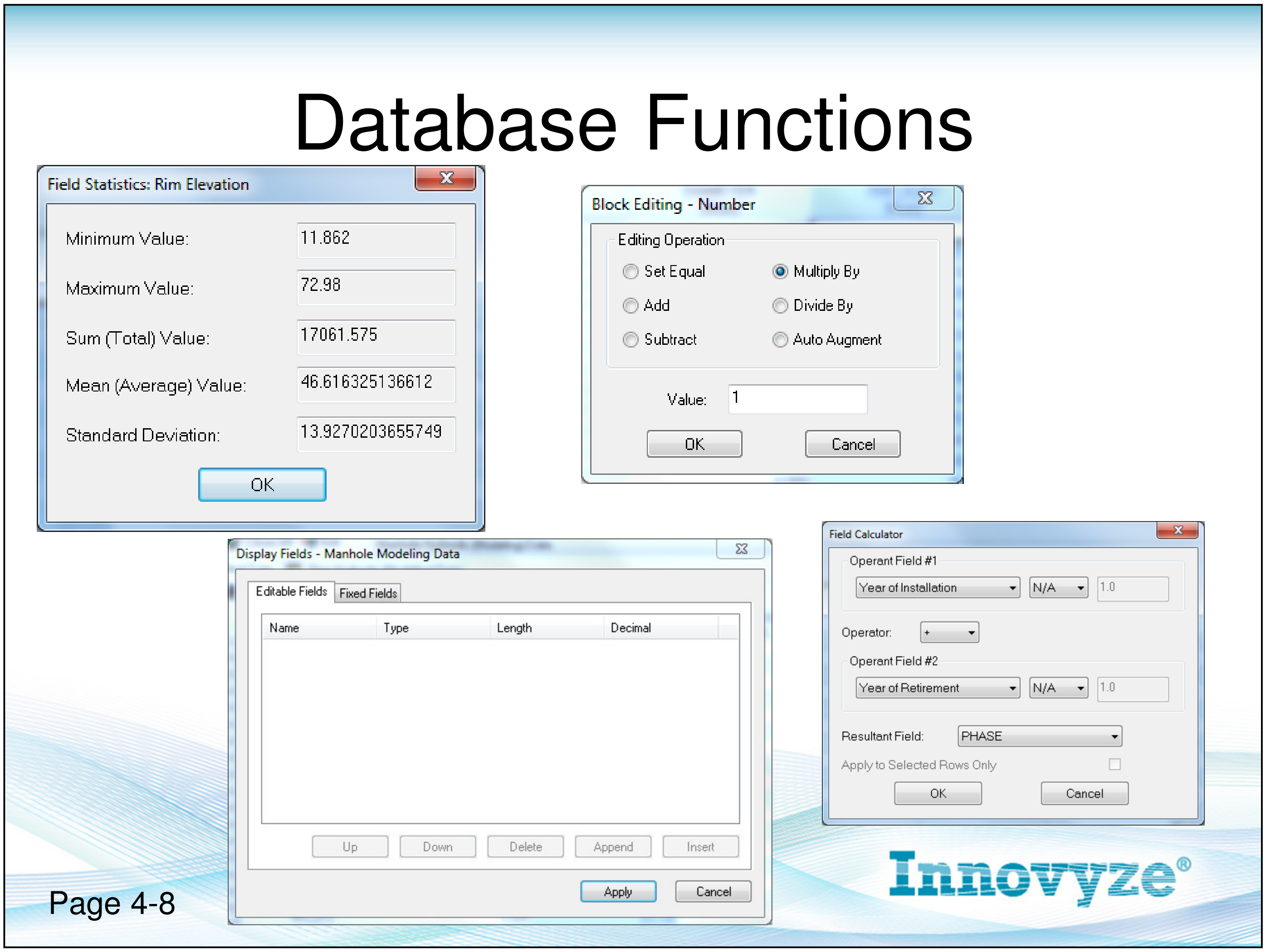
Extended Element Hydraulic Data – Tables containing node inflows (External, Dry Weather, and RDII).

Element Hydrologic Data – All data contained in the modeling section of the attribute browser.

Quality Data – Data representing land use, pollutants, buildup/washoff rates, and treatment.

Real Time Control Data – Tables containing initial status, simple controls, and real time controls.

Curve, Pattern, and Time Series Data – Tables containing all curves, patterns, and time series.



Student Notes:

When a table is opened via the DB Editor, many functions are available to the user. The most common features are described below:

Block Edit - Edit the selected cells with common mathematical operators.

Field Calculator - Calculate one field based on other fields.

Field Statistics - View summary statistics of the field such as mean, sum, and standard deviation.

Field Aliases and Descriptions - Look at field properties. This interface allows creation of fields in the information tables.

Save - Save all changes to the database.

Sort Ascending/Descending - Sort the selected column in ascending/descending order.

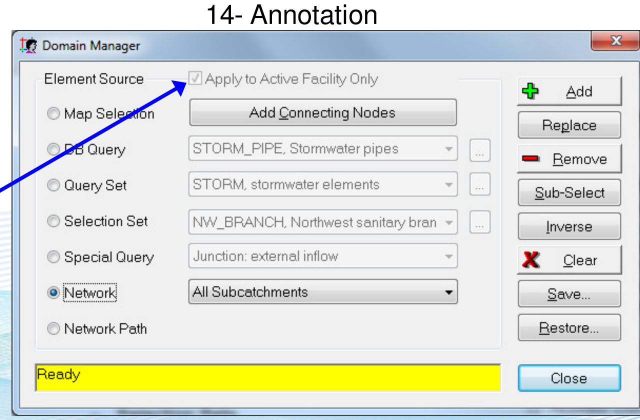
Find - Specified data in the current table.

Domain Management

- Domain – A group of features for data editing or result display
- Created with:
  - User Selection
  - Queries
  - Selection Sets
- May be constrained to the active facility

A list of tools which utilize the domain:

1- Elevation Extractor
2- GIS Gateway
3- Update DB from MAP
4- Update MAP from DB
5- Output Report Manager
6- Query Reports
7- DB Editor
9- Selection Set Creation \ Use
10- ArcMap Selection to Domain
11- Domain to ArcMap Selection
12- Activate Domain (Facility Manager)
13- Contours
14- Annotation

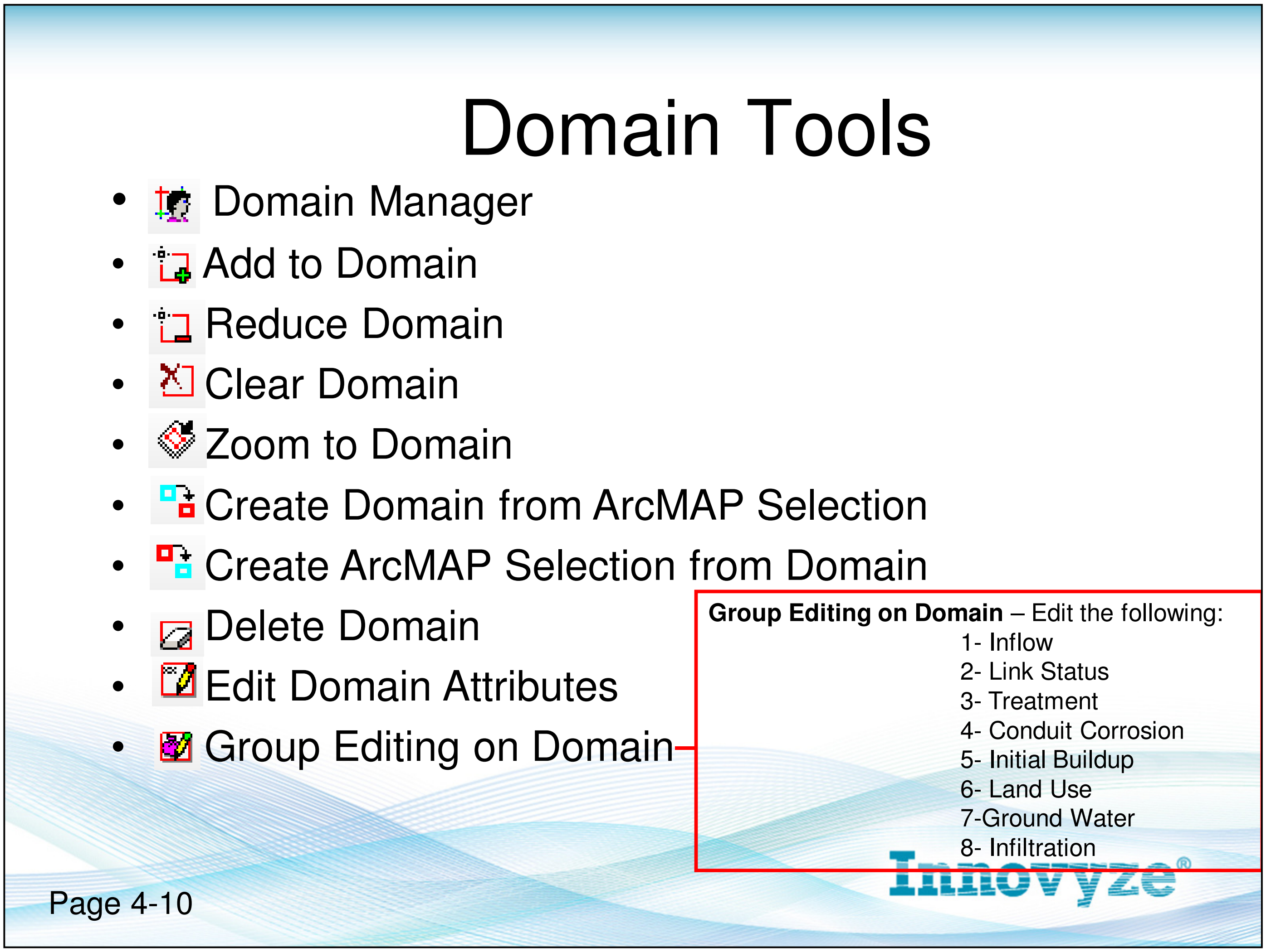


Student Notes:

A domain is a collection of network components that may be edited, queried, and reported. Domains are created by selection or queries. Predefined special queries and network components are also options. A domain **does not impact** on the model operation or simulation.

Important notes about domains:

- Reports and Graphs of output data can be generated for a domain.
- Input data for features currently in the domain can be edited.
- Data exchange can be restricted to a domain.
- A model simulation can **NOT** be restricted to elements in a domain.
- Domain Queries and Selections may be confined to active facilities.
- Network Path is a trace between 2 nodes



Student Notes:

Domain Manager – Interface for creating, enlarging, or reducing domains.

Add to Domain – Adds objects to the domain.

Reduce Domain – Removes objects from the domain.

Clear Domain – Removes all elements from the domain.

Zoom to Domain – Zooms to the extent of the domain.

Create Domain from ArcMAP Selection – Turns all features from an ArcMap selection into an InfoSewer domain.

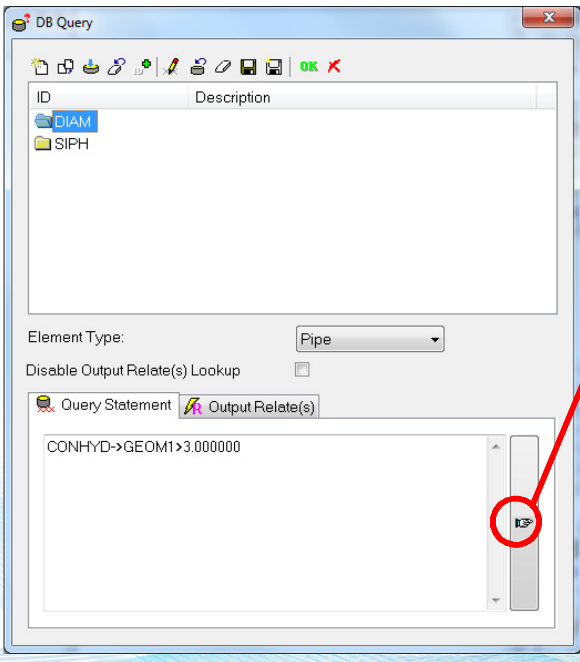
Create ArcMAP Selection from Domain – Turns all features from an InfoSewer domain into an ArcMap selection.

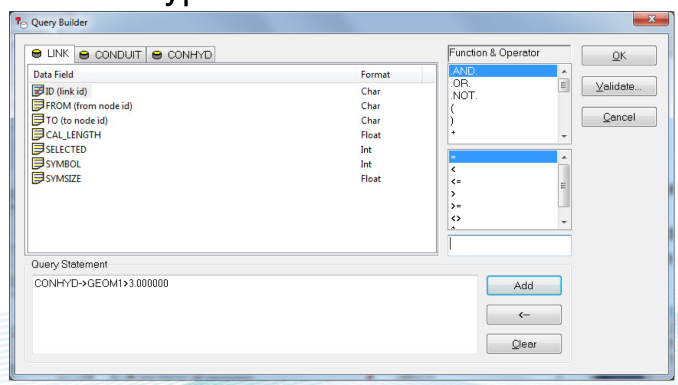
Delete Domain – Deletes all components in a domain.

Edit Domain Attributes – Edit any attribute contained in the information or modeling DB Tables.

Database Queries

- Search for specific data
  - Structured Query Language (SQL)
  - Searches one or more fields
  - Refers to a specific feature type





Innovyze®

Student Notes:

A query searches one or more fields within one or more database tables for specific information. The records (features) that match the query are entered into a database table, domain, or facility.

Queries in InfoSewer must follow SQL rules. Statements may be validated to ensure they comply with SQL rules. Validating queries also will tell the user if the query result is empty, i.e. no record matched the query statement. A few examples of SQL statements are shown below:

JUNCTION->ELEVATION>5200.000000 .AND.

JUNCTION->ELEVATION<5300.000000

Selects all junction with an elevation between 5200 AND 5300.

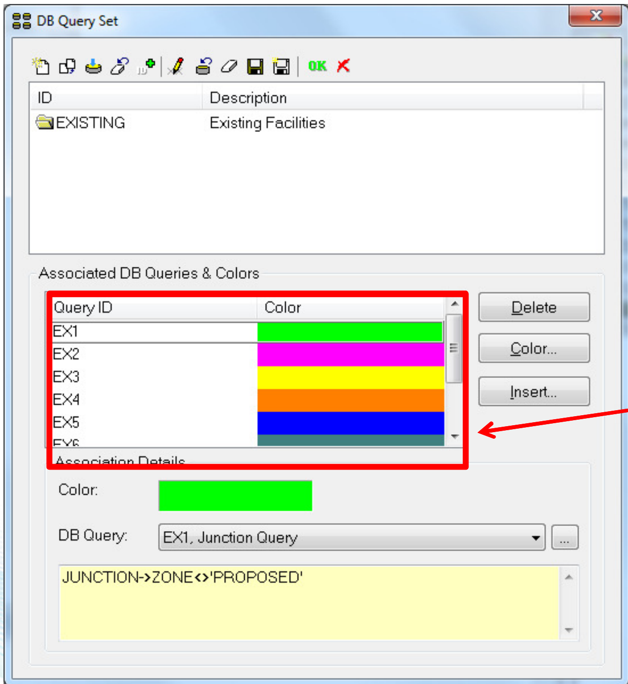
JCTHYD->MAX_DEPTH<5.000000 .NOT.

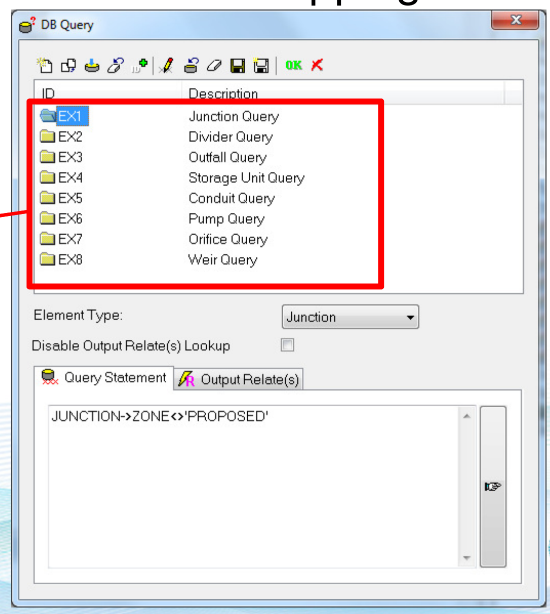
JCTHYD->MAX_DEPTH<3.000000

Selects all junction with a max depth less than 5 but NOT less than 3.

Query Sets

- A group of queries
- Query multiple feature types
- Useful with Facilities and Thematic Mapping





Student Notes:

Query sets are a collection of queries. They are useful for querying multiple network components for addition to a domain or facility. The benefit of query sets is they can select features added to the model after the query set was created if the correct fields of the new feature are populated.

Each component in a query set may be color coded uniquely to detail the records selected by the query set. This feature is especially useful for managing facilities in scenarios and color coding areas of a system.

Examples of query set functionality:

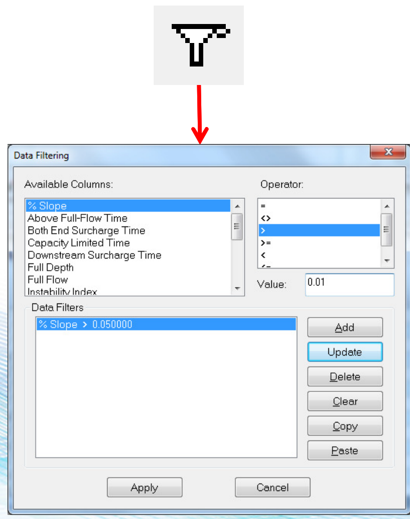
Select all network components in an isolated or critical area in the system with a zone attribute.

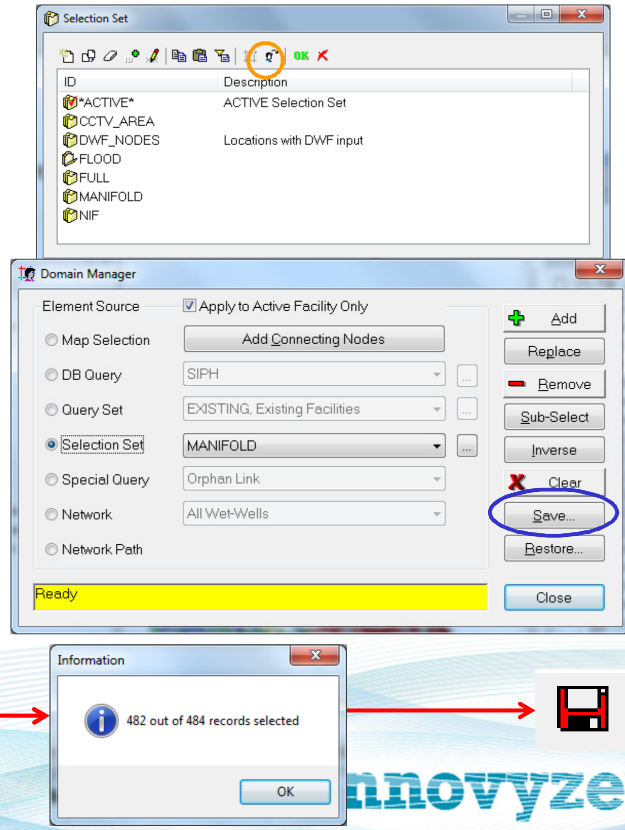
Track existing network components for the analysis of the current system while allowing new network additions to be added and analyzed as necessary. Create a user defined field to track this information.

Create customized reports that contain output data from critical elements.

Selection Sets

- Created with:
  - User Selection
  - Domain
  - Saved Filter





Student Notes:

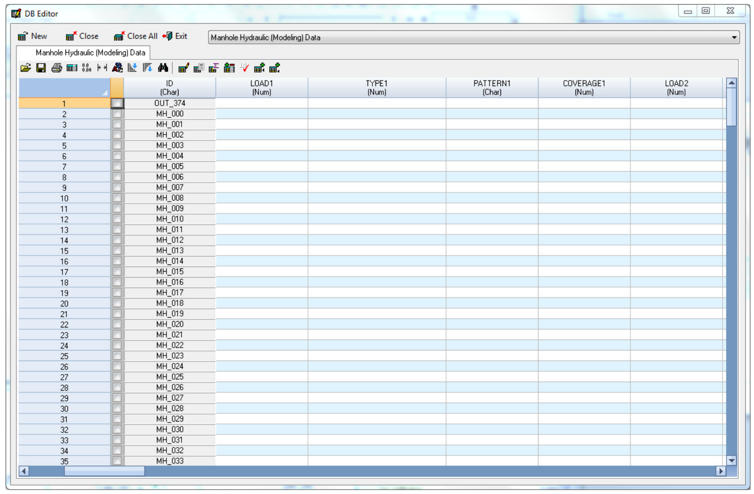
Chapter 5

Building Hydraulic Model Parameters

Student Notes:

Assign Loads

- Loads are assigned to Manholes
- Each Load can have a type, pattern and coverage
- 10 different loads per manhole

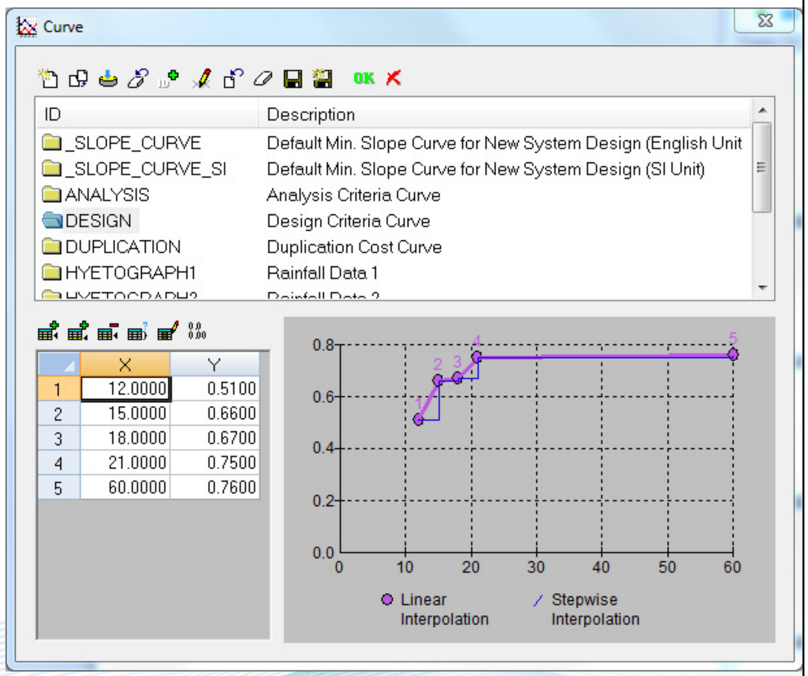


Innovyze®

Student Notes:

Curves

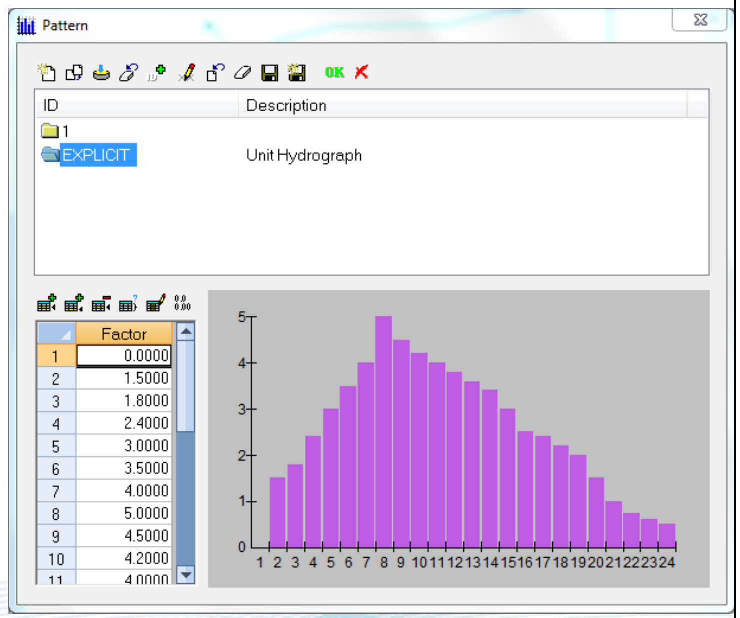
- Design Criteria
- Hyetographs
- Intensity Duration
- Flow Split
- Replacement Cost



Student Notes:

Diurnal Loading Patterns

- Used to modify flows or constituent inflows
- Describe variations
- Enter any number of multipliers for that pattern
- Enter multipliers by type applicable factors
- Modify graphically or by typing



Student Notes:

The Patterns dialog box is used to add or modify loading patterns. A loading pattern is a set of *multipliers* that are applied to baseline sewer loadings.

Step 1. Choose the New button and enter a new pattern identifier and pattern description

Step 2. Enter multipliers for the new pattern in the Time Pattern Factors panel – You may enter any number of multipliers for a pattern. Choose the Insert Row or the Append Row button to add pattern multipliers. As you enter values, the graphical representation of the pattern is updated.

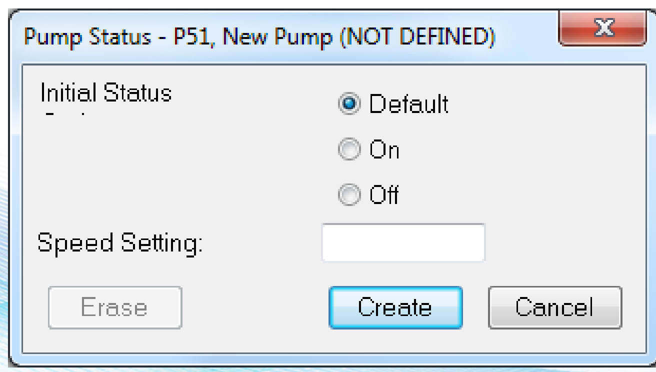
If the pattern does not contain enough multipliers to extend throughout the entire simulation, InfoSewer will simply repeat the pattern as many times as necessary.

Step 3. Press the Save or OK buttons to save the pattern

Step 4. Modify multipliers – You may modify pattern multipliers by selecting the multiplier in the graphical display and dragging the cursor to adjust the multiplier or by selecting the row in the pattern table display and typing the new value.

Initial Operating Status

- Specify whether the pump is opened or closed
- Determines whether wet well is emptying or filling at the beginning of an EPS simulation



Student Notes:

Initial status is used to specify pump status ***at the beginning*** of a simulation. Operational controls (described on the previous page) control changes in a pump's status during an extended period simulation.

Initial status is used during an extended period simulation to inform the program whether a pump is on of off at the beginning of a simulation. This results in having the wet well filling or draining at the beginning of the simulation.

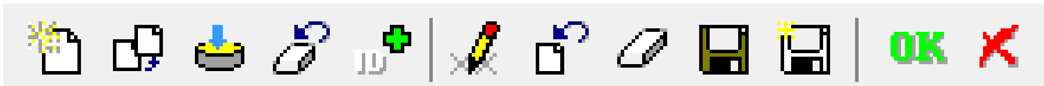
Step 1. Pick a pump from the network map display

Step 2. Choose the Pump Status command from the Tools icon - the Attribute tab of the InfoSewer Browser

Step 3. Enter link status option (On or Off)

Step 4. Press the OK button to close the dialog box

Common Buttons for Options, Curves, and Patterns



- New
- Clone
- Pack Data
- Recall
- ID Counter
- Edit Description
- Reset
- Delete
- Save
- Save as Default
- OK
- Cancel

Student Notes:

New – Create a new option set, curve, pattern, or real time control.

Clone – Copy an existing option set, curve, pattern, or real time control.

Pack Data – Permanently removes any ID from the database that was previously deleted. (Deleted ID's remain in the database for recall capabilities)

Recall - Recalls any ID from the database that was previously deleted.

ID Counter – Opens the dialog that controls the default ID and numbering suggestions for option sets, curves, patterns, and real time controls.

Edit Description – Allows editing of the description of an option set, curve, pattern, or real time control.

Reset – Resets all default options to InfoSewer default values.

Delete – Deletes an option set, curve, pattern, or real time control.

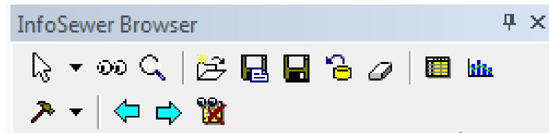
Save – Saves an option set, curve, pattern, or real time control.

Save as Default – Saves all options of the current option set, curve, pattern, or real time control as default values.

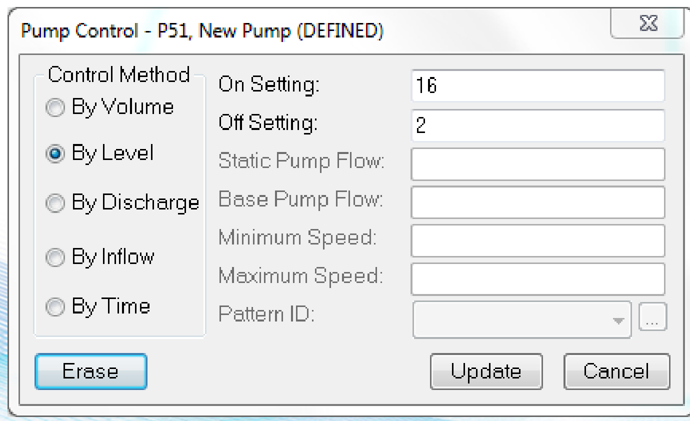
OK – Accepts changes and closes the dialog box.

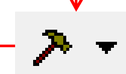
Cancel - Cancels changes and closes the dialog box.

System Operational Controls



- Pump Controls based on:
  - Wet well levels (On/Off):
  - Wet well volumes (On/Off):
  - Targeted pump discharge flow (Speed Setting)
  - Targeted incoming pipe flow (Speed Setting)
  - Time





Student Notes:

You can turn on and turn off a pump by:
Wet Well Level - The level of water in a wet well that causes a pump to turn on or turn off.
Wet Well Volume - The volume of water in a wet well that causes a pump to turn on or turn off.

The pump speed setting can also be adjusted automatically to match a targeted pump discharge flow or a targeted incoming pipe flow. Finally, time controls can also be defined.

In the illustration above, pump 9 is opened when the level in wet well rises above 10 feet and closes when the level in wet well falls below 2 feet.

Step 1. Pick a pump from the network map display

Step 2. Choose the Control command from Tools icon on the Attribute tab of the InfoSewer Browser

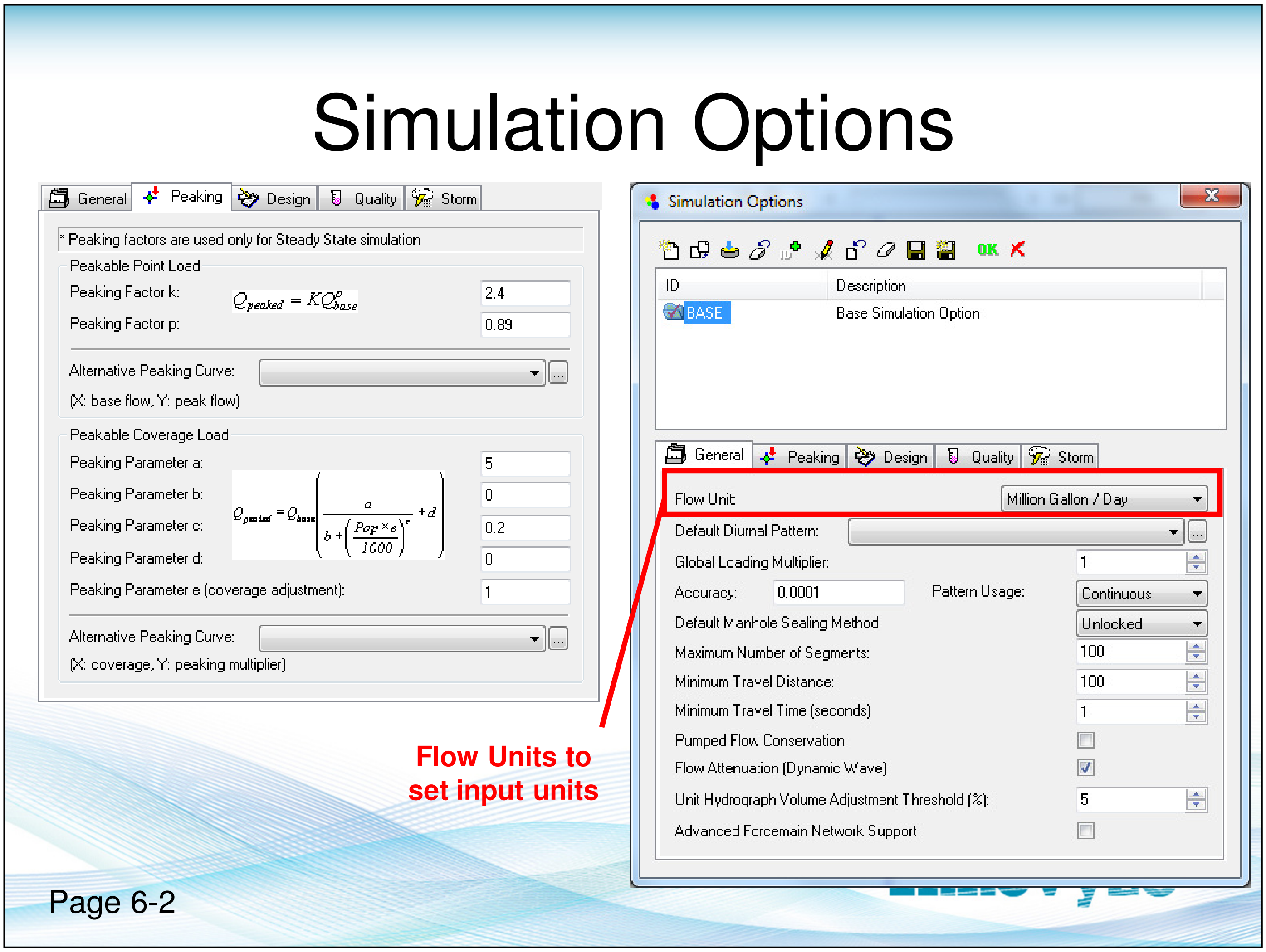
Step 3. Enter controlling level (or volume) for the pump to turn on and off during an extended period simulation.

Step 4. Press the Update button to close the dialog box

Chapter 6

Running a Steady State Analysis and Viewing Results

Student Notes:



Student Notes:

General Tab

Flow Units – Ten choices of flow units are available. Selecting metric units requires metric inputs; selecting imperial units requires imperial inputs. Output units may be changed individually.

Diurnal Pattern Usage - Select whether the pattern will linearly interpolate intermediate values (continuous) or evaluate patterns in a stepwise fashion.

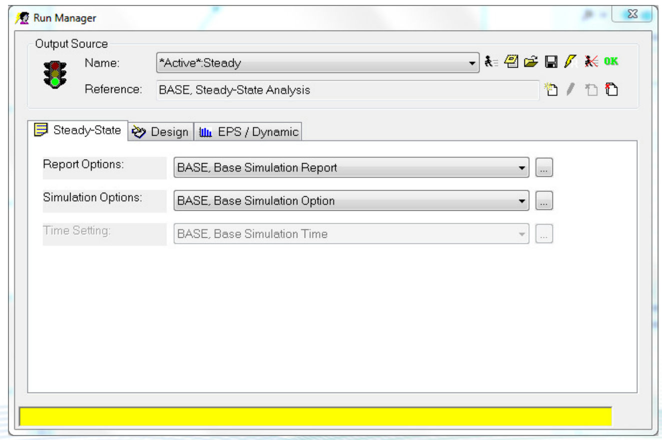
Default Manhole Sealing Method - Use this to specify the default Manhole cover type for Normal Manholes. Choose *Locked* to contain flow inside the manhole structure when the water level rises above the rim elevation and choose *Unlocked* if flow is not contained inside the structure and spills over 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.

Peaking Tab – Peaking factors are used only for Steady State Simulation. Peaking can be calculated in one of two ways: Peakable Point Load (based on Federov equation and Peakable Coveralge Load

Run Manager

- Run simulations
- Load and save output reports
  - Starts the simulation
  - View the Output Report
  - Simulation Status
  - Updates Output Relates



Student Notes:

Set Simulation Options – For a steady state analysis simulation, you may also choose any simulation options including simulation options (peaking equation, units of measurement, etc.), and reporting options.

Steady State Analysis – Performs a steady state analysis that is an instantaneous snap-shot of the collection system. Hydraulic calculations are performed by peaking flows according to a peaking equation and results are produced.

Steady State Design – Similar to a steady state analysis except that d/D and velocity constraints are applied for determining parallel and replacement pipes.

Extended Period Simulation – Flows are determined and routed according to diurnal patterns that are applied over a series of timesteps. Hydraulic conditions and attenuation effects are also determined.

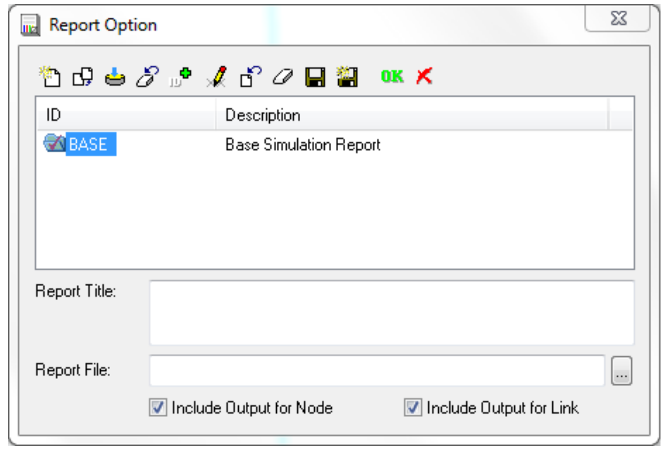
To run a simulation, choose the desired simulation type and then click on the Run button on the Run Manager. After the simulation is complete, a red or green light will appear in the stoplight.

Red – Simulation terminated due to critical error. Error messages appear when this occurs.

Green – The simulation completed with no errors

Output Report (HTML)

- Opens in web browser
- Contains statistics etc.
- Most data can be found through the Report Manager
- Size of report determined by length of simulation, size of network and included report items



Student Notes:

The simulation report opens in the default browser.

Reviewing Simulation Results

- Use any of the following tools to review simulation results in the *Active* output sources
  - Output Report Manager
    - Graphs
    - Tabular Reports
  - Query Report Manager
  - InfoSewer Browser
    - Attribute tab
    - Annotation tab Annotation
    - Contour tabs Contour

Output	
Flow	1.5000 cfs
Head Increase	30.6983 ft
Pump Power	5.2243 hp
Usage	1.0000
Speed	1.0000
Type	Fixed Capacity Pump
From Node	WW48
To Node	JC48

Student Notes:

If the simulation completed successfully or with warnings, you may review simulation results with any of the following tools:

- **Output Report Manager** – Generate graphs and reports showing simulation results for any selected network components.
- **Query Report Manager** - Generate a query report with any combination of model inputs and simulation results.

InfoSewer Browser

Annotation – Annotate (create map text) network components by labeling those components with any combination of simulation results and model input data.

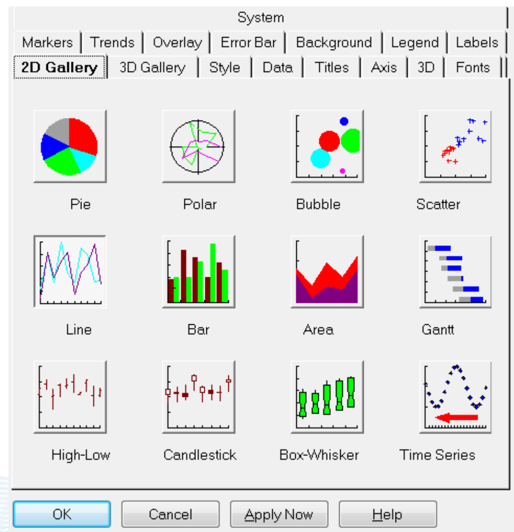
Contour – Generate contours of any nodal numeric simulation result variable.

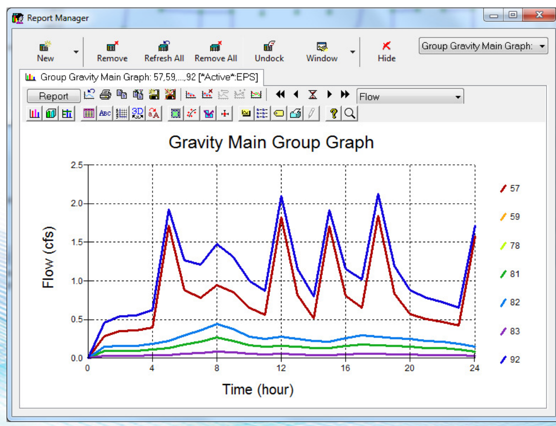
Attribute – Tool for selecting individual network components and viewing simulation results for those components. Discussed in this lesson.

Generation of graphs and reports with the Output Report Manager and the Custom Report Manager are discussed in Section 9.

Output Report Manager

- Edit / manage all output reports and graphs
  - Change styles of text and titles
  - Copy graphs to clipboard
  - Save graphs as images
  - Copy / Paste reports into Excel





Innovyze®

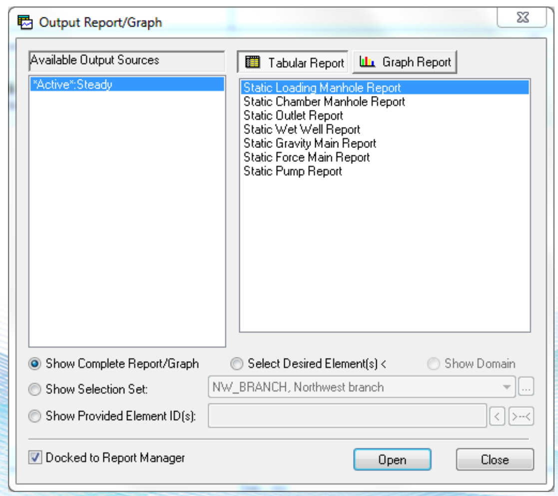
Student Notes:

REPORT MANAGER

The report manager is for viewing, managing, and editing all reports and graphs generated from output data. The toolbar varies for graph and report display. Basic functionality when graphs are displayed is described below. See Chapter 9 for detailed information on report functions.

New Output Report and Graph

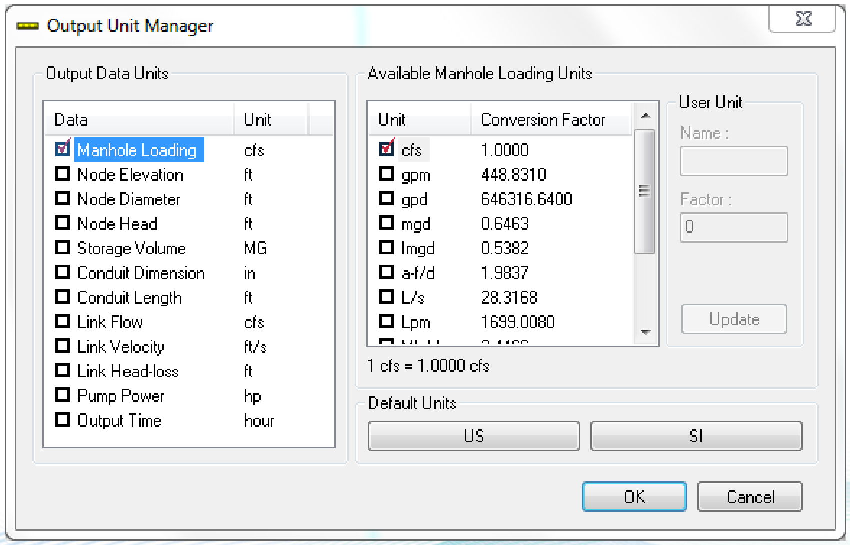
- Create output reports and graphs
  - Select graph or report type
  - Choose elements for graphing or reporting of results
  - Specify a time frame for graphs and reports



Student Notes:

Output Unit Manager

- Changes output display units for:
  - Reports
  - Graphs
  - Contours
  - Labels



Innovyze®

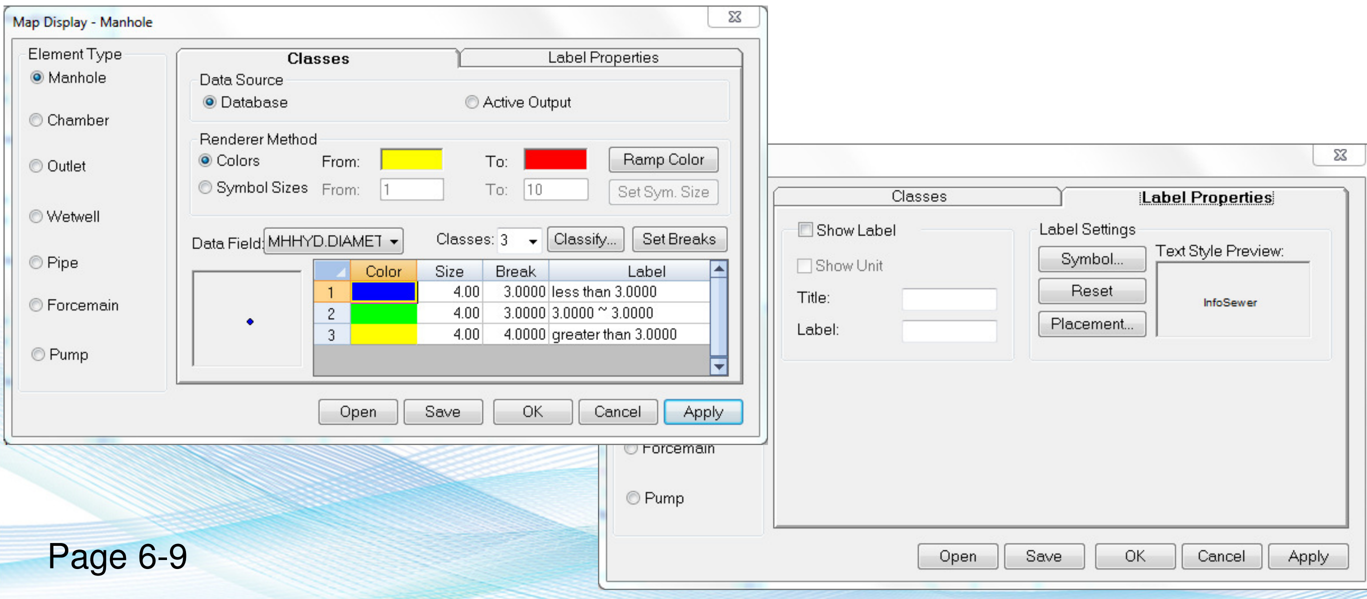
Student Notes:

OUTPUT UNIT MANAGER

The output report manager details the units that will be graphed, reported, or annotated. The output data units show what will be displayed. The data type with the red box has all available options for that data type listed on the right under available units. The red box ( ) under the available units shows which unit is currently selected for output display.

Customizing the Map Display

- Color-code and annotate network components using input data and simulation results



Page 6-9

Student Notes:

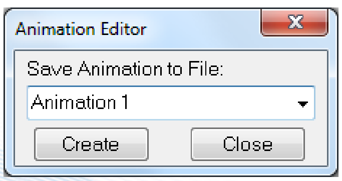
Use the Map Display icon to control map coloring, labeling, and range breaks when displaying values (both input and output) on the map. The user may select any timestep or perform an animated display analysis. Network mapping features include:

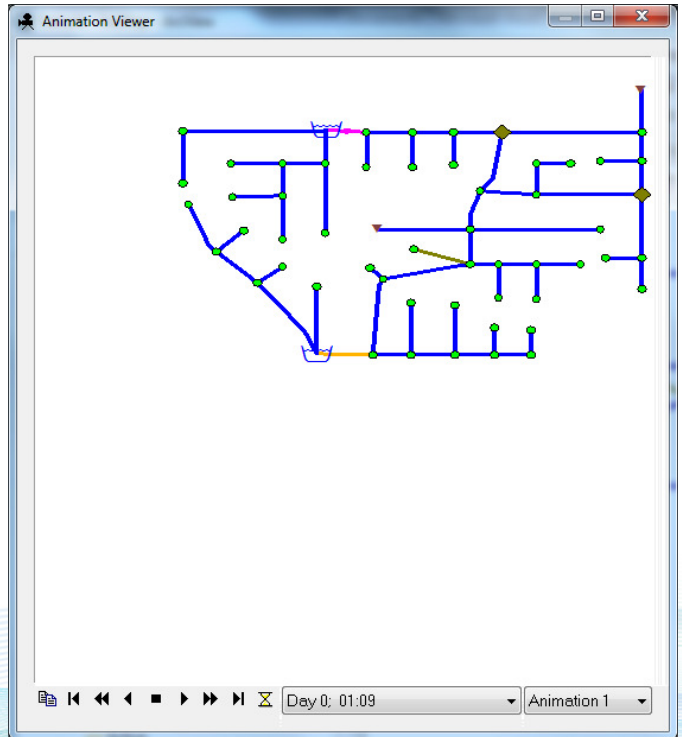
Color-coding - Each component included is colored based on a classification range for a selected result-variable. The range can be customized by the user.

Labeling - Each component in the domain is labeled with the value for the current variable.

Animation

- Movie of the system
- Thematic maps changing over time
- Played using VCR style controls





Student Notes:

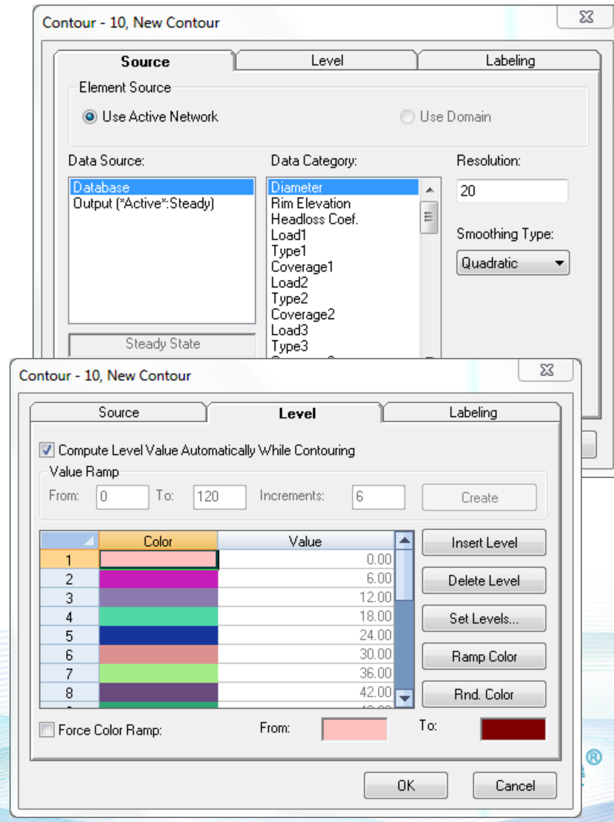
ANIMATION

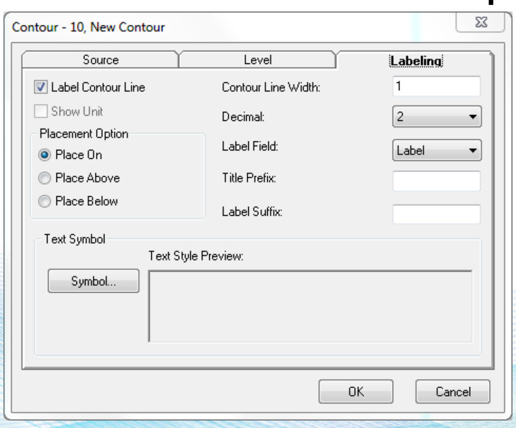
An animation is simply a movie of sequentially progressing thematic maps of the system over time. A thematic map is created for each report time step as specified using the map display dialog. The animation viewer specifies the time the video pauses before showing the thematic map of the next report time step.

The animation editor creates animation files. Always run a simulation with the desired report time step and set the map display prior to creating an animation file. All animation files created within an InfoSewer project are accessible using the animation viewer

Contours

- Input or output data
- Updated after simulations
- Labels optional
- Details one time step





Page 6-11

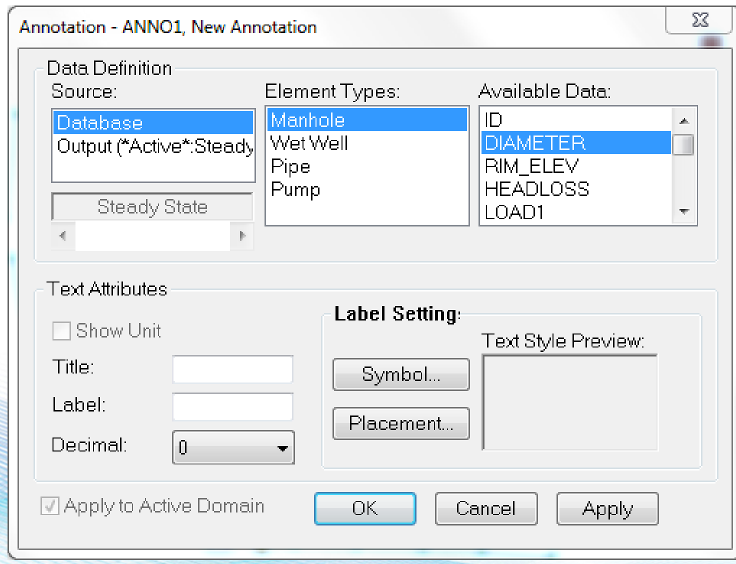
Student Notes:

CONTOURS

Contours can be created from many input and output attributes. The contour interval and label styles may be set by the user. Contours may be updated after simulations. They may cover the entire network of just the current domain. Contours are created for one time step (output data) and will not update as the output time is changed.

Adding Annotations

– Input or Output data
– Updated after simulations
– Details one time step



Student Notes:

Annotations are label layers that can be turned on and off. As with contours, annotations may be updated after a new simulation run. Label placement, style, and size options may be specified. Annotations are created for one time step (output data) and will not update as the output time is changed.

To create an annotation:

- Right click inside the Annotation tab of the InfoSewer Browser and select “New”.
- Specify an ID and description for the Annotation.
- In the Annotation dialog box specify the Source, Element Type, and Available Data that you would like represented in the Annotation.
- Click Apply, then OK.

Chapter 7

Running an Extended Period Simulation and Viewing Results

Student Notes:

Running an Extended Period Simulation

- Simulation run over a series of timesteps – peaking factors are not considered
- Flows are routed and attenuated according to time delays
- Flows, velocities, and other results are determined at each timestep
- Graphs and reports are generated, including the pipe profile graph, showing flow characteristics throughout an extended period simulation

Innovyze®

Student Notes:

An extended period simulation differs from steady state analyses in that it calculates flows over a period of timesteps instead of at an instantaneous “snap-shot”. Also, EPS simulations do not take into account peaking factors that are generally used in determining “worst case” flow conditions within a sewer collection system.

An extended period simulation is the most realistic of simulations as flows are routed and accumulated according to their rates of generation (as determined by a diurnal pattern) within a sewer collection system. Issues related to time delay and attenuation can easily be calculated and viewed following an EPS simulation.

During an EPS simulation, the d/D ratio, flow, and velocity are determined for each pipe in the system. The simulation will also determine surcharged conditions and store volumes until upstream flows subside. Flows are then “bled” back into the system according to the volume stored at the manhole.

At completion of an EPS simulation, numerous graphs and reports can be generated for viewing system behavior. Two significant output results include the pipe profile and a pipe range report. Using these features, the user is able to visualize how flow changes throughout the simulation as well as determine when maximum flows are encountered at each pipe.

Building an EPS Model

- Develop diurnal patterns
- Assign diurnal patterns to manholes
- Specify simulation options/simulation time
- Run extended period simulation analysis
- Review simulation results

Student Notes:

Step 1. Develop diurnal patterns – InfoSewer uses diurnal patterns to apply base loadings over a series of timesteps. For example, if a base loading is 0.2 cfs and the diurnal pattern between hour 2 and hour 3 in the simulation is 0.3, then the load applied during the entire hour will be a constant 0.06 cfs.

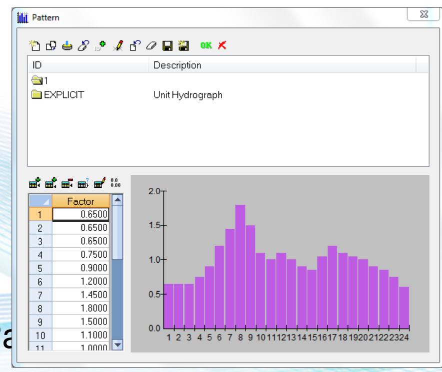
Patterns are used to vary sewage loading throughout the duration of an EPS simulation. It is important to note that the average of all values specified in a diurnal pattern must equal 1.0. This means that if a pattern is specified as having 24 timesteps, the summation of all pattern factors divided by 24 should equal 1.0. Doing this ensures that a pattern is not being inadvertently used as a way of peaking flows.

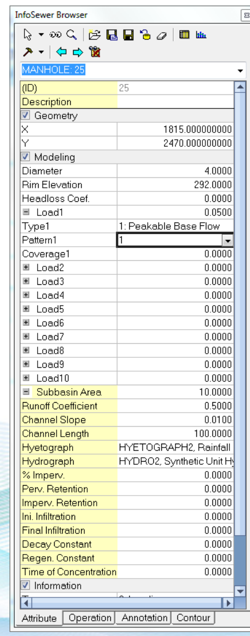
Step 2. Assigning diurnal patterns – Recalling from Section 5, a diurnal pattern was assigned to all manholes in the collection system. InfoSewer allows up to 10 base loads and patterns as well unlimited additional extended loads and patterns to be applied to a single manhole. Multiple loadings and patterns allow you to organize loading conditions from alternate sources of sewage.

Step 3. Specify simulation options, run EPS simulation, and analyze results – Run and review extended period simulation results. These steps are further outlined on the following pages.

Developing Diurnal Patterns

- Set Pattern Time Step (Simulation Time)
- Build patterns for load generation, and assign through:
  - InfoSewer Browser
  - DB Editor
  - Edit Domain/Selection Attributes





Student Notes:

Patterns can either be assigned globally or can be generated for specific manholes within the collection system. A global assignment can either be performed through the “General” tab of the SIMULATION OPTIONS or by using the Domain Manager to assign a pattern. Specific loads and patterns may be assigned to individual manholes through the InfoSewer Browser or by creating a domain.

Assigning loads and patterns to a single manhole

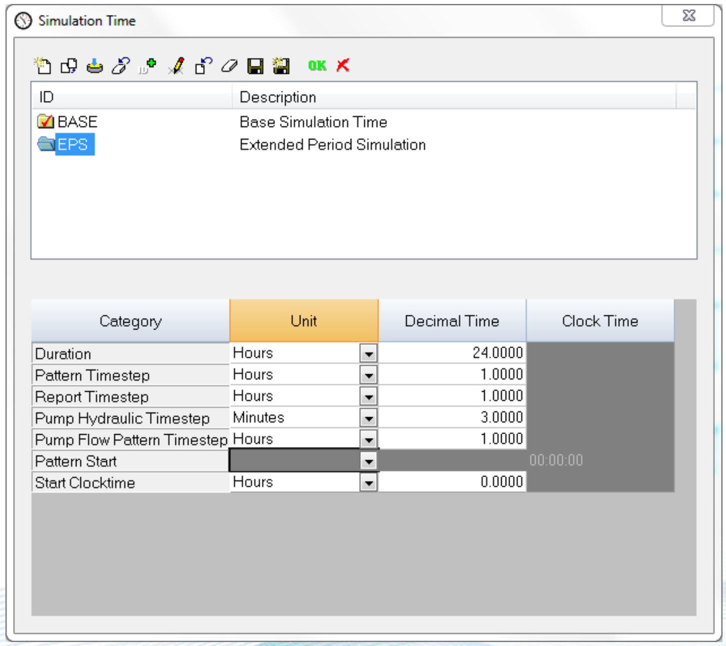
To assign a unique pattern to a single manhole, perform the following:

- Under the Operation tab of the InfoSewer Browser, right click on Pattern folder and select the New command.
- Pick the desired manhole on the map display to assign this new pattern.
- Under the “Modeling” table on the Attribute tab of the InfoSewer Browser, click on the “Pattern1” drop down box to see all patterns available.
- Select the desired pattern from this list to assign this pattern to the selected manhole.

Repeat this step as necessary for assigning patterns to any other manholes. Recalling from Sections 5 & 7, a domain can also be made of the desired manholes and the DB Editor can be used to universally populate information contained in a domain.

Simulation Time

- Computation Time Steps
- Parameter Time Steps
- Clock Time



Student Notes:

Duration - Specify the duration of the simulation.

Pattern Time Step - The amount of time each row in a pattern table represents.

Report Time Step - The simulation outputs will be recorded at the time interval listed here.

Pump Hydraulic Time Step - Time interval for which a hydraulic condition is evaluated for a pump. Decreasing this number will increase result accuracy.

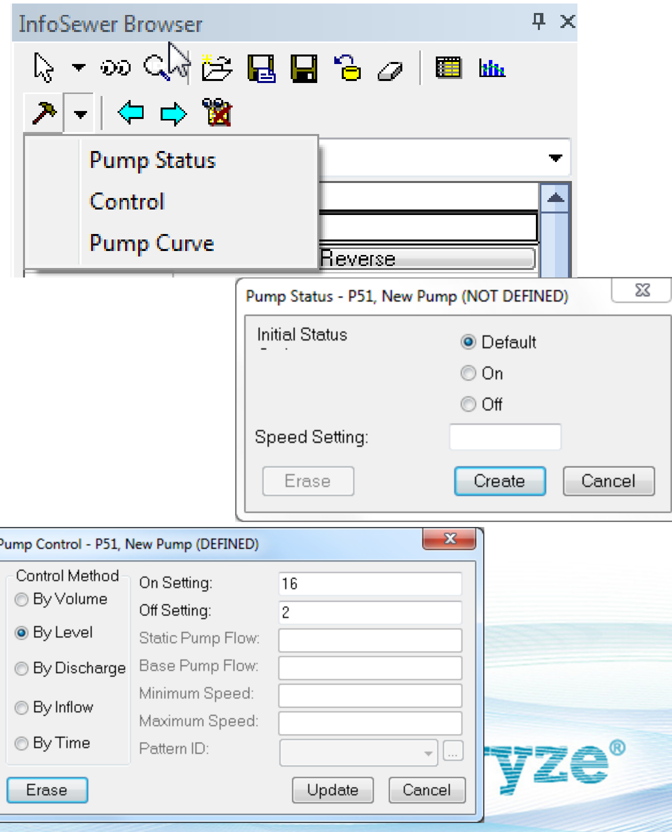
Pump Flow Pattern Timestep - Time step for pump discharge and pump speed patterns that are used for pumps controlled based on time or by desired downstream discharged flows.

Pattern Start – The clock time that corresponds to the first row of all patterns.

Start Clocktime - Specify the time of day corresponding to the start of the simulation.

Initial Status and Controls

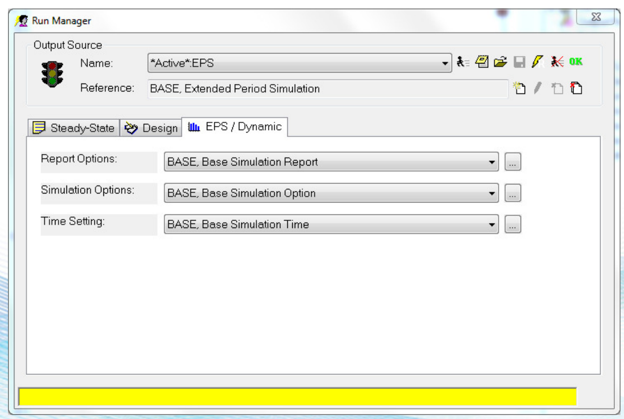
- Initial Status sets status for the first time step
- Changes pump status when criteria is satisfied
- Criteria checked per the rule time step (set in Simulation Time)



Student Notes:

Running an EPS

- Launch the “Run Manager”
  - Choose the EPS/Dynamic tab
  - Specify simulation options
  - Choose the “Run” button on the Run Manager
  - Results are stored in the *ACTIVE*.EPS output source



Student Notes:

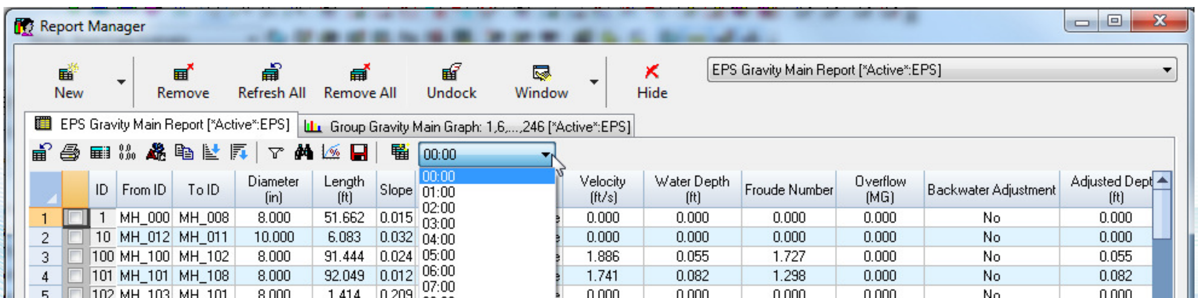
Click the “Run” button at the top of the Run Manager. Be sure the Extended Period Simulation tab is selected prior to running the simulation.

Similar to steady state simulations, EPS simulation results are stored in an output source. For these simulations, the output source is entitled *ACTIVE*:EPS. The Status Indicator (stoplight) should show green. Yellow indicates warnings and red indicates errors during simulation.

Differences from Steady State Results

- The output toolbar controls the time used when displaying the following results:
  - Map Display
  - Attribute Tab

- Reports contain output data from all time steps

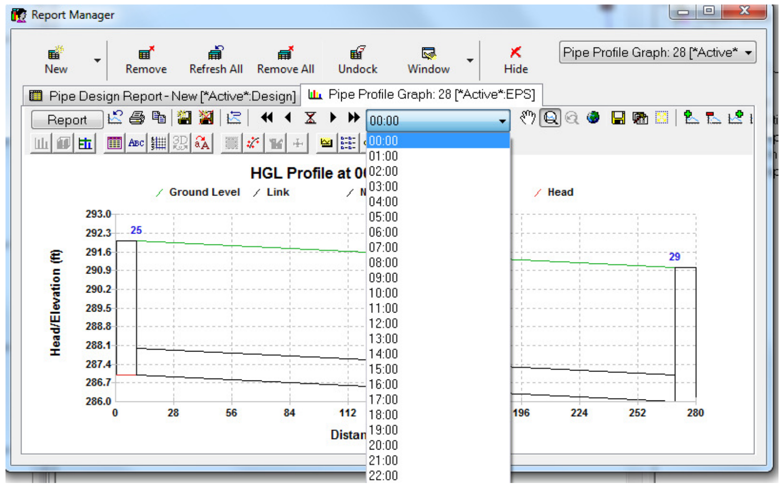


Innovyze®

Student Notes:

Available Reports and Graphs

- InfoSewer Reports
  - Loading manhole
  - Chamber manhole
  - Outlet
  - Wet well
  - Gravity mains
  - Force mains
  - Pump report
  - Gravity main range report



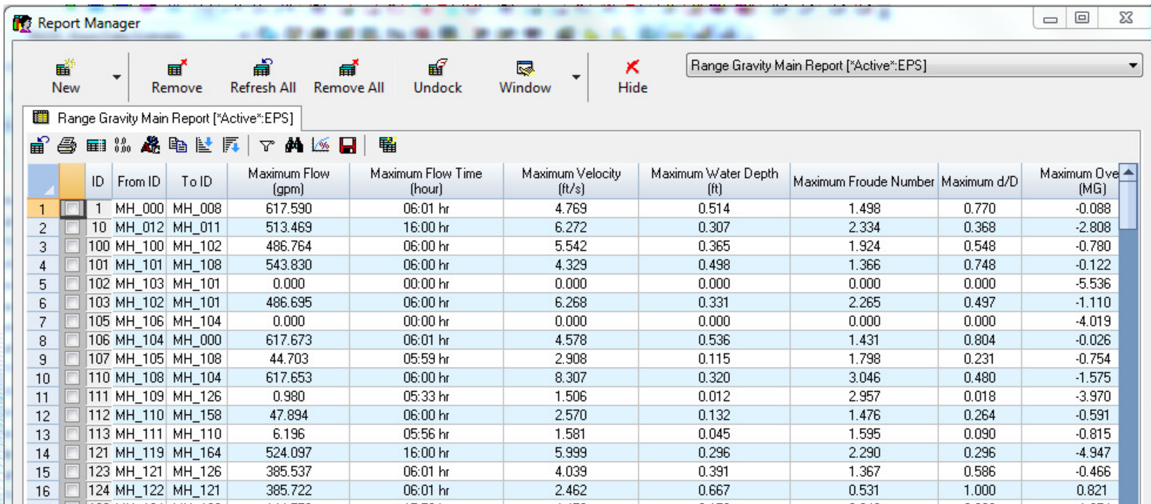
Student Notes:

InfoSewer calculates and reports the following during extended period simulations at any given report timestep:

- **Loading Manhole** - The load, overload and grade.
- **Chamber Manhole** - The hydraulic grade.
- **Outlet** - The flow and hydraulic grade.
- **Wet Well** - The level, grade, volume, %volume, and overflow.
- **Gravity Mains** - The flow, velocity, water depth and d/D ratio.
- **Force Mains** - The flow, velocity and headloss across the pipe.
- **Pump Report** – The flow, head increase and usage (how many pumps are operating).
- **Range Gravity Main Report** - The range gravity main report presents the user with the maximum flow conditions experienced during the EPS simulation and the times that these conditions occurred. This report is also useful in identifying surcharged conditions for viewing a pipe profile.

Range Reports

- Summary output from an EPS Simulation
- Contains information on Minimum, Maximum, and Average Output data



	ID	From ID	To ID	Maximum Flow (gpm)	Maximum Flow Time (hour)	Maximum Velocity (ft/s)	Maximum Water Depth (ft)	Maximum Froude Number	Maximum d/D	Maximum Ove (MG)
1	1	MH_000	MH_008	617.590	06:01 hr	4.769	0.514	1.498	0.770	-0.088
2	10	MH_012	MH_011	513.469	16:00 hr	6.272	0.307	2.334	0.368	-2.808
3	100	MH_100	MH_102	486.764	06:00 hr	5.542	0.365	1.924	0.548	-0.780
4	101	MH_101	MH_108	543.830	06:00 hr	4.329	0.498	1.366	0.748	-0.122
5	102	MH_103	MH_101	0.000	00:00 hr	0.000	0.000	0.000	0.000	-5.536
6	103	MH_102	MH_101	486.695	06:00 hr	6.268	0.331	2.265	0.497	-1.110
7	105	MH_106	MH_104	0.000	00:00 hr	0.000	0.000	0.000	0.000	-4.019
8	106	MH_104	MH_000	617.673	06:01 hr	4.578	0.536	1.431	0.804	-0.026
9	107	MH_105	MH_108	44.703	05:59 hr	2.908	0.115	1.798	0.231	-0.754
10	110	MH_108	MH_104	617.653	06:00 hr	8.307	0.320	3.046	0.480	-1.575
11	111	MH_109	MH_126	0.980	05:33 hr	1.506	0.012	2.957	0.018	-3.970
12	112	MH_110	MH_158	47.894	06:00 hr	2.570	0.132	1.476	0.264	-0.591
13	113	MH_111	MH_110	6.196	05:56 hr	1.581	0.045	1.595	0.090	-0.815
14	121	MH_119	MH_164	524.097	16:00 hr	5.999	0.296	2.290	0.296	-4.947
15	123	MH_121	MH_126	385.537	06:01 hr	4.039	0.391	1.367	0.586	-0.466
16	124	MH_122	MH_121	385.722	06:01 hr	2.462	0.667	0.531	1.000	0.821

Student Notes:

Viewing Results in the Attribute Tab

- Select a feature
- Specify output time step using the output toolbar

Pump

☑ Output	
Flow	657.786 gpm
Head Increase	18.615 ft
Pump Power	3.095 hp
Usage	1.000
Speed	1.000
Type	Design Point Pump
From Node	WW-1

Manhole

☑ Output	
Base Flow	0.000 cfs
Storm Flow	0.000 cfs
Total Storm Flow	0.000 cfs
Grade	16.135 ft
Status	Not Full
Hydraulic Jump	No
Unfilled Depth	1.092 ft
Surcharge Depth	4.480 ft
Type	Loading Manhole
Rim Elevation	17.227 ft
Diameter	48.000 ft

Wet Well

☑ Output	
Water Level	12.134 ft
Grade	16.134 ft
Volume	1.164 MG
% Volume	64.502 %
Overflow	0.000 MG
Type	Variable Area Wet Well
Bottom Level	4.000 ft
Minimum Level	0.000 ft
Maximum Level	20.700 ft

Pipe

☑ Output	
Flow	22.981 gpm
Flow Type	Pressurized
Velocity	2.807 ft/s
Water Depth	0.074 ft
Froude Number	2.185
Overflow	0.000 MG
Backwater	Yes
Adjusted Depth	0.500 ft

Student Notes:

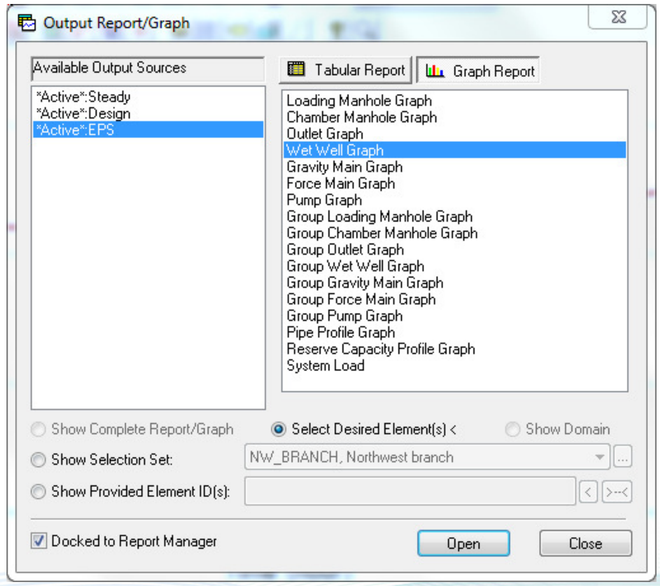
Creating Graphs – Method 1

- Press from the Attribute Tab of the Model Explorer
- Graph opens for the currently selected feature
- Change the parameter displayed using the drop-down pick box
- Press Report to access a report of the graphed data
- Press to graph 2 parameters on the same graph

Student Notes:

Creating Graphs – Method 2

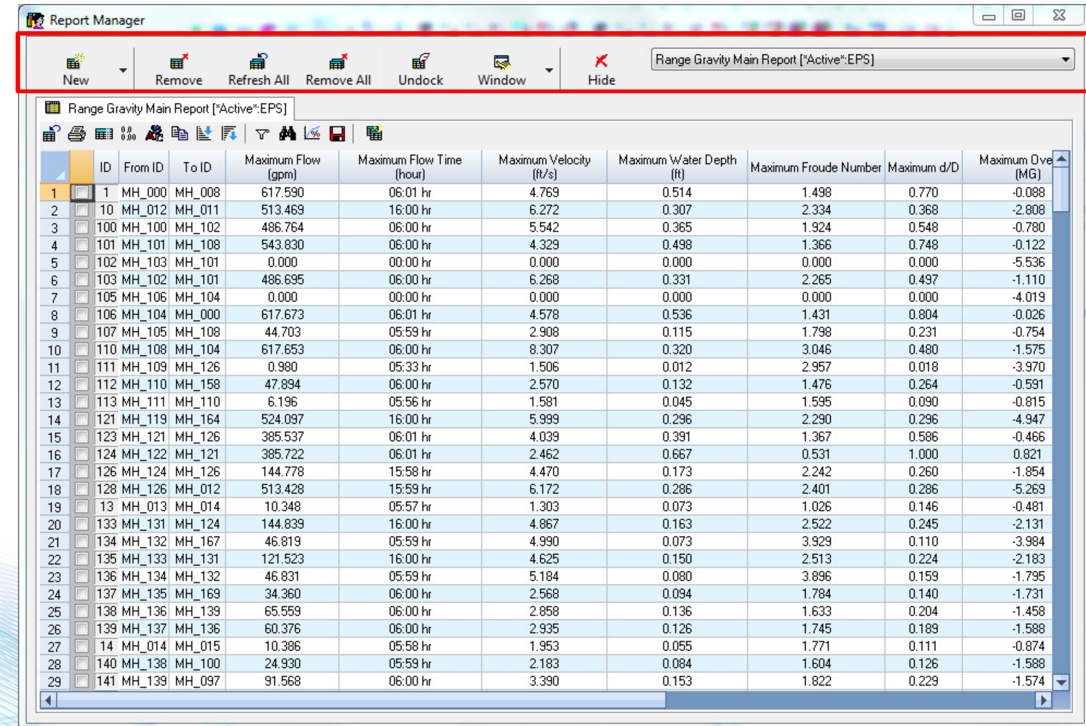
- Open the Report Manager
- Press New to open the Output Report and Graph (if necessary)
- Select the graph type then press OPEN
- Can confine to domain if desired
  – Group Graphs Only



Innovyze®

Student Notes:

Common Buttons for Reports and Graphs



	ID	From ID	To ID	Maximum Flow (gpm)	Maximum Flow Time (hour)	Maximum Velocity (ft/s)	Maximum Water Depth (ft)	Maximum Froude Number	Maximum d/D	Maximum Ove (MG)
1	1	MH_000	MH_008	617.590	06:01 hr	4.769	0.514	1.498	0.770	-0.088
2	10	MH_012	MH_011	513.469	16:00 hr	6.272	0.307	2.334	0.368	-2.808
3	100	MH_100	MH_102	486.764	06:00 hr	5.542	0.365	1.924	0.548	-0.780
4	101	MH_101	MH_108	543.830	06:00 hr	4.329	0.498	1.366	0.748	-0.122
5	102	MH_103	MH_101	0.000	00:00 hr	0.000	0.000	0.000	0.000	-5.536
6	103	MH_102	MH_101	486.695	06:00 hr	6.268	0.331	2.265	0.497	-1.110
7	105	MH_106	MH_104	0.000	00:00 hr	0.000	0.000	0.000	0.000	-4.019
8	106	MH_104	MH_000	617.673	06:01 hr	4.578	0.536	1.431	0.804	-0.026
9	107	MH_105	MH_108	44.703	05:59 hr	2.908	0.115	1.798	0.231	-0.754
10	110	MH_108	MH_104	617.653	06:00 hr	8.307	0.320	3.046	0.480	-1.575
11	111	MH_109	MH_126	0.980	05:33 hr	1.506	0.012	2.957	0.018	-3.970
12	112	MH_110	MH_158	47.894	06:00 hr	2.570	0.132	1.476	0.264	-0.591
13	113	MH_111	MH_110	6.196	05:56 hr	1.581	0.045	1.595	0.090	-0.815
14	121	MH_119	MH_164	524.097	16:00 hr	5.999	0.296	2.290	0.296	-4.947
15	123	MH_121	MH_126	385.537	06:01 hr	4.039	0.391	1.367	0.586	-0.466
16	124	MH_122	MH_121	385.722	06:01 hr	2.462	0.667	0.531	1.000	0.821
17	126	MH_124	MH_126	144.778	15:58 hr	4.470	0.173	2.242	0.260	-1.854
18	128	MH_126	MH_012	513.428	15:59 hr	6.172	0.286	2.401	0.286	-5.269
19	13	MH_013	MH_014	10.348	05:57 hr	1.303	0.073	1.026	0.146	-0.481
20	133	MH_131	MH_124	144.839	16:00 hr	4.867	0.163	2.522	0.245	-2.131
21	134	MH_132	MH_167	46.819	05:59 hr	4.990	0.073	3.929	0.110	-3.984
22	135	MH_133	MH_131	121.523	16:00 hr	4.625	0.150	2.513	0.224	-2.183
23	136	MH_134	MH_132	46.831	05:59 hr	5.184	0.080	3.896	0.159	-1.795
24	137	MH_135	MH_169	34.360	06:00 hr	2.568	0.094	1.784	0.140	-1.731
25	138	MH_136	MH_139	65.559	06:00 hr	2.858	0.136	1.633	0.204	-1.458
26	139	MH_137	MH_136	60.376	06:00 hr	2.935	0.126	1.745	0.189	-1.588
27	14	MH_014	MH_015	10.386	05:58 hr	1.953	0.055	1.771	0.111	-0.874
28	140	MH_138	MH_100	24.930	05:59 hr	2.183	0.084	1.604	0.126	-1.588
29	141	MH_139	MH_097	91.568	06:00 hr	3.390	0.153	1.822	0.229	-1.574

Student Notes:

New – Open a new Output Report/Graph, Customized Report, Query Report, or Query Summation Report

Remove – Close the active report or graph

Refresh All – Refresh all of the open reports and graphs with the latest output

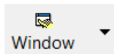
Remove All – Close all of the open reports and graphs

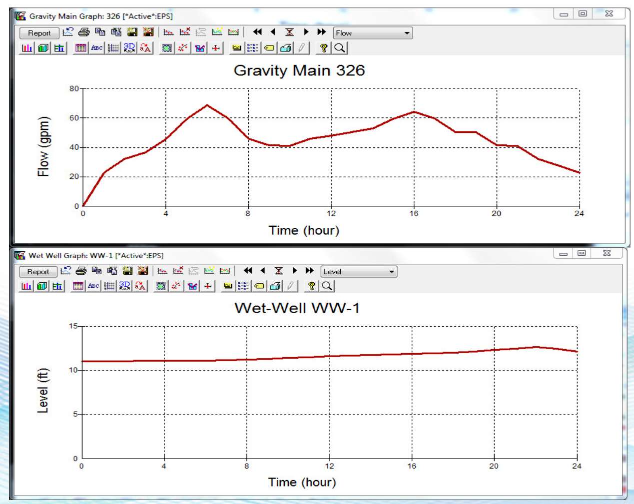
Undock – Remove the active report or graph from the report manager and open it in a separate window

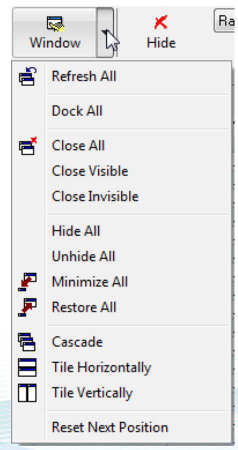
Window – Options to control the undocked reports and graphs

Hide – Close the Report Manager

Floating Graphs

- Press  from the Report Manager Toolbar
- The currently selected graph will undock
- Use the  drop-down to change the undocked graphs placement





Student Notes:

Refresh All – Refresh all of the open reports and graphs with the latest output

Dock All – Dock all of the open reports and graphs to the Report Manager

Close All – Close all undocked reports and graphs

Close Visible – Close all undocked open reports and graphs

Close Invisible – Close all minimized and hidden reports and graphs

Hide All – Hide all undocked reports and graphs

Unhide All – Restore all of the hidden graphs

Minimize All – Minimize all undocked graphs

Restore All – Restore all hidden and minimized undocked graphs

Cascade – Cascade all of the visible reports and graphs

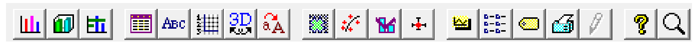
Tile Horizontally – Tile all of the visible graphs horizontally

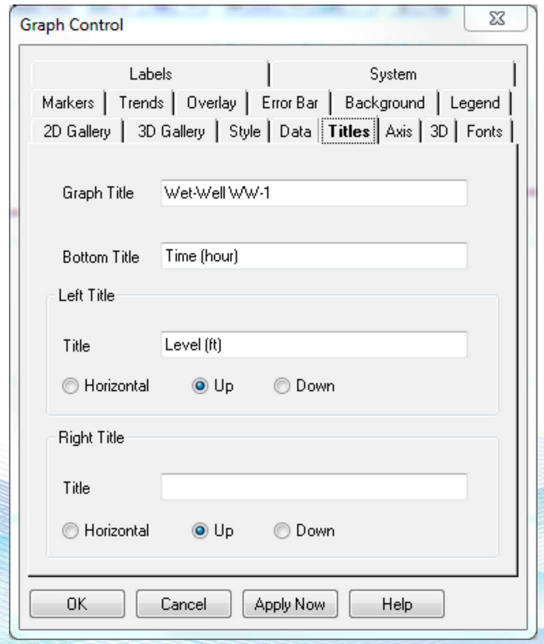
Tile Verticle – Tile all of the visible graphs vertically

Reset Next Position – Undock the next graph to the first set undock position

Modifying and Exporting Graphs

- Print
- Copy to clipboard





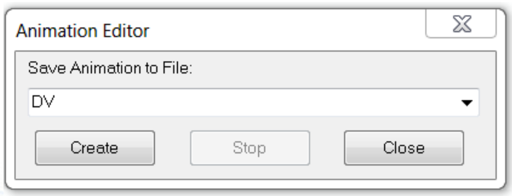
Innovyze®

Student Notes:

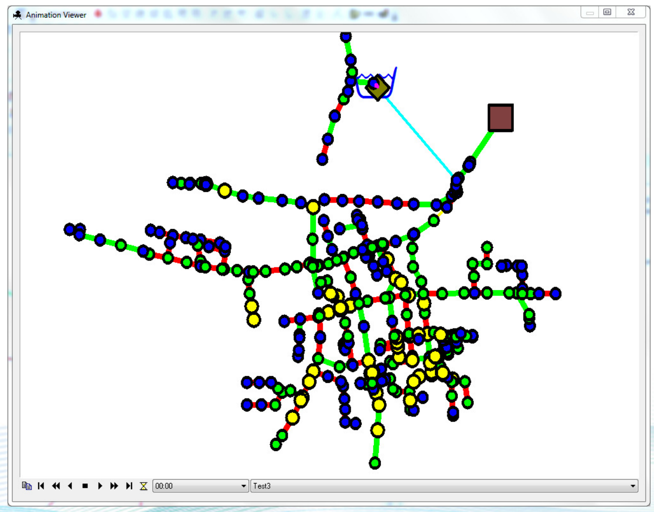
Animations

- Create Thematic Map
- Turn on/off desired background layers
- Set time step to desired start time

- Create Animation



- View Animation



Innovyze®

Student Notes:

Chapter 8

Design Simulation

Student Notes:

Steady State Design Analysis

- Determine parallel and replacement pipes for the collection system
- Flows are peaked according to the prescribed peaking equation
- d/D ratios and velocities are determined and evaluated against analysis criteria curves and velocity constraints
- When d/D ratio exceeds analysis criteria or velocity is not within the range specified, InfoSewer determines parallel and replacement pipes
- Output design report shows deficient pipes

Student Notes:

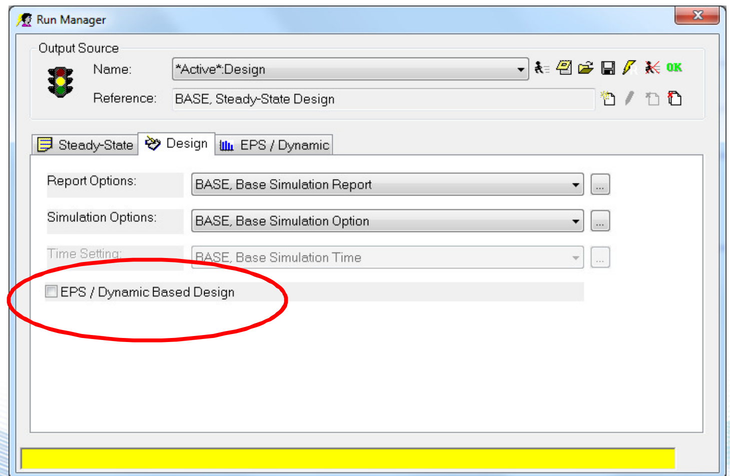
A steady state design simulation is performed independently of other simulation modules, but is based on a steady state analysis simulation. The difference between the two is that a steady state design simulation determines parallel and replacement pipes for those pipes that exceed the analysis d/D criteria curve. Velocity conditions are also evaluated for determining which pipes have velocities that fall outside the specified criteria.

From a steady state design simulation, the user is able to determine which pipes need additional capacity and to calculate a cost for constructing such additional capacity. It is important to note that only those pipe sizes that are contained within the parallel and replacement cost curves are evaluated by InfoSewer. If these curves do not contain enough intermediate points, then program may recommend a pipe that is too large.

At completion of a steady state design simulation, a design report is generated. This report contains calculated velocities of each pipe, whether they fell within the velocity constraints and denotes which pipes need additional capacity in the form of parallel and replacement pipes.

EPS Design Analysis

- Runs an EPS simulation and displays results for worst case



Innovyze®

Student Notes:

An EPS Design Analysis is identical to a Steady State Design Analysis. To enable an EPS Design Analysis, check the EPS/Dynamic Based Design option in the Design Tab of the Run Manager. The displayed results will be derived from the worst case during the EPS run.

Building a Steady State Design Model

- Develop parallel and replacement cost curves
- Assign analysis and design criteria curves
- Specify design simulation options
- Run design analysis
- Review simulation results

Student Notes:

Steps to build and run a steady state design simulation:

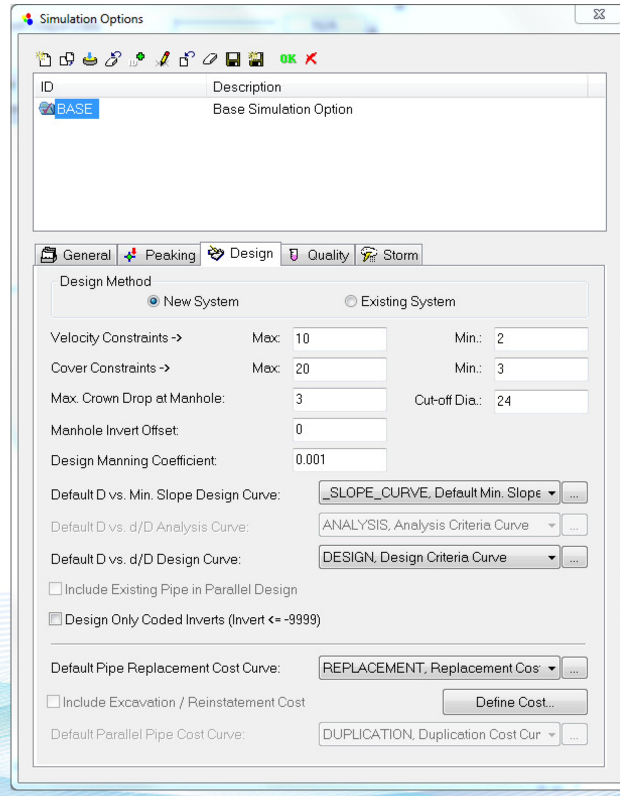
Step 1. Develop parallel pipe and replacement pipe cost curves – InfoSewer will determine parallel and replacement pipes for those pipes in the system that are under-designed. In a steady state simulation, flows are routed and then peaked according to a peaking equation. The d/D ratio at this peaked value is then determined and compared with the analysis d/D ratio. If the peaked value exceeds this value, a parallel and replacement pipe are determined.

Step 2. Assigning criteria and cost curves – Recalling from Section 5, design and analysis criteria curves as well as parallel and replacement cost curves were generated and assigned to the hydraulic model under the "Design" tab of the SIMULATION OPTIONS. These curves are used by the steady state design simulation for determining excess flow conditions and recommending replacement facilities.

Step 3. Specify simulation options, run steady stated design simulation, and analyze results – Run and review steady state design simulation results. These steps are further outlined on the following pages.

Assigning Criteria and Cost Curves

- Specify global values
  - Velocity constraints
  - Design Manning's n
  - Analysis and design criteria curves
  - Replacement and parallel cost curves



Page 8-5

Student Notes:

These curves can either be assigned globally or can be generated for specific pipes within the collection system.

Assigning global criteria and cost curves

To assign global cost and/or criteria curves, perform the following:

- From the InfoSewer Browser – OPERATION tab, select the SIMULATION OPTIONS folder.
- Click on the "Design" tab.
- Enter analysis and design criteria curves as well as replacement and parallel cost curves.
- Choose the "OK" button to close the SIMULATION OPTIONS dialog box.

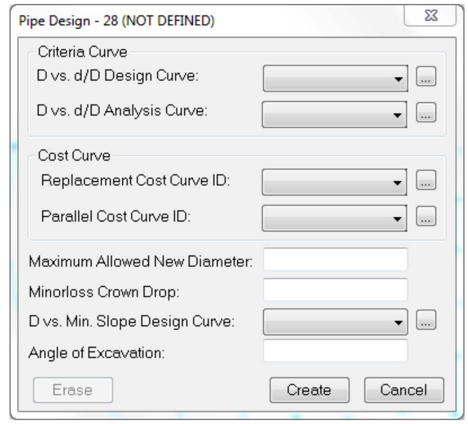
Assigning criteria and cost curves to a single pipe

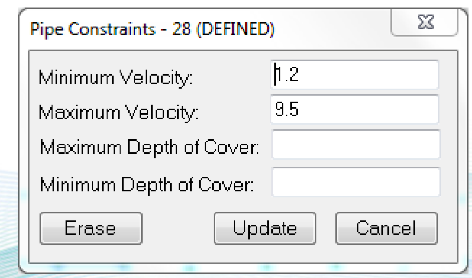
To assign a unique cost or criteria curves to a single pipe, perform the following:

- Pick the desired pipe on the map display.
- Select the "Pipe Design" command from Tools icon on the Attribute tab of the InfoSewer Browser.
- Enter analysis and design criteria curves as well as replacement and parallel cost curves. Any curves assigned at this juncture will over-ride the global condition for that specific pipe.
- Choose the "Create" button to close the "Pipe Design" dialog box.

Unique Design Constraints for Specific Pipes

- Click on any pipe and use the Pipe Design command from Tools icon
- Set unique design and analysis criteria curves for individual components
- Set unique parallel and replacement cost curves for individual components
- Click on any pipe and use the Pipe Constraints command from Tools icon
- Set unique velocity constraints for individual components





Student Notes:

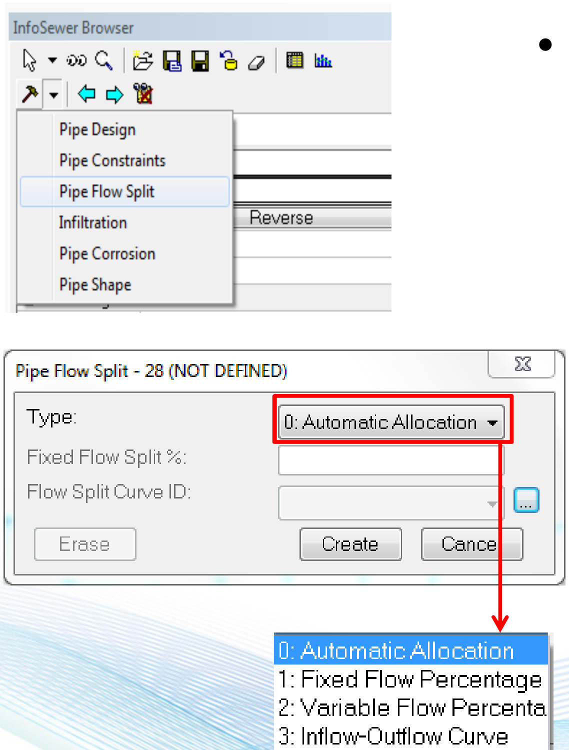
Assigning criteria and cost curves to multiple pipes

To assign criteria and cost curves for multiple pipes, perform the following:

- Create a domain containing the desired pipes. This can include all pipes in a single service area, pipes along a given street or pipe, etc.
- From the InfoSewer Control Center -> InfoSewer button -> Edit menu, choose the Group Edit on Domain command.
- Select the “Pipe Design” tab on the “Group Editing” dialog box.
- Specify the desired curves by clicking on their respective drop-down boxes or click on the […] button to create new curves.
- Choose “APPLY” and then “CLOSE” to save and close the “Group Editing – Pipe Design” dialog box.

Repeat this step as necessary for any additional desired groupings of pipes.

Setting Flow Splits



- Able to specify one of three different methods for splitting pipe flow:
  - 0: Automatic – Flows are automatically calculated based on invert determination and pipe diameter
  - 1: Percentage – Flows are split according to a fixed flow percentage
  - 2: Variable Flow – Specify a curve that represents a flow percentage as a function of upstream flows
  - 3: Inflow – Outflow – Specify a curve that represents the outflow as a function of the inflow

Page 8-7

Student Notes:

Flow splits are created when two or more pipes exit a single manhole. InfoSewer allows flow splits to be modeled by the following methods:

Automatic Flow Split Calculation – When flow splits are digitized, InfoSewer will automatically calculate the flows exiting the flow split. This calculation is done by the invert comparison methodology where it is assumed that the lower pipe invert will accumulate flow until the invert elevation of the second pipe is reached. Flows are then split between the two pipes based upon the carrying capacity of their incremental cross sectional areas.

Fixed Flow Percentage – In this method, you will specify the flow entering a downstream pipe as a percentage of total flow entering the upstream manhole. While the percentage allocation method is practical for steady state simulations, it is highly impractical for an EPS condition where flows fluctuate during a hydraulic simulation.

Variable Flow Percentage – The relationship between the flows entering and exiting a manhole structure can be expressed as a curve. This curve is defined as percentage vs. flow whereby the Y-axis of the curve represents the amount of flow entering the selected downstream pipe as a percentage (0-100) of the total upstream flow (the X-axis). The variable percentage method is the most accurate way of defining flow splits, but is difficult to determine due to lack of accurate flow monitoring data.

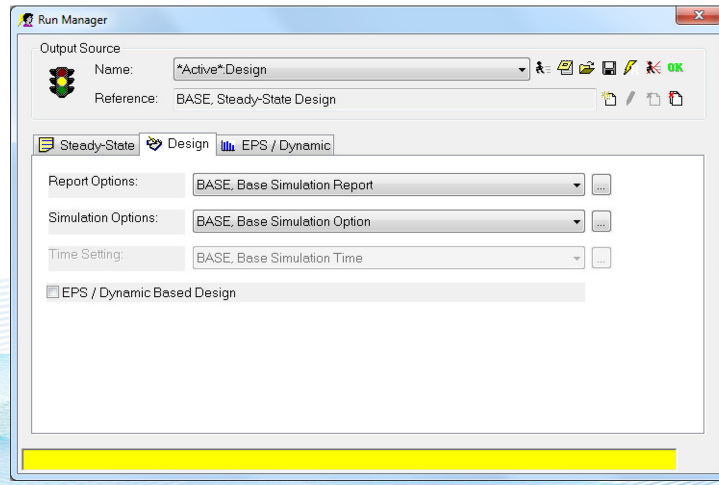
Run Manager

- Run simulations
- Load and save output reports

Starts the simulation

View the Output Report

Simulation Status



Student Notes:

Be sure the Steady State Design tab is selected prior to running the simulation.

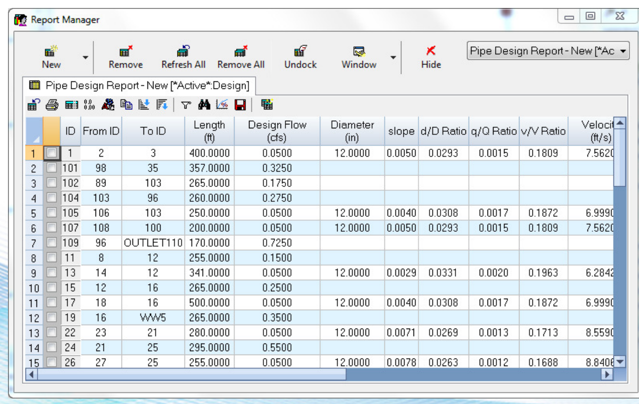
Steady state design simulation results are stored in an output source. For these simulations, the output source is entitled *ACTIVE*:Design..

InfoSewer calculates and reports the following during steady state design simulations:

- ***Velocity Status*** - The pass or fail condition of the pipe velocity. The condition for pass/fail is dependent upon the velocity conditions specified.
- ***Velocity Difference (Min/Max)*** - The algebraic difference between the actual pipe velocity and the high and low ends of the velocity conditions specified.
- ***Replacement Diameter*** – If the existing pipe was to be replaced, this is the diameter that would be used.
- ***Replacement Cost*** - The total cost of the replacement pipe. The value is determined by multiplying the unit cost by the linear footage of the pipe.
- ***Parallel Diameter*** - If the existing pipe was to remain in place, this is the diameter that would be used to parallel the pipe.
- ***Parallel Cost*** - The total cost of the parallel pipe. The value is determined by multiplying the unit cost by the linear footage of the pipe.

Creating Reports – Method 1

- Press from the Attribute tab of the InfoSewer Browser
- Report opens for the currently selected feature type
- Highlighted data can be copied to the Windows clipboard



	ID	From ID	To ID	Length (ft)	Design Flow (cfs)	Diameter (in)	slope	d/D Ratio	q/Q Ratio	v/V Ratio	Veloci (ft/s)
1	1	2	3	400.0000	0.0500	12.0000	0.0050	0.0293	0.0015	0.1809	7.5620
2	101	98	35	357.0000	0.3250						
3	102	89	103	265.0000	0.1750						
4	104	103	96	260.0000	0.2750						
5	105	106	103	250.0000	0.0500	12.0000	0.0040	0.0308	0.0017	0.1872	6.9990
6	107	108	100	200.0000	0.0500	12.0000	0.0050	0.0293	0.0015	0.1809	7.5620
7	109	96	OUTLET110	170.0000	0.7250						
8	11	8	12	255.0000	0.1500						
9	13	14	12	341.0000	0.0500	12.0000	0.0029	0.0331	0.0020	0.1963	6.2842
10	15	12	16	265.0000	0.2500						
11	17	18	16	500.0000	0.0500	12.0000	0.0040	0.0308	0.0017	0.1872	6.9990
12	19	16	WW5	265.0000	0.3500						
13	22	23	21	280.0000	0.0500	12.0000	0.0071	0.0269	0.0013	0.1713	8.5590
14	24	21	25	295.0000	0.5500						
15	26	27	25	255.0000	0.0500	12.0000	0.0078	0.0263	0.0012	0.1688	8.8406

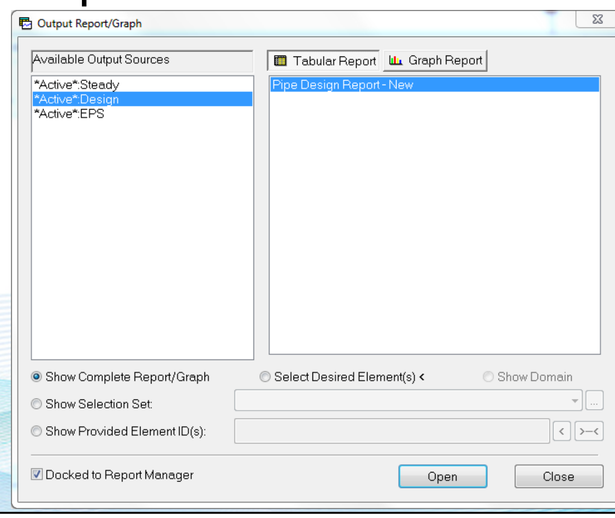
innovyze®

Page 8-9

Student Notes:

Creating Reports – Method 2

- Open the Report Manager
- Press New to open the Output Report and Graph (if necessary)
- Select the report type then press OPEN
- Can confine to:
  - Selected Elements
  - Domain
  - Selection Set



Page 8-10

Student Notes:

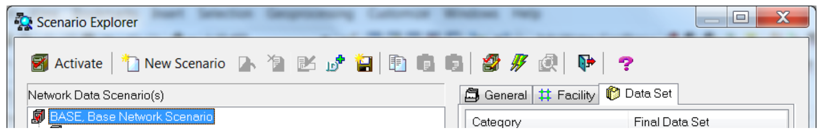
Chapter 9

Scenario Management

Student Notes:

Scenario Management

- Allows you to store numerous scenarios, planning periods, and alternatives in one InfoSewer Project
  - 2015, 2025, 2050 system masterplans
  - Pump controls
  - Tank sizing
  - System outages
  - Developments



- Maintain data integrity for each alternative
- Choose the features that will be modeled for each scenario

Student Notes:

InfoSewer's scenario management feature allows you to develop multiple specific modeling scenarios for a current collection system. Any number of scenarios can be developed for one InfoSewer project.

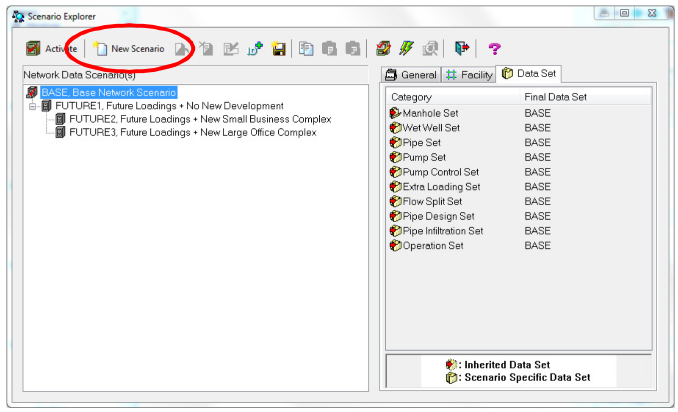
You can, with one InfoSewer project, model existing facilities, existing and future facilities together, a skeletonized version of the model, a single service area, or the same network with projected winter loadings.

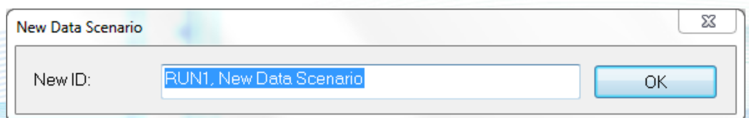
Scenario manager functions

- **Store alternative network layouts** - Store in one InfoSewer project your existing facilities and proposed future facilities.
- **Select subsets of facilities to model** - Model a specified subset of network components such as a single service area, or exclude a set of facilities to assess system performance while those facilities are off-line.
- **Store and retrieve any number of facility data characteristics** – Store facility characteristics (diameters, roughnesses, materials, operating conditions) in different data sets.
- **Model collection system under differing loading and operating conditions** - Store any number of different loading conditions in manhole sets.

Creating a Custom Scenario

- Open Scenario Manager
- Select parent scenario
- Select “New Scenario” button to create a child scenario
- Enter scenario name and description
- Choose options sets
- Choose data sets
- Specify facility set





Innovyze®

Page 9-3

Student Notes:

Perform the following to create a new custom scenario:

Choose the “Scenario Manager” command from the InfoSewer Control Center -> InfoSewer button -> Scenario menu.

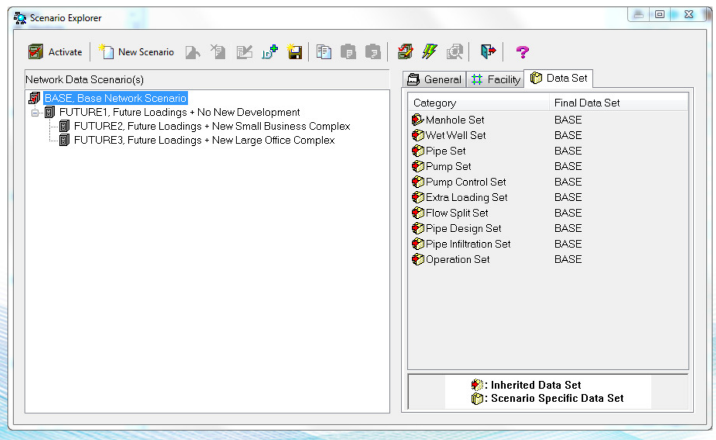
- Choose the parent scenario (described on the next page).
- Choose the “New Scenario” button.
- Enter scenario name and narrative description.
- Select the “General” tab and choose simulation option sets or use BASE option sets.
- Select the “Data Set” tab and choose data sets or have data sets inherited from the parent scenario (described on the next page).
- Select the “Facility” tab and choose facility set definition (described after the first exercise).
- The new scenario is created one level below the parent scenario and inherits one or more properties from the parent scenario.

Saving scenario definitions – To accept the new scenario, choose the “Save” or “OK” buttons.

Activating scenarios – To activate (load) the new scenario and its components, choose the “Activate” button (first left button on the Scenario Manager).

Scenario Components

- A scenario is comprised of the following:
  - – Simulation option sets (General tab)
  - – Facility set (Facility tab)
  - – Data Set (Data Set tab)

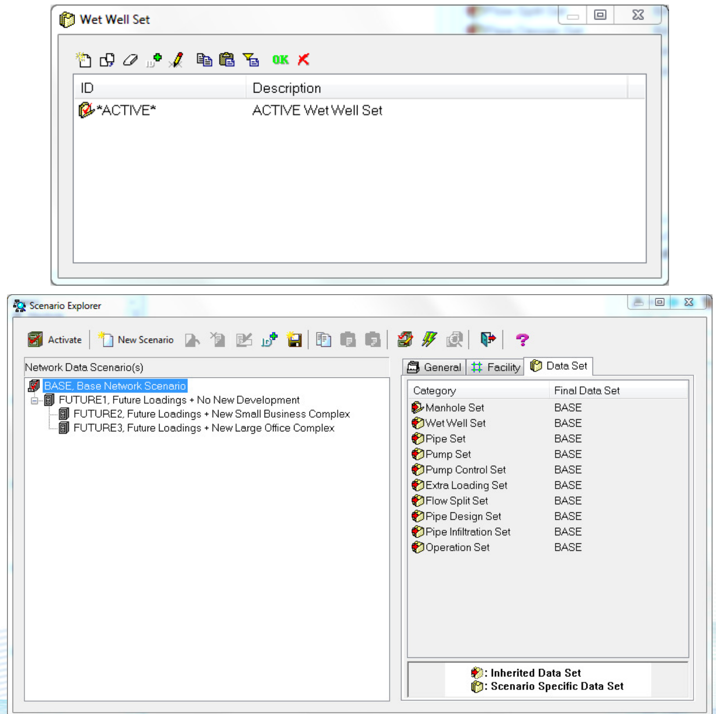


Student Notes:

- **Simulation Option Sets** – Defines simulation options associated with the scenario. There are three different option set types, each storing a logical grouping of simulation options. One option set of each type must be associated with a scenario.
- **Facility set** – Defines which network components are active during a simulation. A facility set may include the entire network or a portion of the network.
- **Data sets** – Stores modeling data (pipe diameter and roughness, manhole loadings, pump curves, operating controls, etc.) associated with each facility. There are ten different data set types, each storing a logical grouping of modeling data. One data set of each type must be associated with a scenario. A data set may be inherited from a parent scenario or may be explicitly defined for a given scenario.
- Simulation results are stored in **Output Sources**:
- **Output Source** – Results for a simulation performed while a given scenario is active are stored in output sources. When the scenario is active, the output source is identified as *ACTIVE*. When the scenario is unloaded (and another scenario activated), that output is immediately renamed to match the name of the scenario with which it is associated. For example, an EPS output source is created (an EPS simulation performed) while a scenario entitled “FUTURE LOADING” is active. When you switch to another scenario, that output source is now referred to as *FUTURE LOADING*:EPS.
- **Compare results between scenarios** – You may display results from two or more scenarios on any graph or report in the Output Report Manager.

Data Sets

- BASE
- Additional Datasets
- Sets include:
  - Manhole Set
  - Wet Well Set
  - Pipe Set
  - Pump Set
  - Pump Control Set
  - Extra Loading Set
  - Flow Split Set
  - Pipe Design Set
  - Pipe Infiltration Set
  - Operation Set



Page 9-5

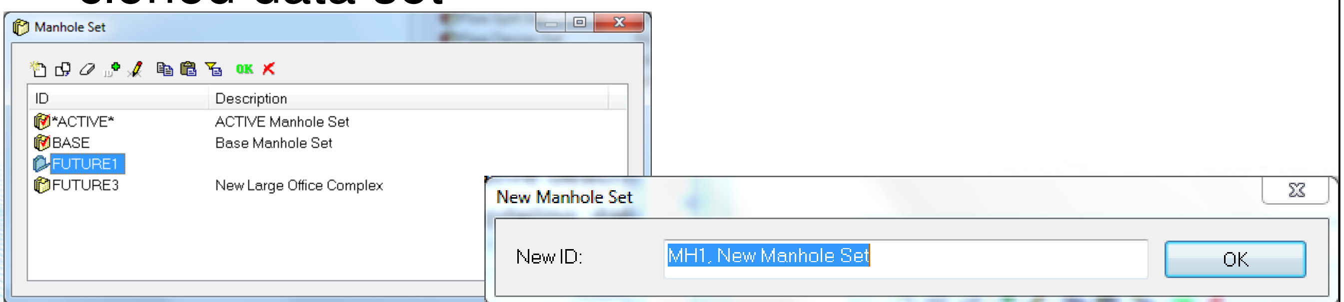
Student Notes:

- **Manhole Sets** – Store manhole loadings and diurnal pattern identifiers.
- **Wet Well Sets** – Store wet well modeling data including type (cylindrical, variable area) and operating parameters.
- **Pipe Sets** – Store pipe modeling data including diameter and Manning roughness coefficients.
- **Pump Sets** – Store pump modeling data including pump curve.
- **Control Sets** – Store pump initial status (open, closed) and operational control rules for pump.
- **Extra Load Sets** – 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 Sets** – 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 Sets** – Design and analysis criteria curves, max and min velocities as well as replacement and duplicate curves assigned by the user to various facilities.
- **Pipe Infiltration Sets** – Store all pipe infiltration data.
- **Operation Sets** – Store all pattern and curve data referenced by other scenario data sets.

You may create any number of data sets of each data set type. Choose the "… Set" commands from the "Scenario" menu or right-click on the data set in the Scenario Manager .

Creating Data Sets

- Modify *ACTIVE* modeling data in current project
- Open desired data set, select **Active**, and choose “Clone”
- Assign unique name and narrative description
- Data copied from *ACTIVE* modeling data to the cloned data set



Student Notes:

Use the Data Set Manager dialog boxes to create and maintain data sets:

- Choose the desired ”Data Set” command from the “Scenario” menu, or
- Choose the “Scenario Manager” command from the “Scenario” menu, choose the “Data Set” tab, and then right-click on the desired data set and choose “Select”.

Creating new data sets – When the desired Data Set Manager appears on the screen, perform the following:

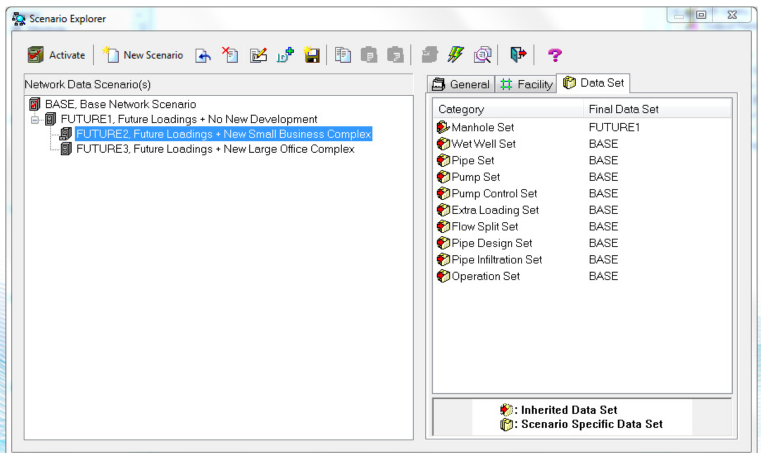
- Modify the *ACTIVE* (currently loaded) modeling data.
- Select the “*Active*” data set and choose the “Clone” button.
- Enter a unique identifier and narrative description for the new data set.
- Press the “OK” button to create the data set.

Data are copied from the *ACTIVE* modeling data (i.e., those data currently loaded and active in the open InfoSewer project) into the new data set. For example, when you create a new manhole set, all baseline loads and diurnal pattern identifiers are copied from the current manhole nodes in the open InfoSewer project to the new manhole set.

Modify existing manhole sets - To modify data in a manhole set, you must modify the *ACTIVE* data and then move those data to the data set. To modify data in an existing data set, load the data set to make its data *ACTIVE*, modify the data values as necessary, then recreate the data set.

Inheritance

- Parent Scenario – Passes properties to children scenario
- Data sets may be inherited from parent or unique to each scenario



Student Notes:

Inheritance refers to the relationship between two or more scenarios where one or more properties of a scenario are dependent or alternately, independent from, another scenario. These dependent scenarios are referred to as **child scenarios** and the scenarios from which properties are inherited are referred to as **parent scenarios**.

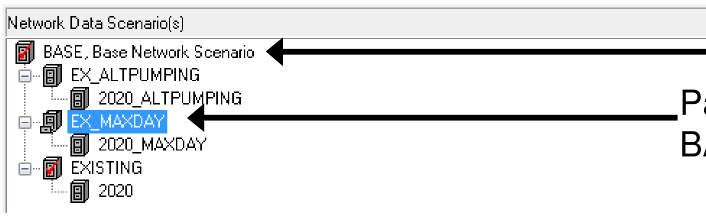
Where a property is inherited from a parent, that inheritance relationship is dynamic. By changing the property of the parent scenario, that same property is reflected in the child. However, where a property is explicitly changed in a child scenario, the inheritance relationship for that property is broken and the change is reflected only in the child.

The inheritance tree – The inheritance tree is a graphical representation of the relationship between the “BASE” scenario and all custom scenarios in a given InfoSewer project. You may change relationships between scenarios by dragging a scenario to the desired parent scenario.

What scenario properties can be inherited – One or more data sets may be inherited from a parent scenario. Option sets and facility sets are not inherited from parent scenarios.

Inheritance

- Changing the parent data set changes the child dataset, ONLY IF they currently share the same data set



Parent of ALL Scenarios

Parent of 2020_MAXDAY, Child of BASE

Datasets of BASE

Category	Final Data Set
Manhole Set	BASE
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

No inheritance for the Demand Data Set

Datasets of EX_MAXDAY

Category	Final Data Set
Manhole Set	EX_MAXDAY
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

Student Notes:

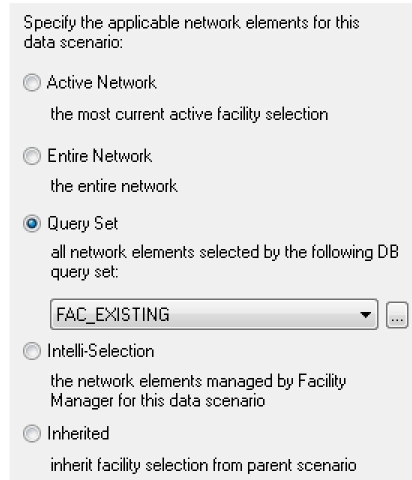
Datasets are the key component of scenarios. Tracking datasets and changes made to them can be difficult. The dataset manager can be used to view all datasets. Datasets are specified using the scenario manager.

The dataset manager is used to view, clone, and create new datasets. However, dataset attributes cannot be modified from the dataset manager. The dataset manager is best used to get a grasp on all the different datasets contained in a model.

Datasets are modified by editing attributes and parameters. The dataset edited depends on the element and property being edited. Many single elements (e.g. one junction) can have a BASE dataset and a non-BASE dataset (Demand Set and Fire Flow Set). Almost every scenario will still contain many BASE datasets, which is why editing BASE datasets from the BASE scenario is highly recommended. A common mistake is editing one non-BASE dataset, then another feature attribute that references a BASE dataset. The user should make an effort to understand exactly which parameters are contained in each dataset.

Facility Sets

- Facility sets define the network facilities to be included in a simulation
  - E.g.: Existing facilities, future facilities, North Service Area, etc.
  - Query Statements
  - Query Sets
  - Selection Sets
  - Intelli-Selection
  - Inherited
- Unlike data sets, facility sets simply list the facilities to be included in a simulation
- Facility sets are activated for simulation



Innovyze®

Student Notes:

A working facility set must:

- Contain at least one loading manhole and one outlet;
- Not contain any disconnected elements; and
- Contain all pump components referenced as part of a pump control rule.

Facility sets may be defined by one or more of the following methods:

- **Query Statement or Query Sets** - Lists of components included in a facility set defined by one or more database query statements.
- **Selection Sets** - Lists of elements included in a facility set defined by graphical selection or by the current domain.
- **Special Queries** - One or more pre-defined queries.
- **Intelli-Selection** - Remembers the *active* facility set from the Facility Manager and will reinstate that facility set every time the scenario is activated.
- **Inherited** – Inherits facility selection from parent scenario

Properties of facility sets – Unlike data sets, which store actual modeling data for different scenarios in database tables, facility sets are simply pointers referencing the identifiers of the components included in each facility set. Facility sets are then activated for simulation. InfoSewer disregards all facilities that are not activated at the time a simulation is performed. Inactive components are removed from the map display until they are re-activated.

Facility Sets Defined by:

- Graphical selection by user
- Query Statements and Query Sets
  - One or more database query statements defining facilities to be included
  - At least one query statement for each InfoSewer feature type
- Selection Set
  - Previously selected set of components, selected via:
    - Graphical Selection
    - Inclusion from the current domain
- Pre-defined special queries
- Network

Page 9-10

Innovyze®

Student Notes:

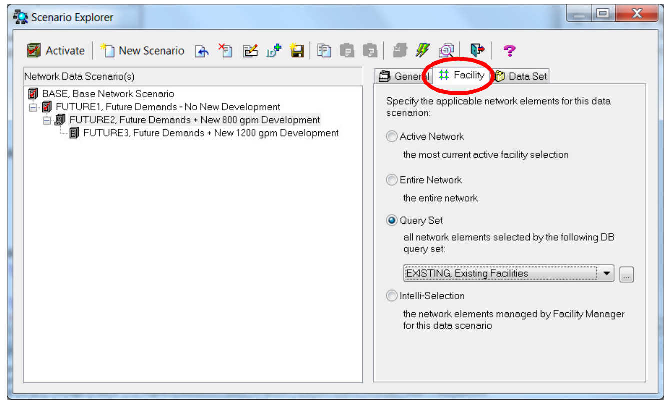
The following methods can be used to develop a facility set:

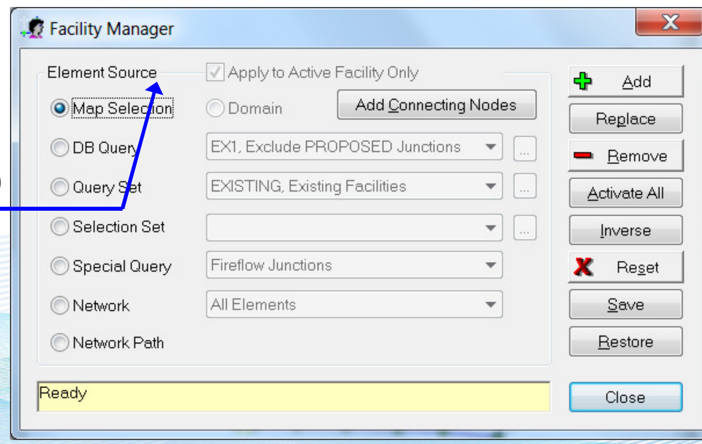
- **Graphical Selection** - You can include all components in the current domain, or graphically select components "on the fly" for inclusion in a facility set.
- **Query Statements and Query Sets** - A facility set can be defined by a single logical query statement or a Query Set. Define one or more database query statements used to select network components for inclusion in a query set. The query set normally contains at least four database query statements to define the links to be included in the set; one for pipes (force and gravity), one for pumps, one for wet wells and one for manhole nodes. A query set could be used to select network component by their properties such as pipe diameter.
- **Selection Sets** - A previously-saved set of network components, defined by inclusion in the domain or via graphical selection by the user. A selection set could be used to keep a permanent record of an area to be included in a scenario.
- **Pre-defined Queries** - InfoSewer offers several pre-defined queries for easy inclusion of facilities in a facility set.
- **Network** - Include all network components, or alternately, all components of a single component type, in the facility set.

Saving facility sets for future activation – Unlike data sets, only one facility set can be active at a time, and facility sets cannot be stored for future loading. To develop a facility set for easy re-activation in the future, develop one or more query sets and/or selection sets to store the criteria by which facilities are activated.

Facility Management

- Facility – A group a features to be used during the simulation.
- Created with:
  - User Selection
  - Queries
  - Selection Sets
- May be constrained to the active facility





Student Notes:

A facility is a collection of network components that may be activated or deactivated for a simulation. Facilities are created by selection or queries. Facilities **impact** the model operation and simulation.

Important notes about facilities:

- Active facilities will be modeled during a simulation.
- Deactivated facilities will **NOT** be modeled during a simulation.
- A model simulation **IS** restricted to elements in the active facility.
- Facility Queries and Selections may be confined to active facilities.
- Deactivated elements are displayed in gray by default.

The scenario manager best utilizes the power of facility management.

Facility Tools

- Facility Manager
- Add to Facility
- Reduce Facility
- Reset Facility
- Activate All
- Activate Domain

Innovyze®

Student Notes:

Facility Manager – Interface for activating and deactivating facilities.

Add to Facility – Adds selected objects to the facility.

Reduce Facility – Removes objects from the facility.

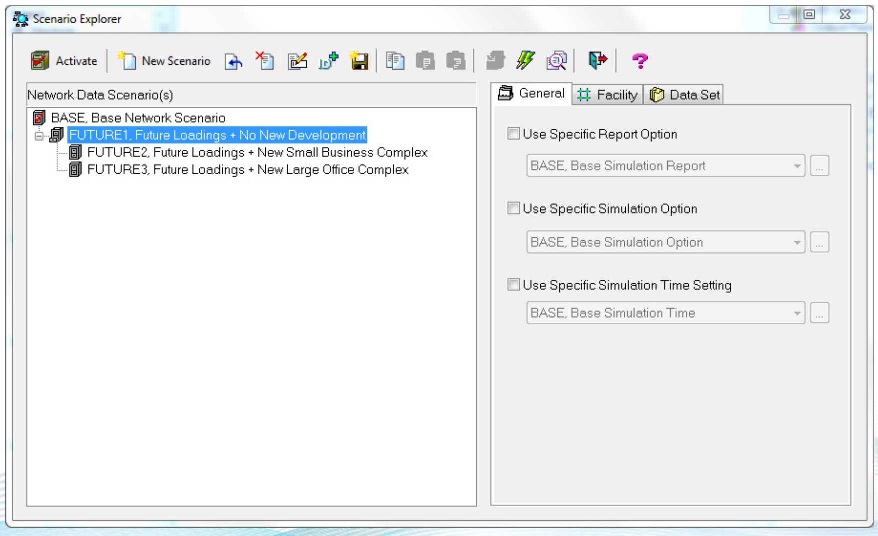
Reset Facility – Resets the facility to the scenario default.

Activate All – Activates all network features.

Activate Domain – Activates all features from a domain.

Scenario Options and Settingss

- Report Options
- Simulation Options
- Time Settings



Student Notes:

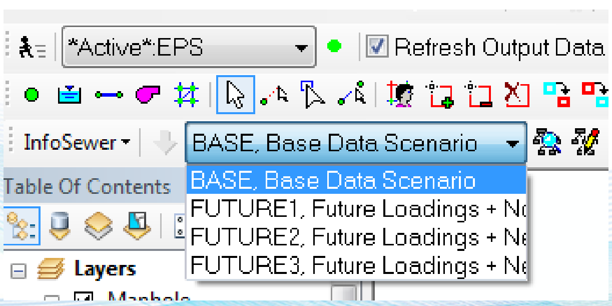
Each scenario has three option sets:

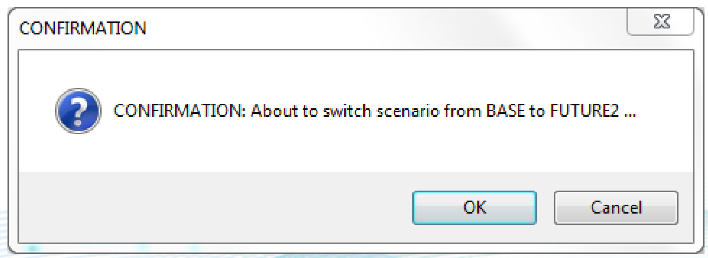
- **Report Options** – Defines standard InfoSewer report options. When you run a simulation, you may choose to have InfoSewer generate, or not generate, a report of simulation results. Those options are stored in Report option sets and associated with each InfoSewer scenario. You may create and maintain report options by choosing the corresponding "**…**" button on the Scenario Manager. Useful for creating custom reports of any scenario. This can be set in conjunction with a unique reporting time step to produce the desired reports.
- **Simulation Options** – Defines standard simulation options including units of measurement, minimum travel distance, or diurnal pattern usage. Other options include peaking factor equations and global criteria and cost curves. Those options are stored in Simulation option sets and associated with each InfoSewer scenario. You may create and maintain simulation options by choosing the corresponding "**…**" button on the Scenario Manager. Useful for comparing simulation options.
- **Time Setting Options –** Defines the duration of each simulation and time steps for extended period simulations. Those options are stored in time sets and associated with each InfoSewer scenario**.** Useful for comparing hydraulic time steps.

You may create and maintain time options by choosing the corresponding "**…**" button on the Scenario Manager.

Activating a Scenario

- Unsaved changes to *ACTIVE* modeling data are automatically saved
- Facility set definition is applied
- Data an option sets are loaded
  - NOTE: *ACTIVE* modeling data currently in the project will be over-written by modeling data in the scenario data sets





Student Notes:

To activate a scenario, perform the following:

- Choose “Scenario Manager” from the InfoSewer Control Center -> InfoSewer button -> Scenario menu;
- Choose the scenario to activate; and
- Select the “Activate” button.

Or alternately: Choose the desired scenario from the drop-down list on the InfoSewer toolbar.

What happens when I load a scenario?

- Any unsaved changes to the *ACTIVE* (currently loaded) data sets are saved to those data sets.
- The facility set definition for the scenario being loaded is evaluated and applied, activating those facilities meeting the scenario’s facility set criteria (discussed after the first exercise)
- Replaces the *ACTIVE* modeling data with data in the data sets associated with the scenario being loaded.

You may now use any other InfoSewer feature to modify the newly-loaded modeling data, may run a scenario to create an output source for the active scenario, and may use any display and analysis tool to review simulation results.

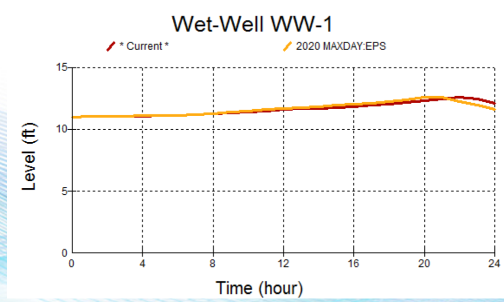
Batch Simulations / Comparing Results

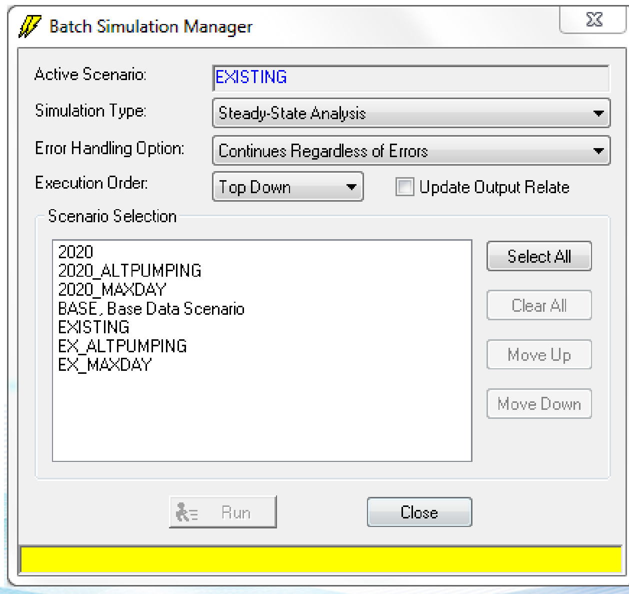
- Run multiple simulations at once
- Update Output Relates
- Graph multiple scenario results on a single graph
- Compare Reports

Scenario Explorer

Tools Menu

Batch Simulation...





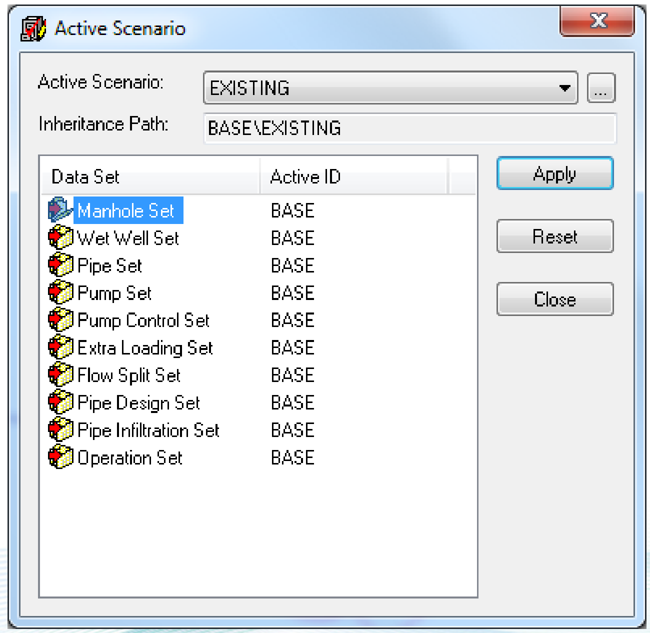
Student Notes:

A batch simulation simply runs simulations of multiple scenarios and makes all output files active. The scenarios to be simulated are selected from the batch simulation dialog. The results of all scenarios simulated in a batch may be compared in reports and/or graphs.

When a batch simulation is performed and the model is saved, closed, and reopened, only the output from the current scenario when the model was closed will be retained for reporting, graphing, or querying. Group graphs cannot be used to compare scenarios.

Editing an Active Scenario

- Special tool required
- Cannot edit BASE scenario
- Only Datasets may be changed
  – Not facilities
  – Not options or settings



Innovyze®

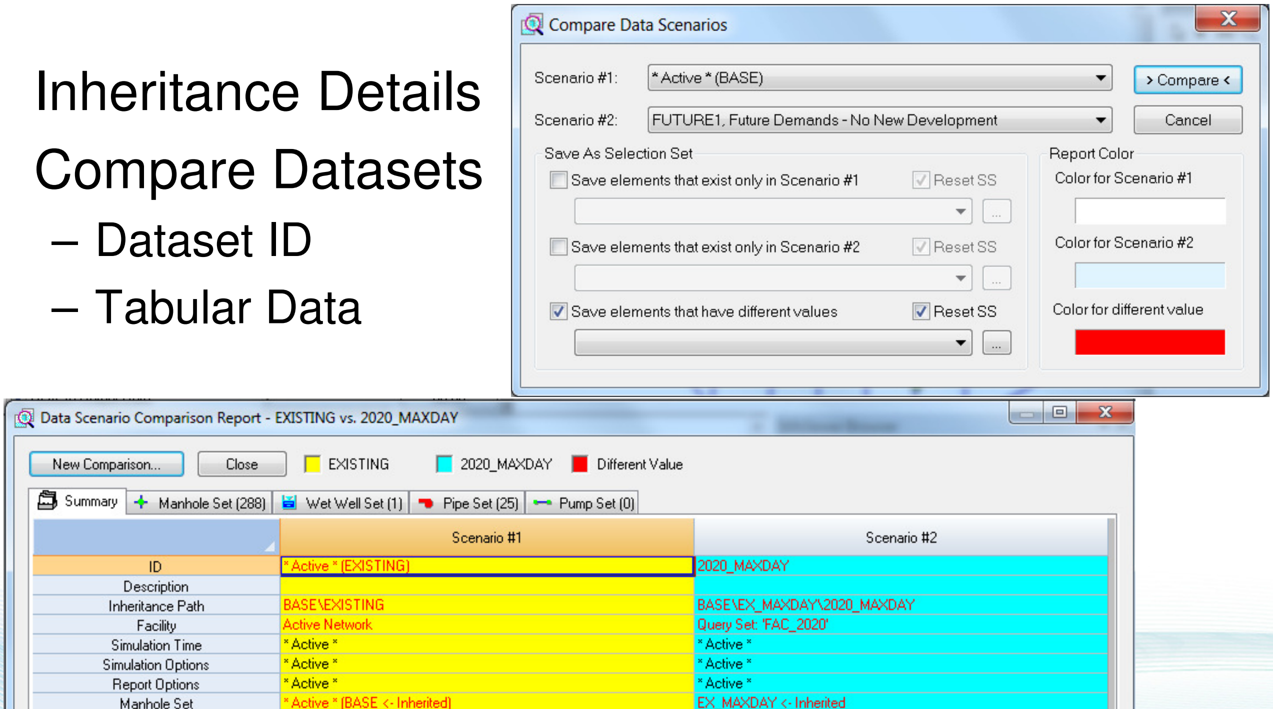
Student Notes:

The active scenario can only be edited using this special feature. The BASE scenario cannot be edited using this feature. Only datasets of other scenarios may be changed; facility and options may not be changed with this feature.

The primary purpose of this feature is to allow the user to create a new dataset in a scenario when the user realized one is needed to replace the dataset without needing to first change scenarios.

Comparing Scenarios

- Inheritance Details
- Compare Datasets
  - Dataset ID
  - Tabular Data



Student Notes:

The compare scenarios tool details differences between two scenarios. This tool will detail the inheritance of all datasets for each scenario. In addition, values of individual datasets may be compared side by side.

Summary - Details Inheritance and datasets

Demand Set - A tabular comparison of demand data from the two scenarios.

Tank Set - A tabular comparison of tank data from the two scenarios.

Reservoir Set - A tabular comparison of reservoir data from the two scenarios.

Pump Set - A tabular comparison of pump data from the two scenarios.

Valve Set - A tabular comparison of valve data from the two scenarios.

Pipe Set - A tabular comparison of pipe data from the two scenarios.

Chapter 10

Storm Modeling

Student Notes:

Storm Modeling In InfoSewer

- InfoSewer models:
  - Sanitary sewer systems
  - Storm sewer systems
  - Combined sewer systems
- Rational Method used for Steady State simulations
- Unit Hydrograph method used for dynamic simulations
- Unit Hydrographs can be natural or synthetic

Student Notes:

InfoSewer 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 systems, storm sewer systems, and combined 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).

Rational Method

For estimation of peak flow during steady state simulations, InfoSewer utilizes the Rational Method, which is a very popular tool of choice used by many practicing engineers across the globe.

Unit Hydrograph

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, and can be calibrated automatically to match observed data using powerful genetic algorithm optimization technology.

Rational Method

- Widely used to estimate peak storm load
- Peak runoff is proportional to rainfall intensity and basin area
- Runoff coefficient depends on land use, soil type, and slope
- Time of Concentration is internally estimated
- Design/Analysis rainfall intensity is derived from IDF curve
- The method has some basic assumptions

Student Notes:

The concept of the Rational Method is 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, and then the peak flow rate will be proportional to the product of rainfall intensity and watershed area.

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.

Time of Concentration – Time of Concentration 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.

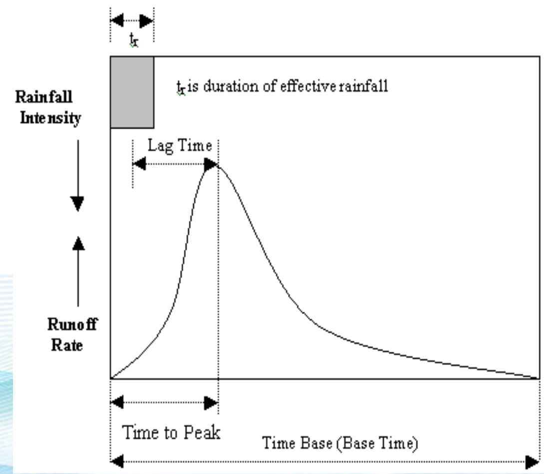
IDF Curve – The model derives rainfall intensity from the IDF curve corresponding to the duration equal to Time of Concentration calculated for the manhole following the techniques described above.

Assumptions – 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

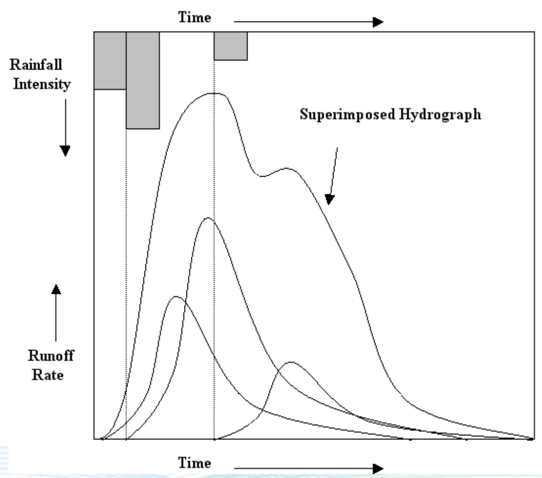
- Runoff resulting from a unit depth of effective rainfall produced within a specific duration
- Duration is an important parameter
- The theory relies on assumptions
  - Linearity
  - Time invariance
  - Principle of superposition



Student Notes:

Unit Hydrograph

- The figure demonstrates the principle of superposition
- Multiple unit hydrographs are handled
- No limitations on number of rainfall events
- Multiple hyetographs are handled
- Unit hydrographs can be natural or synthetic



Innovyze®

Student Notes:

Excess rainfall resulting from single or multiple storm events is discretized at intervals of unit hydrograph duration. 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.

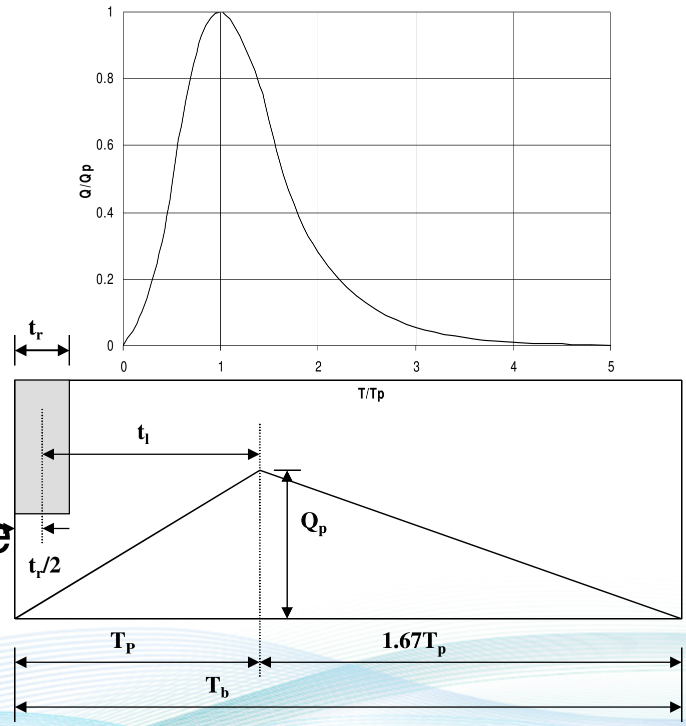
Multiple Unit Hydrographs – An entire watershed of a collection system may not be well represented by a single unit hydrograph owing to variability in topography, land use, and soil characteristics of the subwatersheds. Accordingly, InfoSewer allows usage of multiple unit hydrographs, each representing part of the watershed being modeled.

Multiple Hyetographs and Multiple Rainfall Events – InfoSewer accepts rainfall data in the form hyetograph, which is a plot of rainfall intensity against time. The model has absolutely no limitation, what so ever, regarding the number of rainfall events to be modeled as well as the number of hyetographs (rain gauge stations in the watershed) to be used.

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.

Synthetic Unit Hydrograph

- Three options are offered to synthesize a unit hydrograph
  - SCS dimensionless
  - SCS triangular
  - Tri-triangular
- Parameters of the SCS hydrographs are derived from watershed characteristics



Innovyze®

Student Notes:

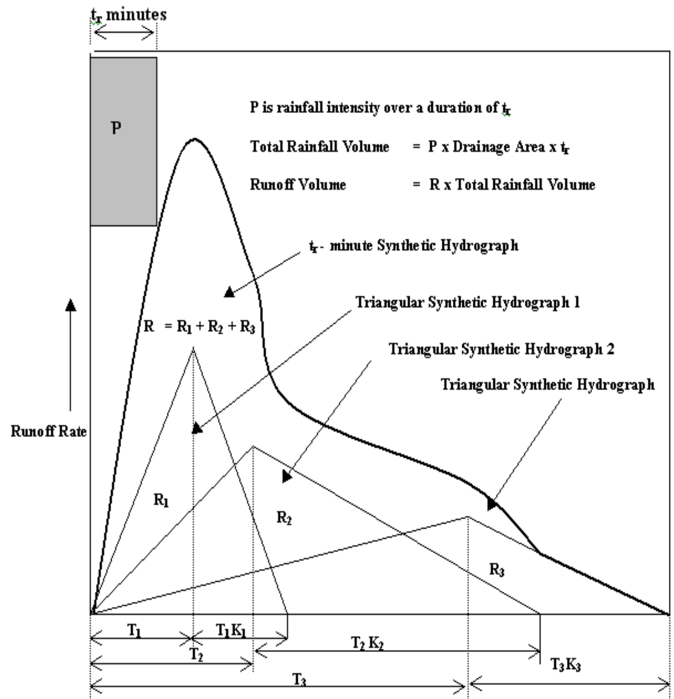
If a natural hydrograph is available for a watershed, InfoSewer can utilize it to generate storm runoff at different locations in the collection system. In the absence of a natural unit hydrograph, the model can derive a synthetic unit hydrograph using three different methods.

SCS Dimensionless Unit Hydrograph – The SCS dimensionless unit hydrograph is widely used in practice. To generate a t_r-hour unit hydrograph for a watershed, time to peak (T_p) and the peak flow rate (Q_p) are determined using watershed characteristics. Once the parameters are known, actual time and flow rate ordinates of the 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.

SCS Triangular Unit Hydrograph – The SCS has also developed a triangular unit hydrograph that is an approximation to the dimensionless unit hydrograph. The triangular unit hydrograph is entirely defined in terms of three points, Q_p, T_p, and T_b. These parameters are calculated using the same equations as for the dimensionless unit hydrograph.

Tri-Triangle Method

- The synthetic hydrograph is obtained by superimposing the triangles
- Each of the triangles are defined in terms of three parameters
- The parameters can be calibrated

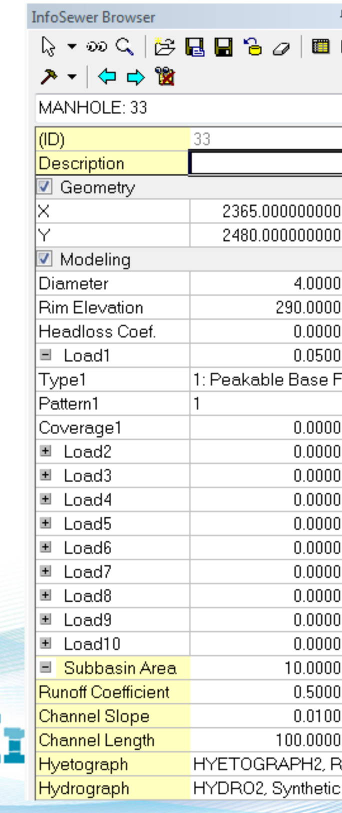


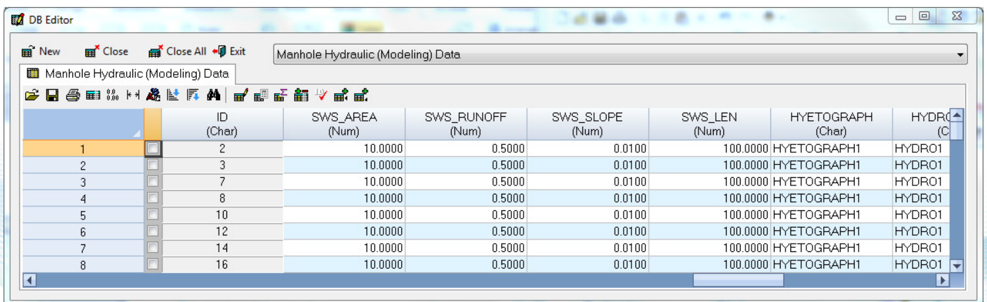
Student Notes:

The shape of unit hydrograph is too complex to be well captured by a single triangle, such as the NRCS's triangular unit hydrograph. InfoSewer offers the option to use up to three triangular synthetic hydrographs 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.

Building Steady-State/Design Simulation Data

- Specify subwatershed data
- Area, Runoff Coefficient, Average Slope and Length



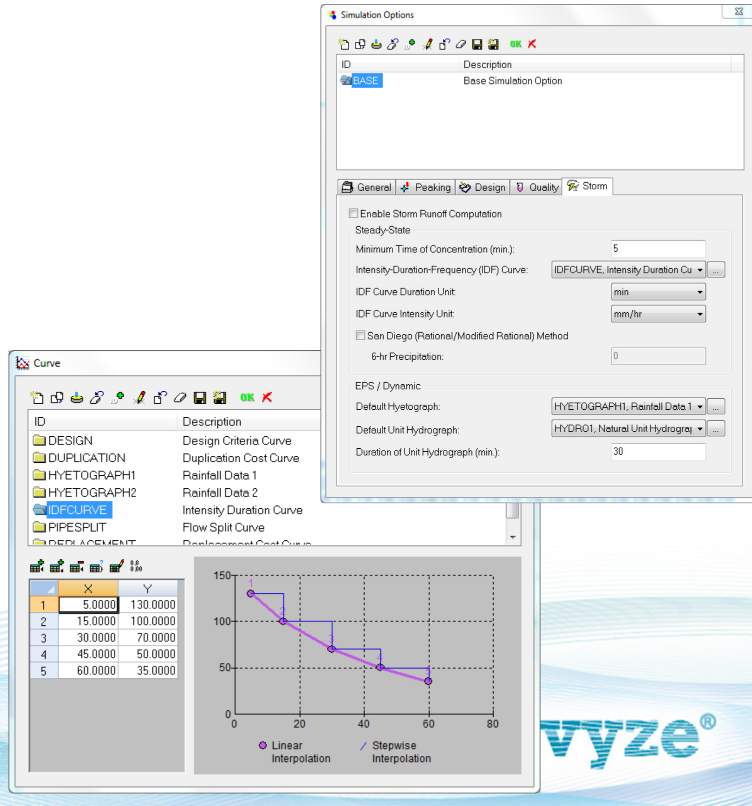


Student Notes:

InfoSewer uses the Rational Method during steady-state and design simulations of stormwater. For each loading manhole, application of the Rational Method requires area of the subwatershed that drains to the manhole, average slope of the subwatershed, length of the longest flow path in the subwatershed, and runoff coefficient of the subwatershed. These data can be entered in the Attribute tab of the InfoSewer Browser window, one manhole at a time, or globally using the DB Editor dialog box.

Building Steady-State/Design Simulation Data

- Build an Intensity-Duration-Frequency (IDF) curve
- Specify intensity and duration units for the IDF curve
- Specify Minimum Time of Concentration



Student Notes:

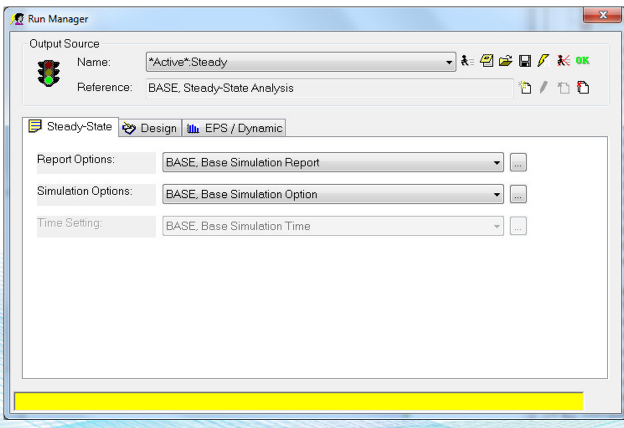
In addition to the subwatershed data described in the previous slide, steady state simulation of stormwater requires an Intensity-Duration-Frequency curve, commonly named as an IDF curve. At each loading manhole, the model uses the IDF curve to extract design storm intensity corresponding to the time of concentration (duration) internally calculated for the manhole based on the subwatershed data. InfoSewer accepts IDF curve in the form of a curve where the X axis represents duration and the Y axis represents rainfall intensity.

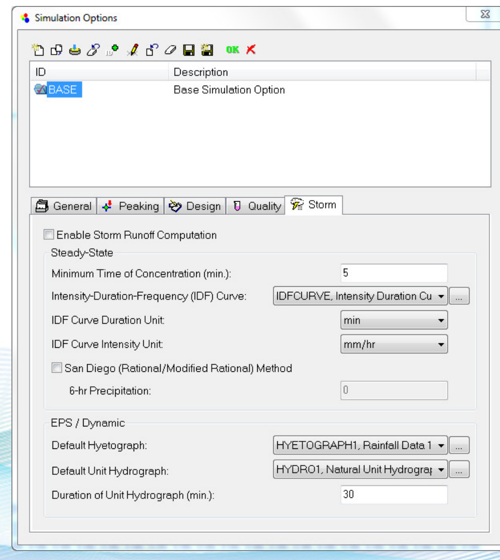
Duration unit for the IDF curve can be either in minute, second, or hour. Rainfall intensity can be in inches/hour, mm/hour, or in mm/minute. The user has to specify the units used for the duration and the rainfall intensity.

In order to avoid unreasonably low storm duration and unreasonably high rainfall intensities, InfoSewer allows the user to specify a minimum time of concentration (rainfall duration), 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 is 10 minutes.

Running Steady-State Simulation

- Enable storm runoff computation
- Run steady-state simulation
- Review storm results for gravity pipes and loading nodes





Student Notes:

Once the data required to execute steady-state/design simulations is provided, perform the following to run a simulation and to review results.

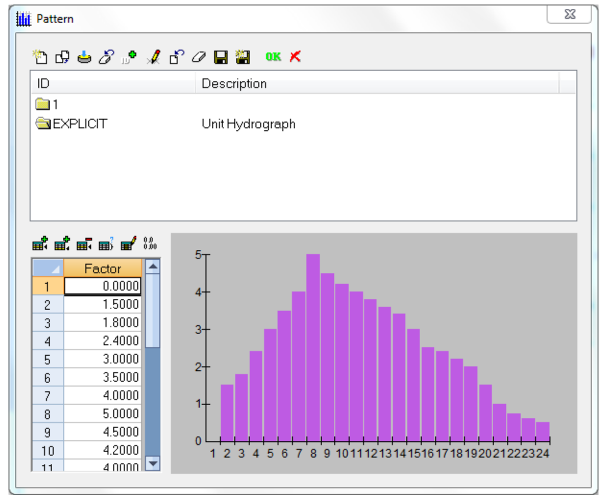
Activate storm runoff computation by checking on the Enable Storm Runoff Computation option from the Storm tab of the Simulation Options dialog box.

Select the Steady - State tab from the Run Manager dialog box, and click on the “Run” button.

Upon successful completion of the simulation, you may review the analysis results using the InfoSewer Browser window, for one network element at a time.

Building Dynamic Simulation Data

- Create a Unit Hydrograph
  - Can be natural or synthetic
- Natural unit hydrographs are accepted in the form of a pattern
  - Create Pattern
  - Define Pattern as an explicit unit hydrograph
- Assign drainage area



Innovyze®

Student Notes:

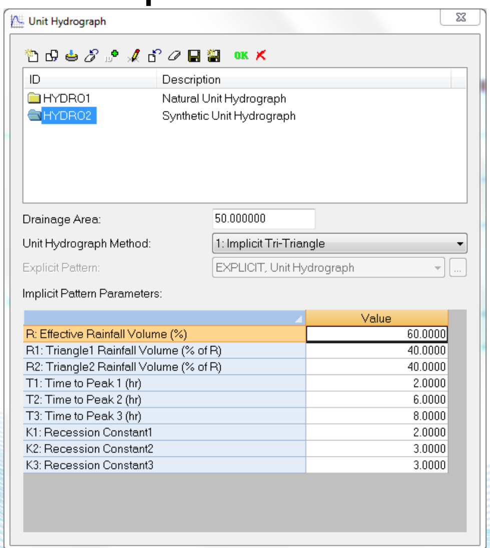
To derive a storm hydrograph, InfoSewer uses the Unit Hydrograph Theory. If a natural unit hydrograph (a unit hydrograph created from observed field data) is available for the watershed being investigated, the user may provide it in the form of a pattern. If a natural unit hydrograph does not exist, InfoSewer can generate a synthetic unit hydrograph using three different techniques (i.e., SCS dimensionless unit hydrograph, SCS triangular unit hydrograph, or the tri-triangle method). SCS stands for Soil Conservation Service.

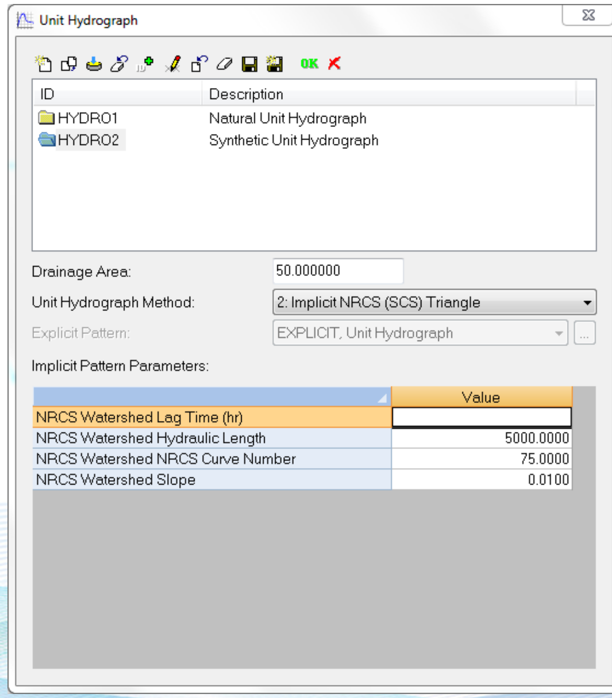
For natural unit hydrographs, once created, the runoff pattern needs to be defined as an explicit unit hydrograph. This should be done using the “Unit Hydrograph” icon from the Control Center.

- Right click on the Unit Hydrograph icon, and select the “New” command.
- Assign the desired name and optional description for the new unit hydrograph, and then click on the “OK” button.
- Select “0:Explicit” for the “Unit Hydrograph Pattern Type” option.
- Choose the pattern in the “Explicit Pattern” drop down box.
- Specify the area of the watershed draining to the location where the unit hydrograph (runoff pattern) is derived.

Creating Synthetic Unit Hydrograph

- If implicit Unit Hydrographs are used, specify their parameters





Student Notes:

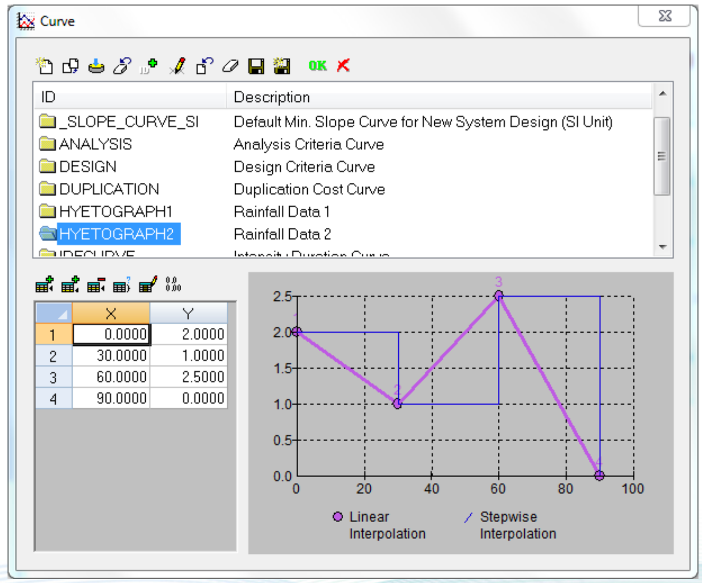
In the absence of natural unit hydrograph, InfoSewer offers three widely used methods to synthesize unit hydrograph. The model relies on user supplied parameters to generate the unit hydrograph internally.

For the SCS dimensionless and the SCS triangular unit hydrographs, the modeler needs to specify either the lag time, or the watershed characteristics (i.e., hydraulic length, SCS curve number, and average slope). If the lag time is provided, the model does not require the latter three parameters.

The tri-triangular method requires the definition of nine parameters. The default values may be used as a starting point. If measured data are available for the watershed, the model can calibrate the parameters of the triangles using genetic algorithms optimization technology.

Creating Hyetograph

- Hyetographs are accepted in the form of a curve
- Multiple hyetographs can be handled
- X-axis column (time) must be in minutes
- Y-axis column (intensity) is either in US units, or in SI units



Page 10-13

Student Notes:

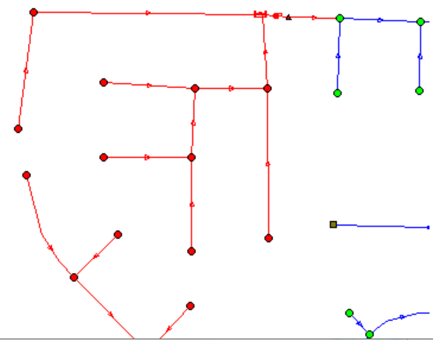
The objective of any stormwater modeling is to estimate the runoff resulting from known rainfall pattern. Therefore, rainfall is the driving force of stormwater modeling. Rainfall is commonly measured at rain gauge stations.

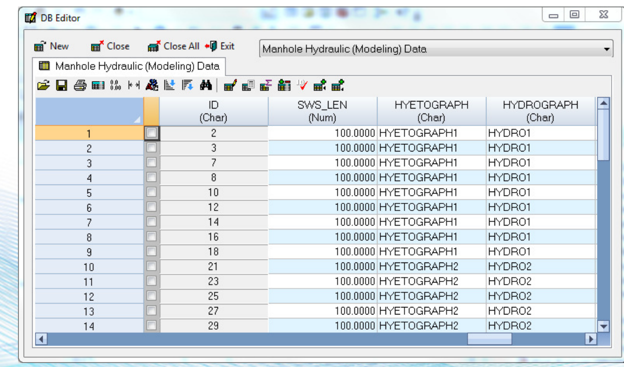
A watershed may have more than one rain gage station to capture spatial variability of a rainfall across the watershed. InfoSewer has the capability to accept more than one rainfall pattern, (i.e., data from multiple rain gages), to model a sewer collection system.

The model accepts rainfall data in the form of a curve. This temporal distribution of rainfall, commonly known as hyetograph, provides rainfall intensity (in inches/hr or mm/hr) versus rainfall duration (in minutes).

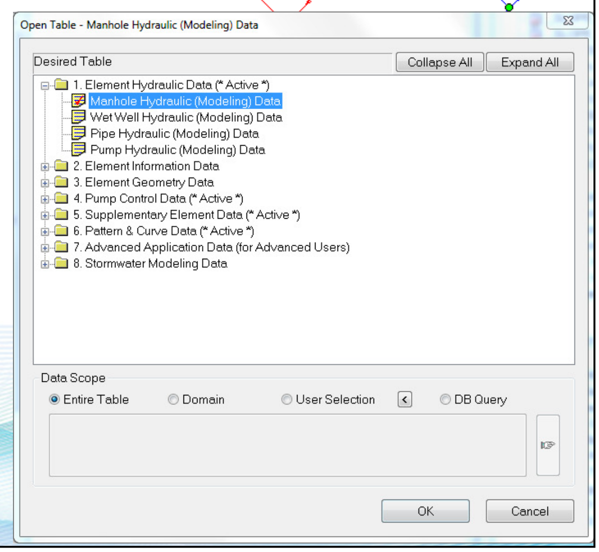
Assign Data to Loading Manholes

- Assign unit hydrograph and hyetograph to loading manholes
- Use domain if multiple unit hydrographs and/or hyetographs are available





		ID (Char)	SWS_LEN (Num)	HYETOGRAPH (Char)	HYDROGRAPH (Char)
1		2	100.0000	HYETOGRAPH1	HYDRO1
2		3	100.0000	HYETOGRAPH1	HYDRO1
3		7	100.0000	HYETOGRAPH1	HYDRO1
4		8	100.0000	HYETOGRAPH1	HYDRO1
5		10	100.0000	HYETOGRAPH1	HYDRO1
6		12	100.0000	HYETOGRAPH1	HYDRO1
7		14	100.0000	HYETOGRAPH1	HYDRO1
8		16	100.0000	HYETOGRAPH1	HYDRO1
9		18	100.0000	HYETOGRAPH1	HYDRO1
10		21	100.0000	HYETOGRAPH2	HYDRO2
11		23	100.0000	HYETOGRAPH2	HYDRO2
12		25	100.0000	HYETOGRAPH2	HYDRO2
13		27	100.0000	HYETOGRAPH2	HYDRO2
14		29	100.0000	HYETOGRAPH2	HYDRO2



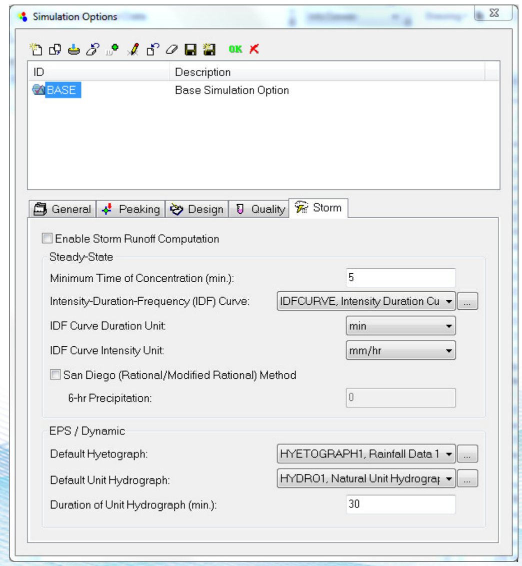
Student Notes:

Once created, the hydrograph(s) and the hyetograph(s) have to be assigned to loading manholes. The assignment can be done using the InfoSewer Browser, for one manhole at a time, or globally using the DB Editor.

InfoSewer allows usage of multiple unit hydrographs and multiple hyetographs to model a watershed. If multiple unit hydrographs and/or hyetographs are available, the domain may be used to assign the hyetographs and the hydrographs.

Default Data and Unit Hydrograph Duration

- Specify default hyetograph, default hydrograph, and duration of unit hydrograph in the Simulation Options dialog box



Innovyze®

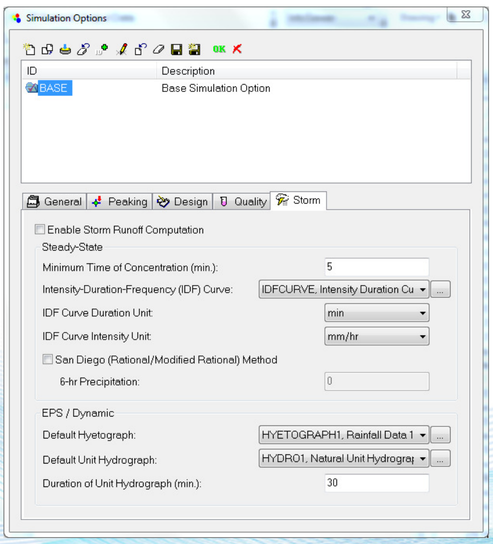
Student Notes:

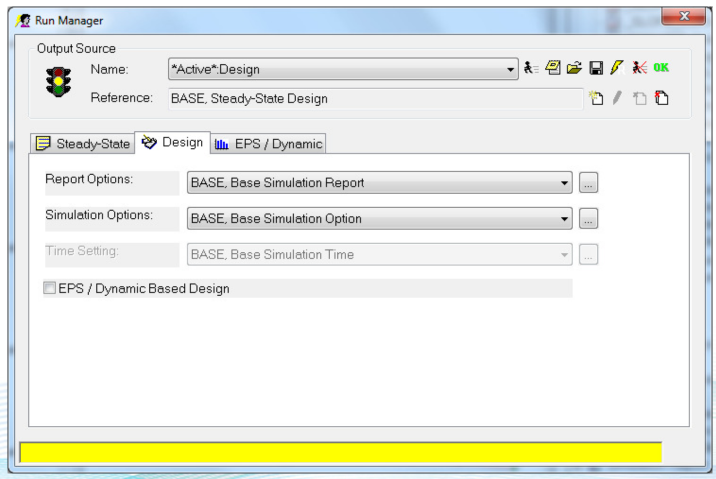
If no hyetograph and/or unit hydrograph are/is assigned to certain loading manholes in the collection system, the model uses default hyetograph and/or default unit hydrograph data for the manholes lacking the respective data.

The default hyetograph and the default unit hydrograph, along with the duration of the unit hydrograph, should be specified using the Storm tab of the Simulation Options dialog box.

Running Dynamic Simulation

- Enable storm runoff computation
- Run dynamic simulation
- Review storm results for load nodes





Student Notes:

Perform the following to run dynamic simulation and to review storm runoff results.

Activate storm runoff computation by checking on the “Enable Storm Runoff Computation” option from the Storm tab of the Simulation Options dialog box.

Select the EPS/Dynamic tab from the Run Manager dialog box, and click on the “Run” button.

Upon successful completion of the simulation, you may review the analysis results using the InfoSewer Browser window or using the output report manager

Chapter 11

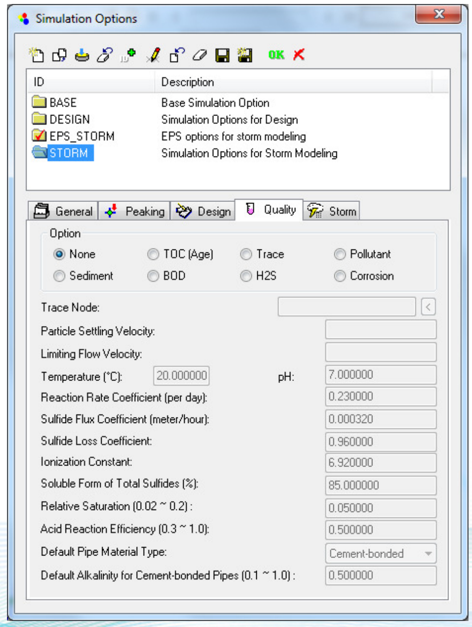
Water Quality Modeling

Innovyze®

Student Notes:

Water Quality Options

- Only Available for EPS/Dynamic Runs
- Types of Analysis:
  - TOC (Age)
  - Trace
  - Pollutant
  - Sediment
  - BOD
  - H2S
  - Corrosion

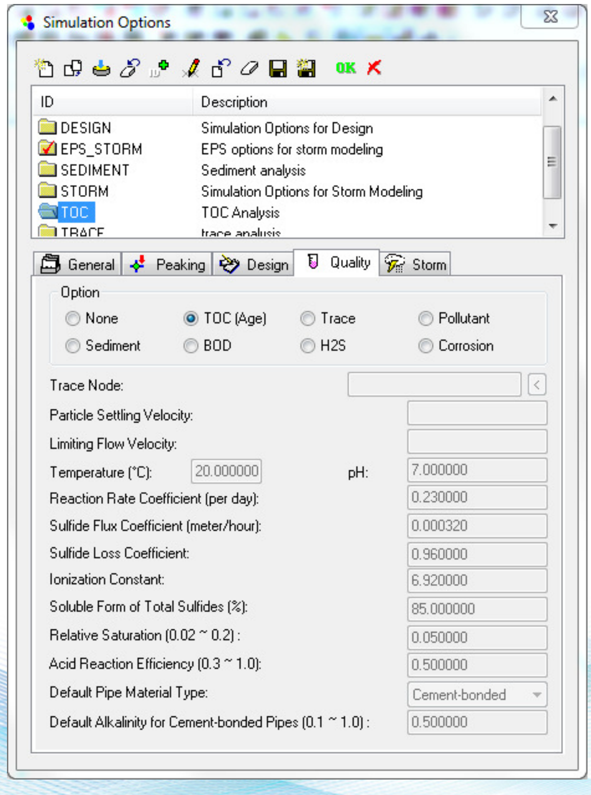


Innovyze®

Student Notes:

InfoSewer provides seven options for sewage quality analysis in conjunction with a hydraulic simulation: TOC (Age), Trace, Pollutant, Sediment, BOD, H2S, and Corrosion. 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 modeling is only available for EPS/Dynamic runs, and may be chosen on the Quality panel from the Simulation Options dialog box.

Time of Concentration (Age)



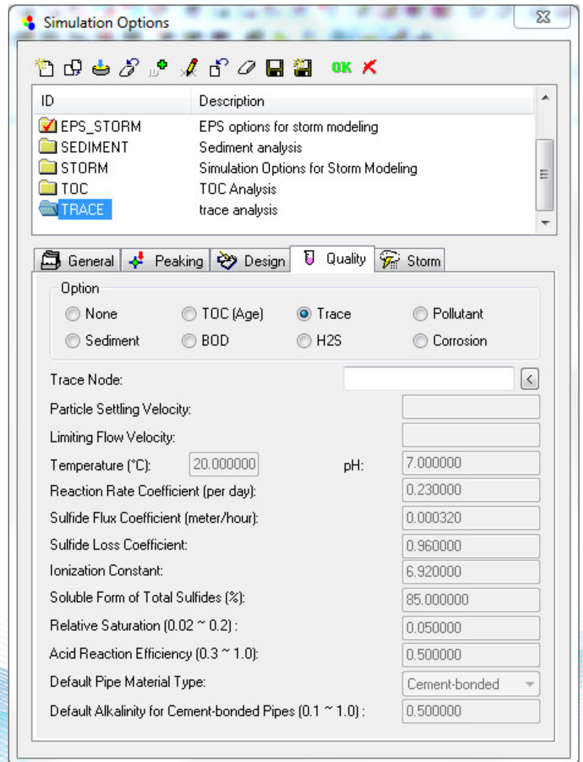
Innovyze®

Student Notes:

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.

No additional input parameter is required.

Source Tracing



Page 11-4

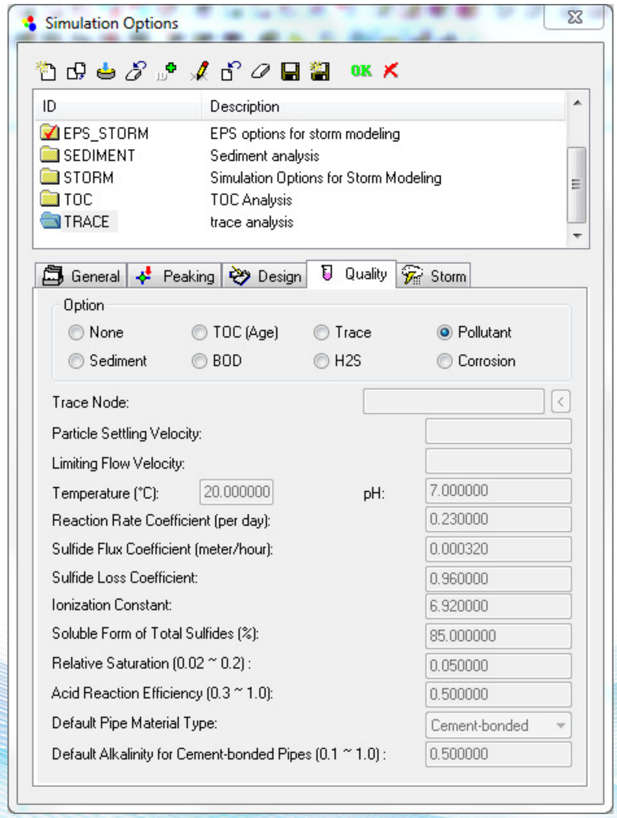
Innovyze®

Student Notes:

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 option in the quality tab of the Simulation Options and then specifying a 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



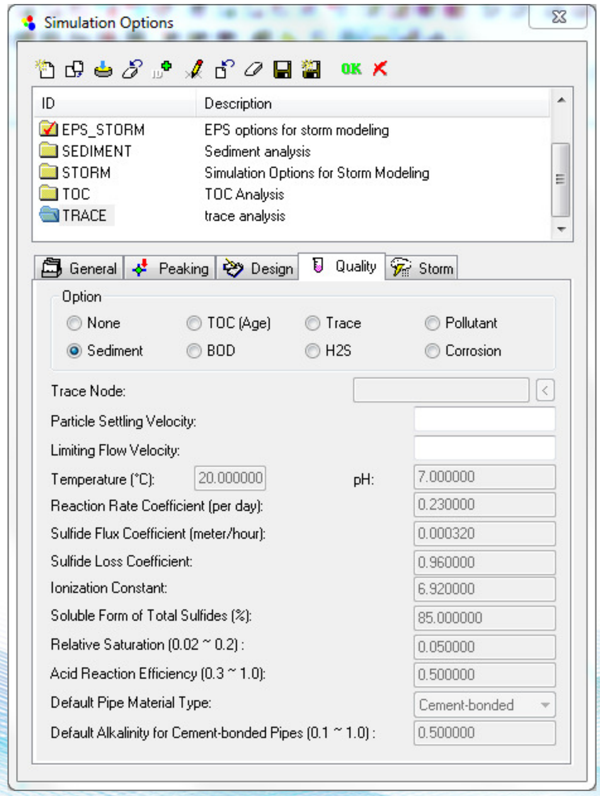
Student Notes:

InfoSewer can 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 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) must be specified.

Sediment



Student Notes:

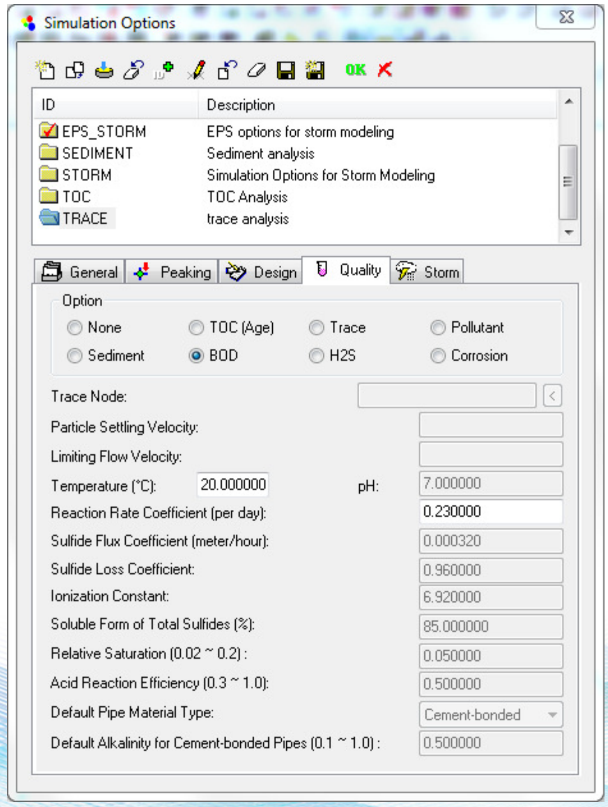
Sanitary sewer systems can carry substantial loads of suspended solids which can accumulate and cause blockages thereby impairing the hydraulic capacity of the sewer pipes. InfoSewer can simulate the transport and gravitational settling of sediments (total suspended solids including grit) over time throughout 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.

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.

BOD



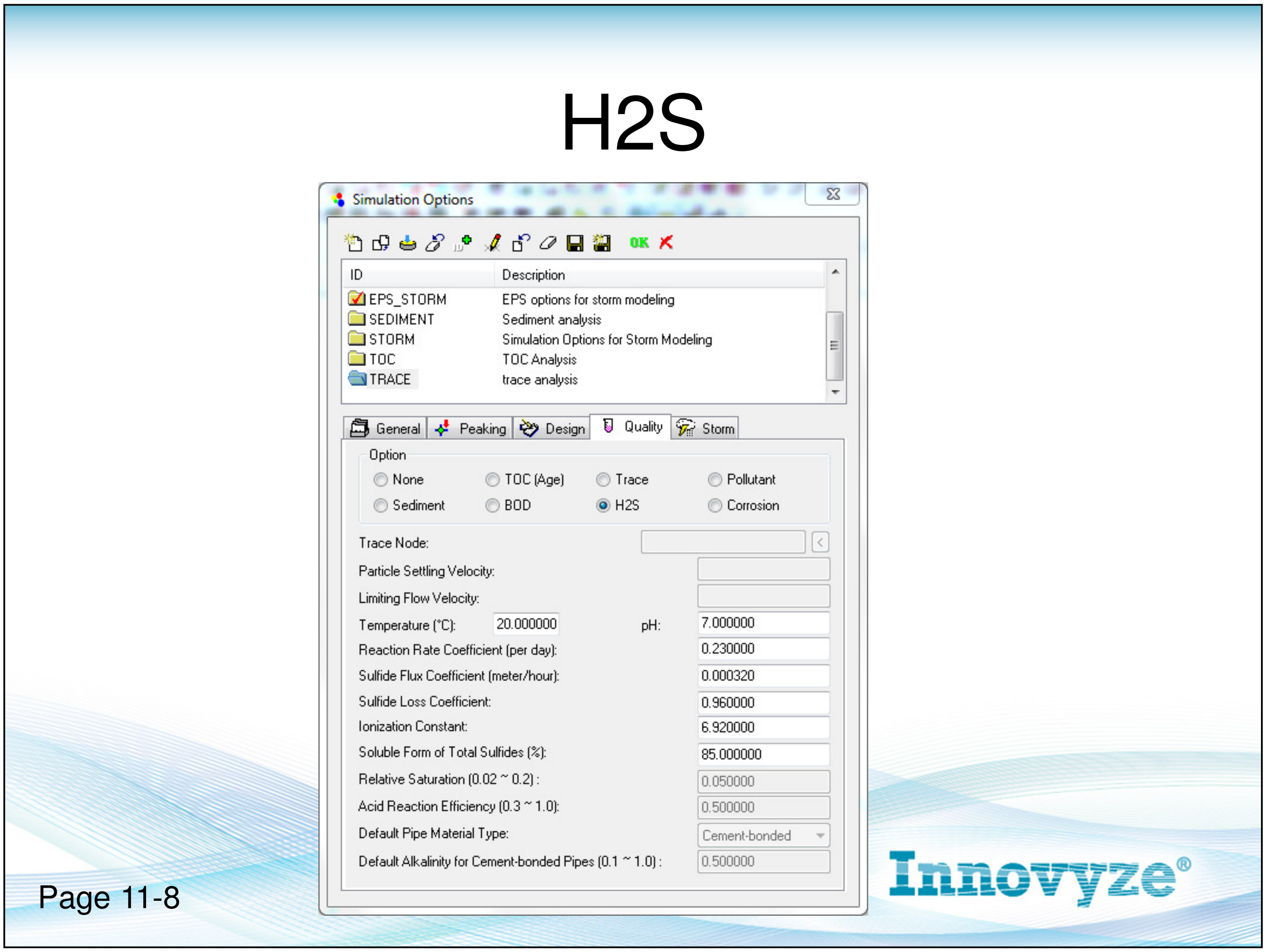
Student Notes:

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, InfoSewer determines 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, as well as source nodes and their corresponding UBOD . 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.

Source nodes can only be loading manholes and/or wet wells during source tracing analysis, and modeling of pollutant transport, sediment transport, and BOD.

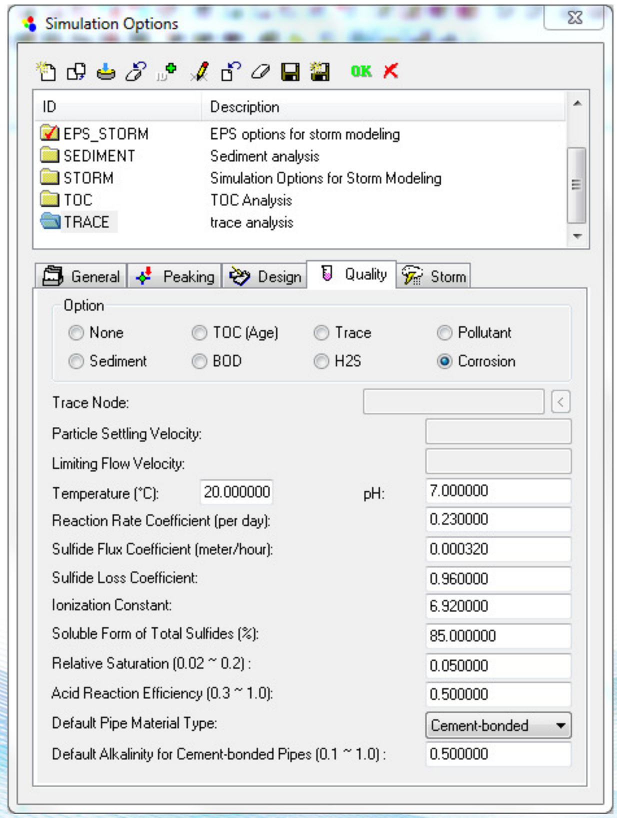


Student Notes:

Modeling of hydrogen sulfide using *H_2S Detector* requires minimal input data from the user. The required data include:

- average daily temperature for the region (in °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°C and specific electrical conductance of 50, 000 micromhos/cm) to 7.74 (at a temperature of 10°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.

Corrosion



Innovyze®

Student Notes:

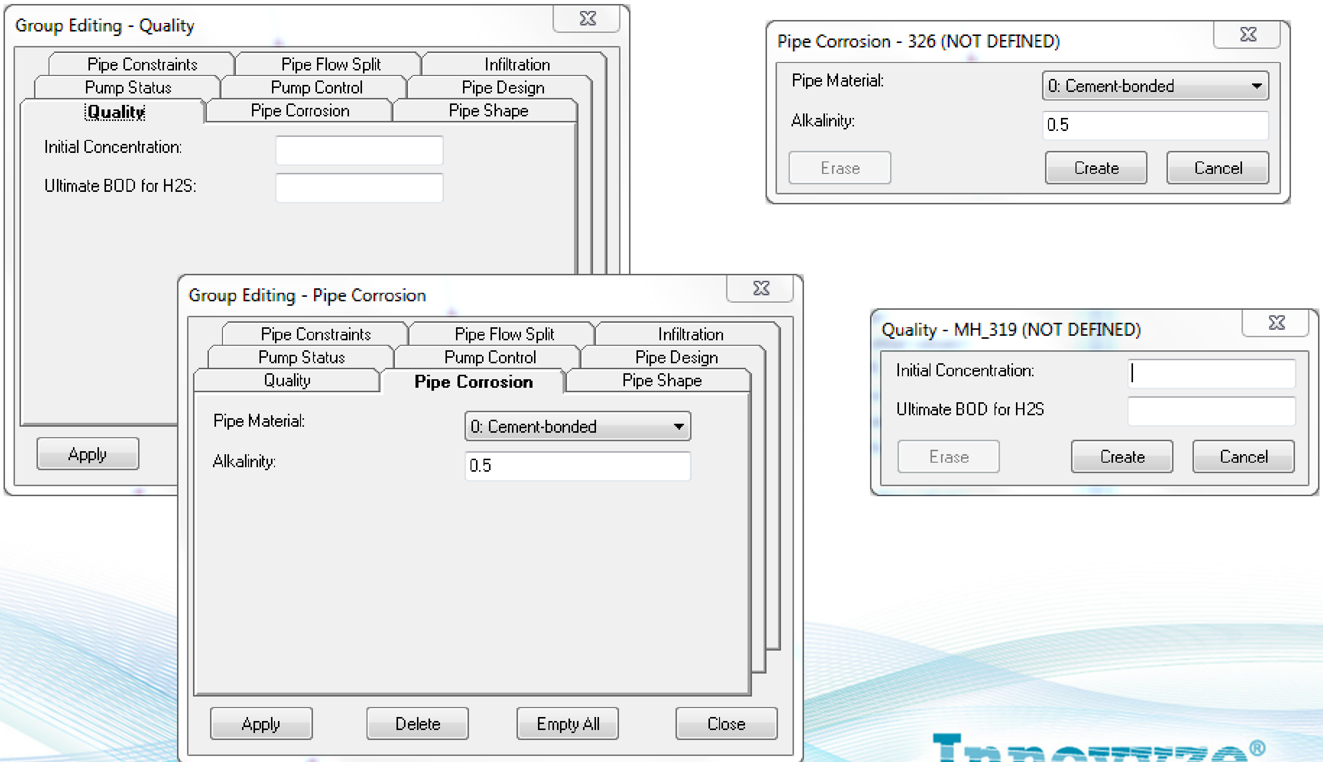
Data required:

- Temperature (ºC)
- Reacton Rate Coefficient (per day)
- pH
- Sulfide Flux Coefficient Sulfide Loss Coefficient Ionization Constant Soluble Form of Total Sulfides
- Relative Saturation
- Acid Reaction Efficiency
- Default Pipe Material
- Default Alkalinity for Cement-bonded pipes

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.

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.

Assigning Values



Student Notes:

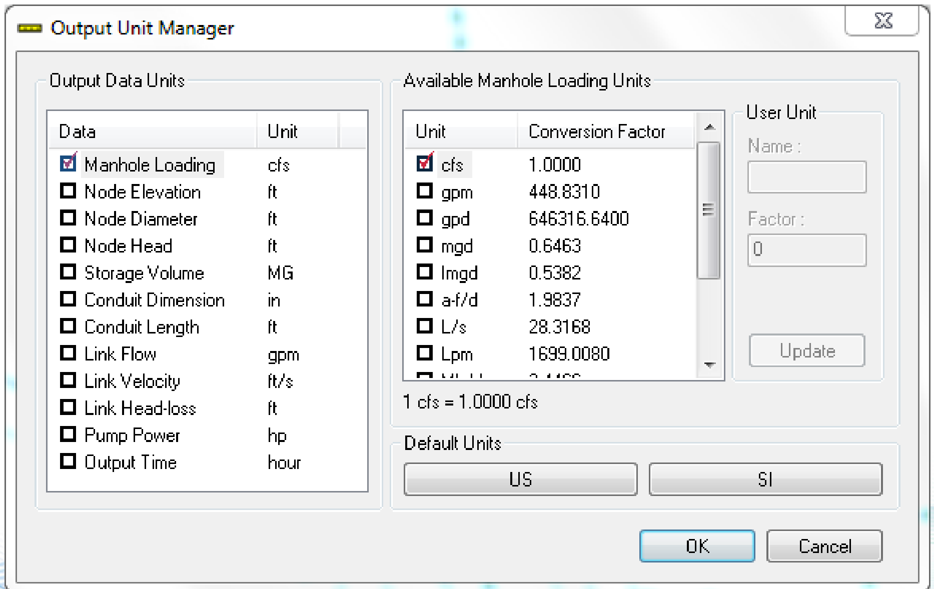
Chapter 12

Additional InfoSewer Features

Student Notes:

Output Units

- Changes output units:
  - Reports
  - Graphs
  - Contours
  - Labels



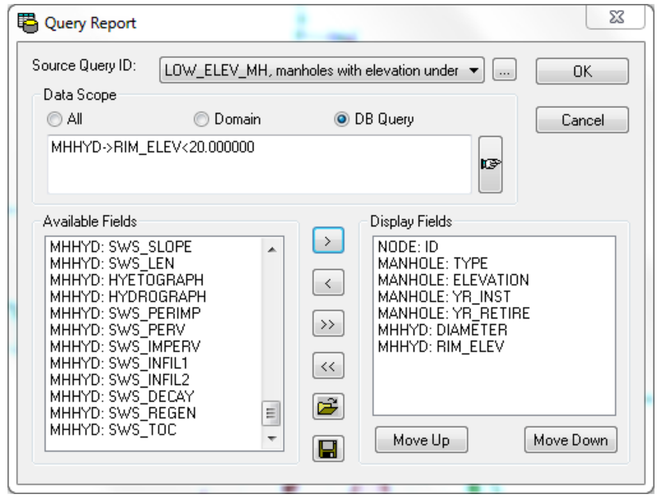
Student Notes:

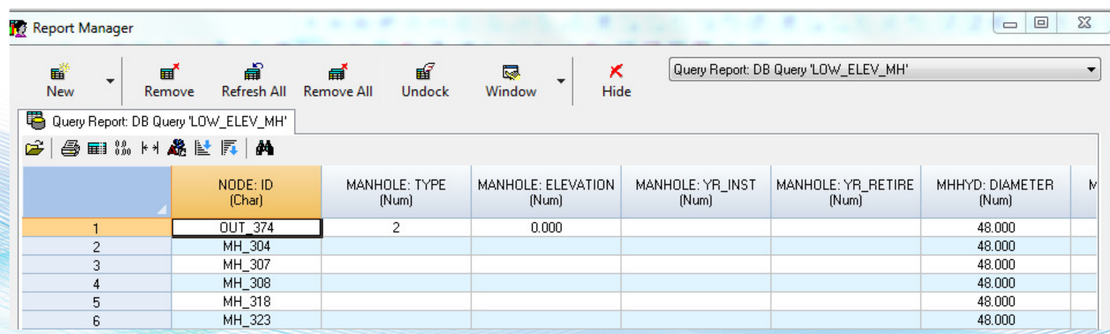
The output unit manager details the units that will be graphed, reported, or annotated. The parameter with the current check mark has all available options for that data type listed on the right under available units. The red check under the available units shows which unit is currently selected for output display.

Use this interface to specify the output data display units. The selected units will be displayed on all output data, graphs, and reports regardless of the input units

Query Reports

- Similar to Customized Reports
  - Specified Data Scope
  - Selected Fields
  - Must Query Output Relate to get output data



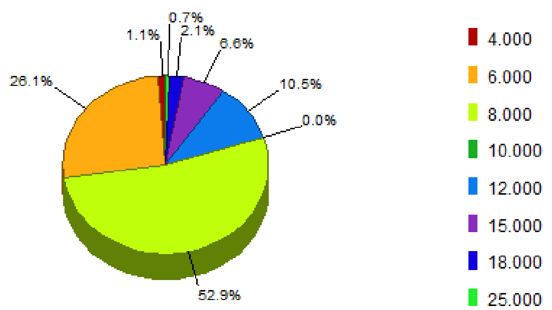


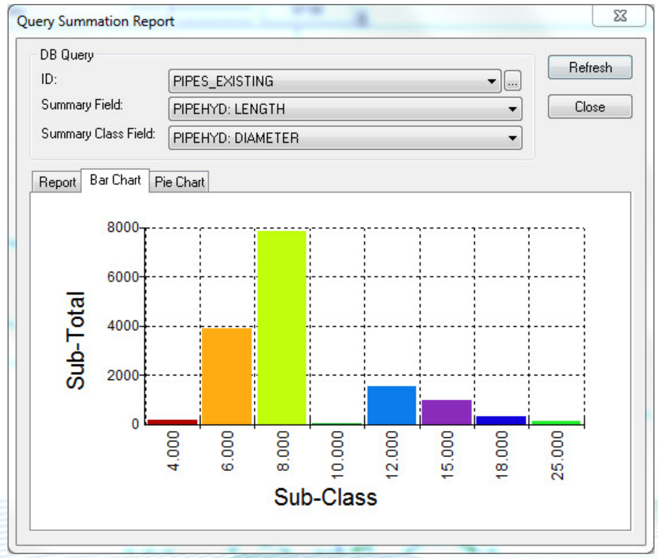
Student Notes:

Query reports are useful for viewing input and information data for specified elements, those matching the specified Database (DB) Query. Any DB query may be used to create a query report, but a query set may not be used in conjunction with the query report manager. The settings of a query report may be saved and reloaded at a later time. To access output data using a query report, a db query must be used that references an output relate.

Query Summation Reports

- Sum one field based upon another
- Multiple views:
  - Report
  - Bar Chart
  - Pie Chart



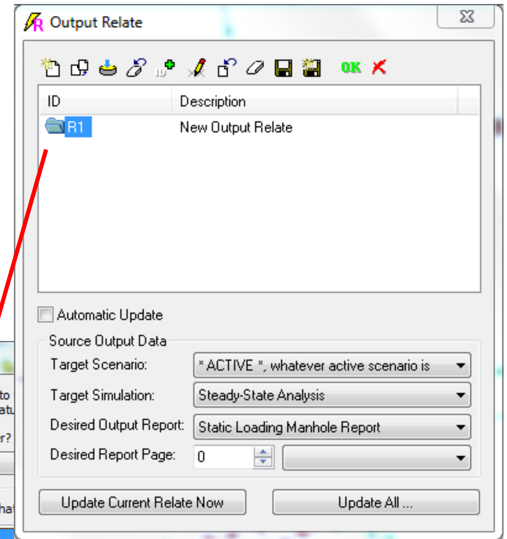


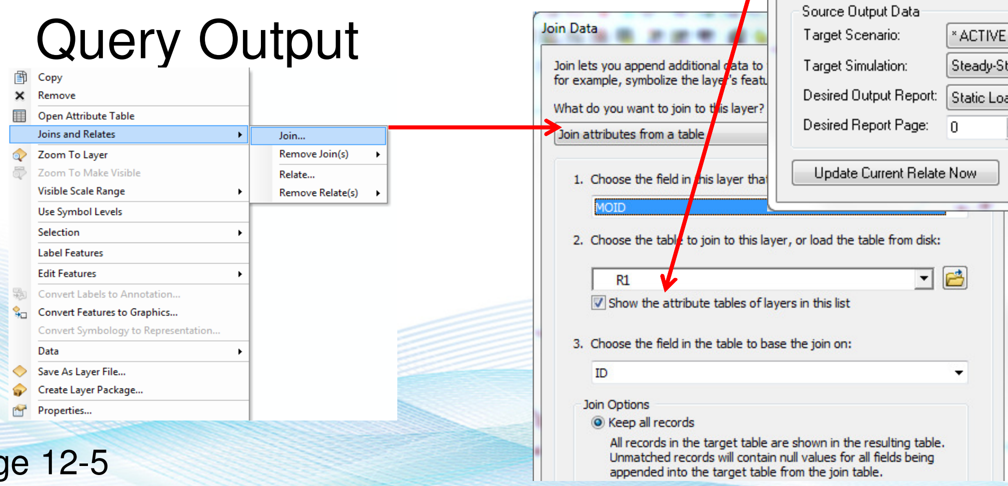
Student Notes:

A query summation report groups elements according to values in a specified field. Within each group, the values of another field are summed. This information is reported in a table or chart format. The figures above depict length of pipe for each diameter value. Functionality of the charts is the same as that of the graphs in the output report manager.

Output Relates

- Link Output Data to GIS Data
  - Tabular join
  - Relationship class
- Update Automatically
- Query Output



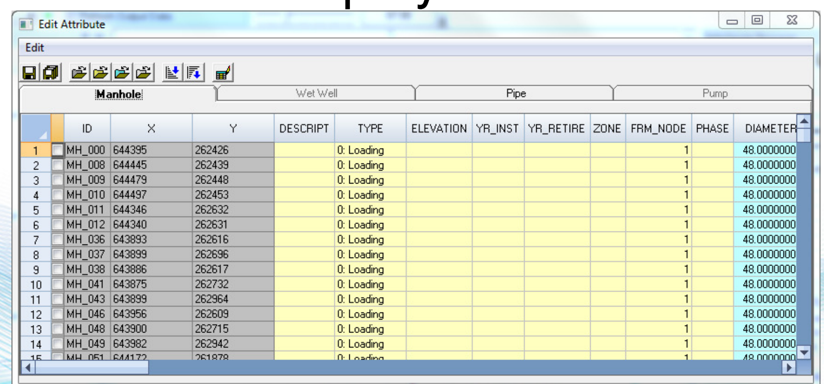


Student Notes:

Output relates are databases (dBase IV) files that can be linked to other GIS data sourced. The output relate can be automatically updated after each simulation, or updated by pressing one button. The relate may be associated with GIS data using either an ArcView join or an ArcInfo relationship class. Refer to the ArcGIS Helpfile for details on joins and relationship classes.

Domain Attribute Editor

- Access modeling, information, geometry, and output tables
- Access databases for all feature types in one interface
- Restrict the display to the desired tables

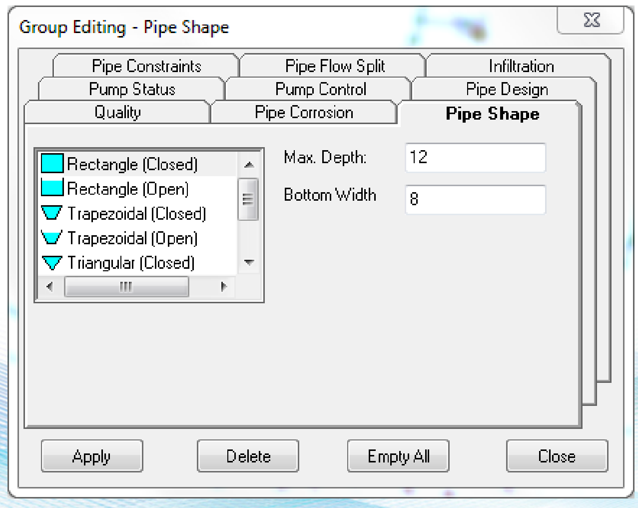


Innovyze®

Student Notes:

Group Editor

- Access advanced database tables
- Delete current values in domain
- Empties all records



Student Notes: