Innovyze®

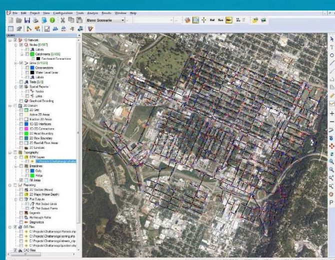
Stormwater and Flood Modeling XPSWMM & XPStorm

Lecture and Exercise

innovyze.com

Stormwater and Sanitary Modeling

xpswmm & xpstorm



Innovyze®
Empowering water experts

1

Introductions

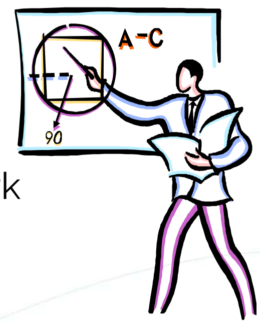
- Where are you from?
- Do you have experience using xpswmm/xpstorm?
- Experience with other water resources software?
- What do you expect to learn?
- Something interesting about yourself!

Innovyze®

2

Workshop Format

- PowerPoint Lecture
- Software Demonstration
- Independent and Guided Work
- Review Questions/Discussion



Innovyze®

3

Storm/Sewer/Flood Family

Rank	Question*	InfoWorks ICM	xpswmm	xpstorm	InfoSWMM®	InfoSewer®
1	What is the primary problem you are trying to understand?	Urban Sewers & Urban/River Flooding	Integrated 1D/2D Urban Sewer/River Modeling	Integrated 1D/2D Urban Flood Modeling	Sewer systems (storm & sanitary)	Sewer
2	How large is the problem?	City & Watershed (200,000 nodes)	Medium Scale (32,000 nodes)	Medium Scale (32,000 nodes)	City Scale (100,000 nodes)	City Scale (100,000 nodes)
3	What is your preferred level of GIS integration?	Standalone CAD & GIS	Standalone CAD & GIS	Standalone CAD & GIS	Full GIS Integration (ArcMap)	Full GIS Integration (ArcMap)
4	Are you looking to support a workgroup?	Team Scalable Simultaneous Users	Single User	Single User	Single User	Single User
5	Does the solution need to support FEMA Flood Studies or Letter of Map Amendments (LOMRs)?	Regional FEMA approved	National FEMA Approved	National FEMA Approved	via SWMM5	None
6	Do you support casual or new modelers?	Expert guidance	Yes Visual UI	Yes Visual UI	Yes, GIS background helpful	Yes, GIS background helpful
7	Do you need green infrastructure?	CS-SUDS, SWMM5 LIDs 2D SUDS	Catchment, BMP, 2D	Catchment, BMP, 2D	SWMM5 LIDs SUSTAIN	None

*Always consider local regulations or unique situations that might drive the selection of a specific model.

Innovyze®

4

Storm/Sewer/Flood Family

	InfoWorks ICM	xpswmm	xpstorm	InfoSWMM	InfoSewer
Key features	• Powerful Modeling & Advanced RTC Controls • Data Flagging • Unstructured Mesh • 3D Viewer • GPU & Multicore processing	• Design to Model • Visual UI • Global Storms & Statistical Analysis • Emergency Route Planning • GPU & Multicore processing • Steps for 2D cell counts	• Design to Model • Visual UI • Global Storms & Statistical Analysis • Emergency Route Planning • GPU & Multicore processing • Steps for 2D cell counts	• Scenario Management • Suite Modules like: • RDII Analyst, Subcatchment Manager, Calibrator, DWF Allocator, Sustain • Multicore processing • Unlimited 2D elements	• Scenario Management • Steady State • Peaking Factors • DWF Design tool • Fast solution • Load Allocator • HS2
Key Hydrology Methods	17 methods • EPA SWMM 5 • SCS Hydrology • UH (Clark, Snyder) • UK Methods (Variable PR, Wallingford, ReFH) • Japanese Methods • Res. (Desbordes, Sprint) • Prob. Density Model • Rain on Mesh	23 methods • EPA SWMM • SCS Hydrology • Unit Hydrographs (Clark, SBUH, Snyder, Nash, T-A) • UK Methods (Variable PR, Wallingford, ReFH, FEH, FSR) • Rain on Grid	23 methods • EPA SWMM • SCS Hydrology • Unit Hydrographs (Clark, SBUH, Snyder, Nash, T-A) • UK Methods (Variable PR, Wallingford, ReFH, FEH, FSR) • Rain on Grid	13 methods • EPA SWMM 5 • SCS Hydrology • Unit Hydrographs (Clark, SBUH, Snyder, Espey, CUHP, Delmarva) • Modified Rational • German Runoff • Rain on Mesh	Requires Pro Version: • Rational for Steady State • Unit Hydrograph For EPS (User defined, SCS, CUHP)

Innovyze®

5

Online Storm and Flood Agenda

Day 1 – Morning session
- Module 1 – Introduction/User Interface Skills
- Module 2 – Hydrologic Analysis

Day 1 – Afternoon session
- Module 3 – Hydraulic Analysis
- Module 11 – Advanced Modeling Tools

Day 2 – Morning session
- Module 10 – 1D River Modeling and Mapping
- Module 12 – 1D/2D River Modeling

Day 2 – Afternoon session
- Module 13 – Integrated 1D/2D Urban Modeling

Innovyze®

6

Workshop Storm and Flood Agenda

Day 1 – Morning session
- Module 1 – Interface and Hydrologic Analysis
- Module 3 – Hydraulic Analysis

Day 1 – Afternoon session
- Module 6 – Low Impact Development
- Module 11 – Advanced Modeling Tools

Day 2 – Morning session
- Module 10 – 1D River Modeling and Mapping
- Module 12 – 1D/2D River Modeling

Day 2 – Afternoon session
- Module 13 – Integrated 1D/2D Urban Modeling
- Module 14 – Levee Failure and 2D Scenarios

Innovyze®

7

Workshop Storm and Sanitary Agenda

Day 1 – Morning session
- Module 1 – Interface and Hydrologic Analysis
- Module 3 – Hydraulic Analysis

Day 1 – Afternoon session
- Module 6 – Low Impact Development
- Module 11 – Advanced Modeling Tools

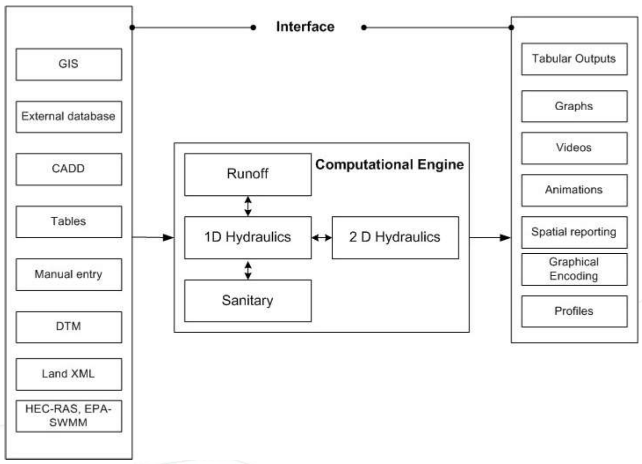
Day 2 – Morning session
- Module 10 – 1D River Modeling and Mapping
- Module 4 – Sanitary Sewer Modeling

Day 2 – Afternoon session
- Module 7&8 – RTC; Pumps and Force Mains
- Module 13 – Integrated 1D/2D Urban Modeling

Innovyze®

8

Model Structure



Innovyze®

9

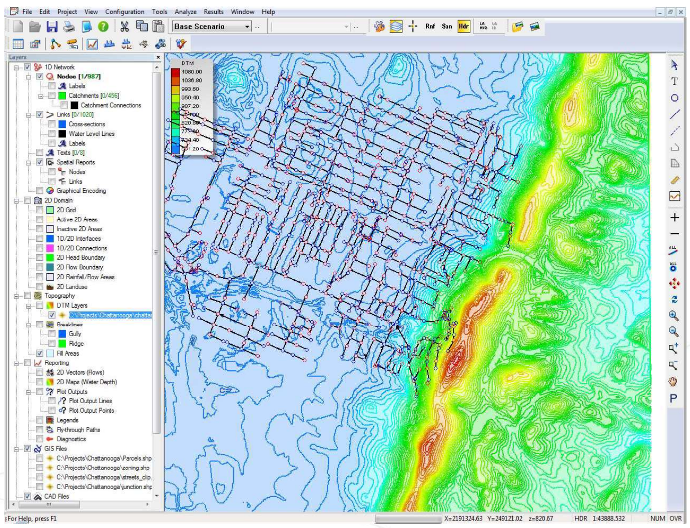
1D Modeling Components

- Backdrop (Optional)
  - CAD, GIS, Aerial Photos
- Catchments
  - Optionally visualized as polygons
  - Imported from GIS, CAD, Databases etc.
- Nodes
  - Catchment Outlet in Runoff
  - Manhole/Inlet/Junction in Hydraulics
- Links
  - Open and Closed Conduits (Dual Drainage)
  - Pumps, Orifices, Weirs, Rating Curves…

Innovyze®

10

Backdrops

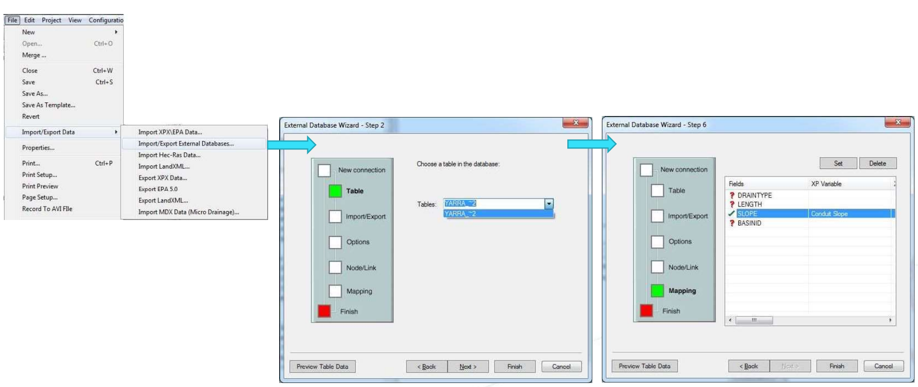


1. ESRI Shape Files
2. Geo-referenced Images
3. CAD Files
4. DTM Model

Innovyze®

11

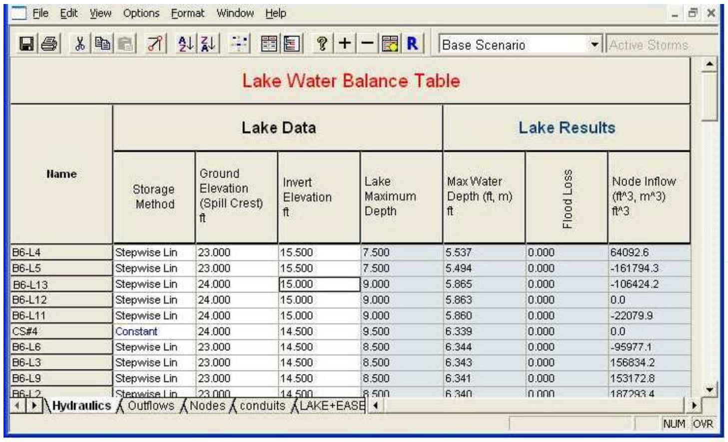
Model Build – External Database Links



Innovyze®

12

XPTABLES: Tabular Reports



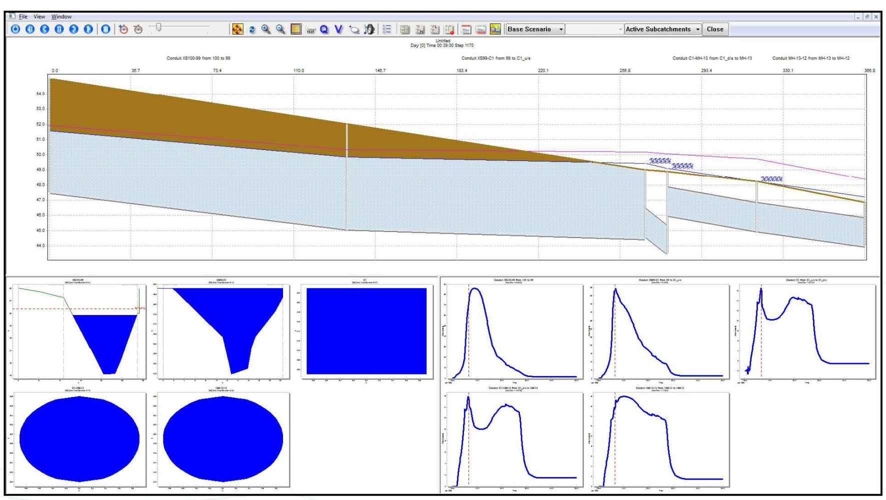
Lake Water Balance Table

Name	Lake Data				Lake Results		
	Storage Method	Ground Elevation (Spill Crest) ft	Invert Elevation ft	Lake Maximum Depth	Max Water Depth (ft, m) ft	Flood Loss	Node Inflow (ft^3, m^3) ft^3
B6-L4	Stepwise Lin	23.000	15.500	7.500	5.537	0.000	64092.6
B6-L5	Stepwise Lin	23.000	15.500	7.500	5.494	0.000	-161794.3
B6-L13	Stepwise Lin	24.000	15.000	9.000	5.865	0.000	-106424.2
B6-L12	Stepwise Lin	24.000	15.000	9.000	5.863	0.000	0.0
B6-L11	Stepwise Lin	24.000	15.000	9.000	5.860	0.000	-22079.9
CS#4	Constant	24.000	14.500	9.500	6.339	0.000	0.0
B6-L6	Stepwise Lin	23.000	14.500	8.500	6.344	0.000	-95977.1
B6-L3	Stepwise Lin	23.000	14.500	8.500	6.343	0.000	156834.2
B6-L9	Stepwise Lin	23.000	14.500	8.500	6.341	0.000	153172.8
B6-L2	Stepwise Lin	23.000	14.500	8.500	6.340	0.000	187293.4

Innovyze®

13

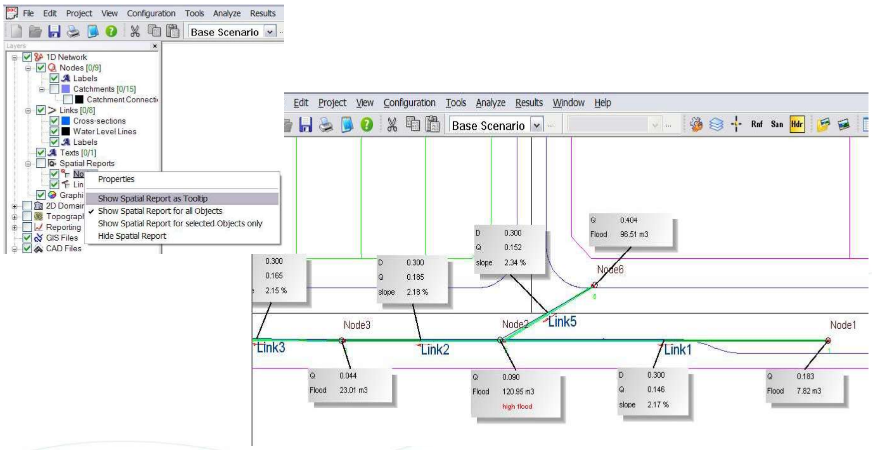
Dynamic Section Views: Animate HGL



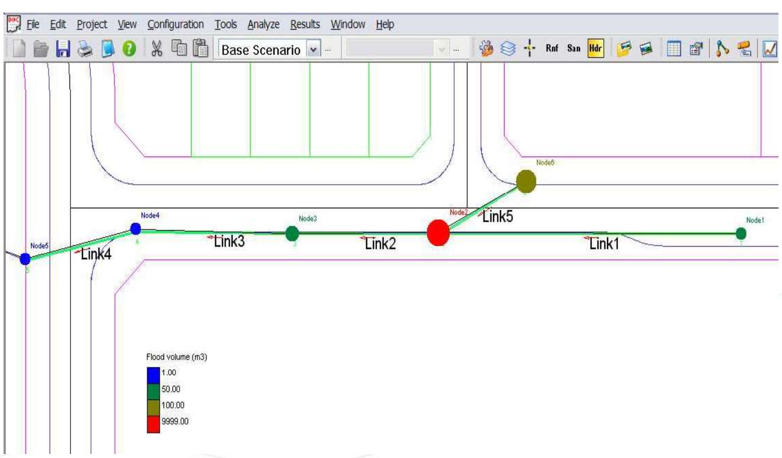
Innovyze®

14

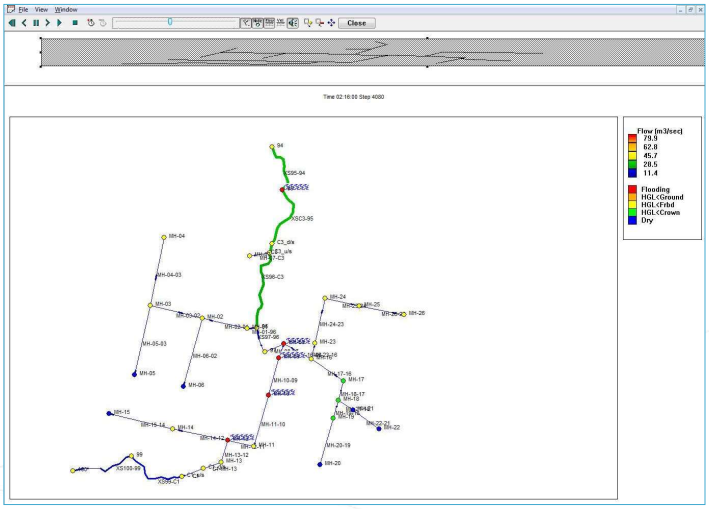
Graphical Reporting: Spatial Reports



Graphical Reporting: Graphical Encoding



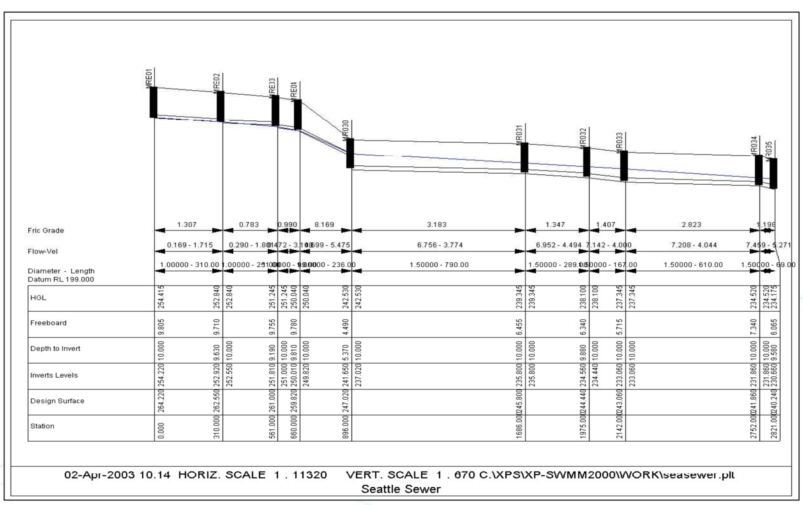
Dynamic Plan View



Innovyze®

17

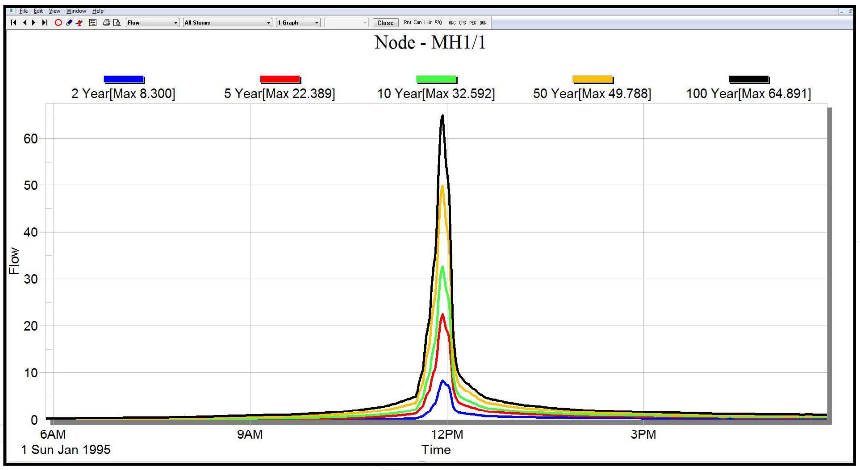
Profile Plotting



Innovyze®

18

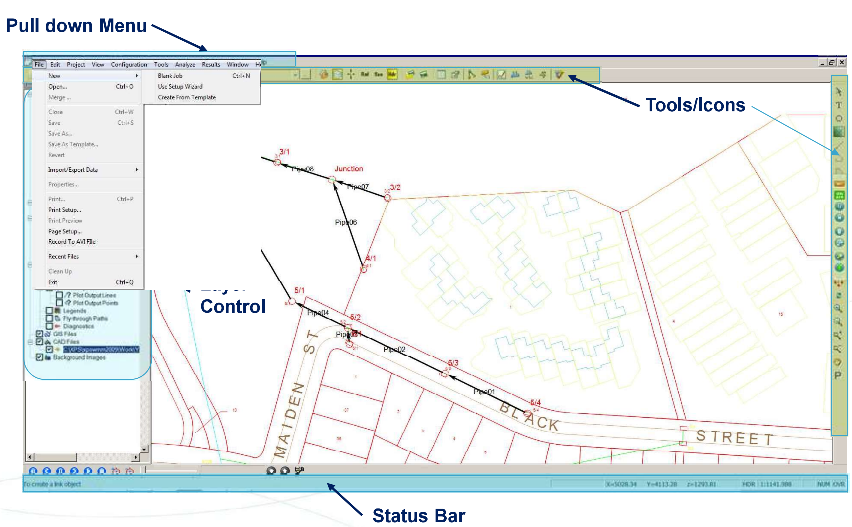
Review Results: Time Series Graphing



Innovyze®

19

The XP User Interface

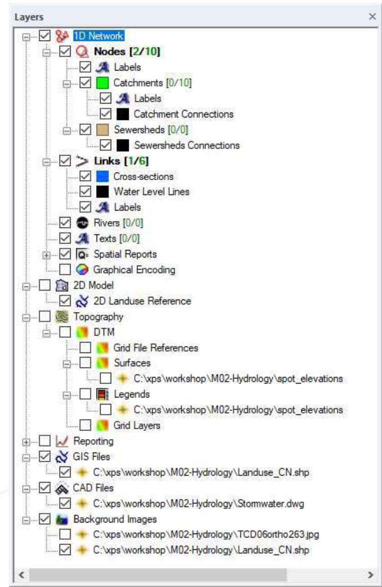


Innovyze®

20

Layer Control Panel

- Display check box
  - On
  - Off
- Nodes
  - Lock
  - Unlock
  - Properties
  - Selected/Count [1/5]
- Links
  - Lock
  - Unlock
  - Properties
  - Selected/Count [0/4]



Innovyze®

21

Runoff Mode Highlights

- Continuous Simulation (Some methods)
- Runoff Method is Deterministic: Rainfall, Snowmelt, Infiltration, Evaporation, and Groundwater are simulated
- Surface Hydrology using 23 methods: Runoff, SCS, Kinematic Wave, and many Unit Hydrograph Methods
- Infiltration by Horton, Green-Ampt, Initial, Proportional and Continuing Losses, SCS Loss Method
- RDII using RTK method of 3 unit hydrographs
- 2 Compartment Groundwater Module (Mounding)
- Water Quality (nonpoint source pollutographs)
- LID (WSUD)
  - Redirect Surface Flows
  - BMP modeling in Runoff at node or catchment
- Global Storms

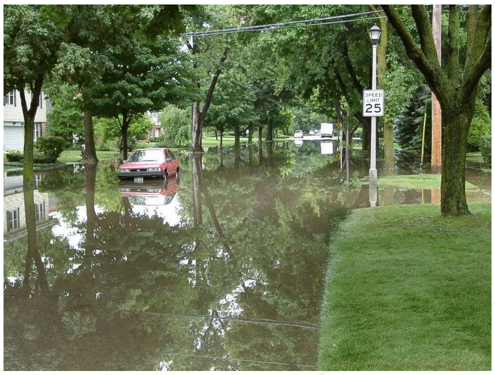
Innovyze®

22

Hydrology

The Science or "Art" of changing rainfall to runoff

It can be complicated!



Innovyze®

23

Simplified Hydrologic Budget

PRECIPITATION – LOSSES = RUNOFF

- PRECIPITATION
  - Rainfall intensity, duration and volume
  - Snow and subsequent Snow Melt
- LOSSES
  - Evapotranspiration, Infiltration, Depression Storage
- RUNOFF
  - Hydrograph (Peak flow rate, time to peak and volume)

Innovyze®

24

Precipitation Data Types

Point Source
- Fixed Time Intervals
- Variable Time Intervals
- Design Storm Patterns (ATLAS 14, SCS, HUFF)

Aerial
- Radar
- NEXRAD Weather Sensing Radar Doppler (Since 1991)
- Satellite Sensors

Duration
- Event
- Design Storms
- Continuous

Innovyze®

25

Storage and Losses

- Interception
- Evapotranspiration
  - Daily/Monthly fixed
  - Recorded Daily Values
- Infiltration
  - Soil moisture storage tracking
  - Groundwater coupled
- Depression storage
  - Ultimately Infiltrated or Evaporated
- Surface Detention
  - Slope, roughness, width and method dependent
- Some Additional Factors
  - ***Season parameter variation***
  - ***Annual parameter variation***
  - Duration of Rainfall Event
  - Antecedent Conditions
  - Temperature, (snowfall events)

Innovyze®

26

Runoff

- Hydrograph
  - Flow over time with peak, time to peak and volume
  - Many parameters influence the shape, but rainfall is most dominant input
- Related to rainfall frequency and antecedent conditions
- Continuous or event depending on source input
- Varies based on method and catchment parameters
- Parameter sensitivity different for low intensity vs. high intensity rainfall

Innovyze®

27

SCS Hydrology

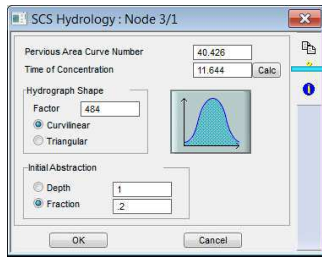
- CONCEPT:
  - Developed by the USDA NRCS (Soil Conservation Service)
- Data Needs
  - Drainage Area
  - Curve Number
  - Time of Concentration
  - Shape Factor
  - Initial Abstraction
- Limitations
  - Basic Infiltration description
  - No simulation of soil storage for continuous rainfall
  - Not for storms less than 0.5 inches

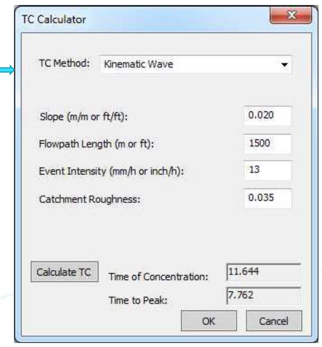
Innovyze®

28

SCS Details

- Curve Number
  - 20 to 98
- Time of Concentration
  - Direct input or calculate
- Shape Factor
  - Q_p=484A/t_p
  - Curvilinear/triangular
  - 100 – 800
  - Default 484
- Initial Abstraction
  - Depth or Fraction options
  - This loss is satisfied prior to start of runoff





Innovyze®

29

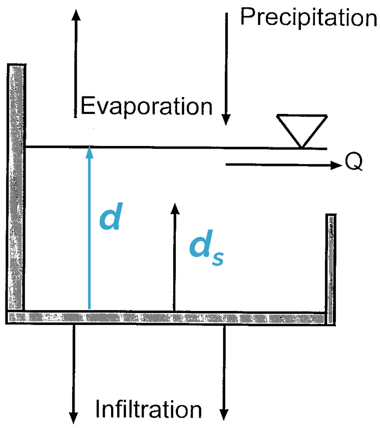
SWMM Runoff

- CONCEPT:
  - Developed by the USA EPA as a deterministic approach to runoff
- Data Needs
  - Drainage Area
  - Percent Impervious (Directly Connected or DCIA)
  - Width
  - Basin Slope
  - Infiltration Method and Infiltration Parameters
  - Evaporation (can be zero)
- Limitations
  - Lumped Catchment Parameters

Innovyze®

30

SWMM Runoff



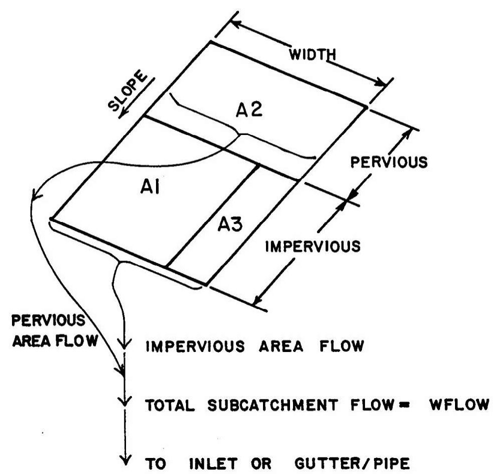
$$Q = \frac{1.486}{n} \bullet W \bullet (d - d_s)^{5/3} \bullet S^{1/2}$$

EPA Surface Runoff
Non-linear Reservoir

Innovyze®

31

Catchment Detail

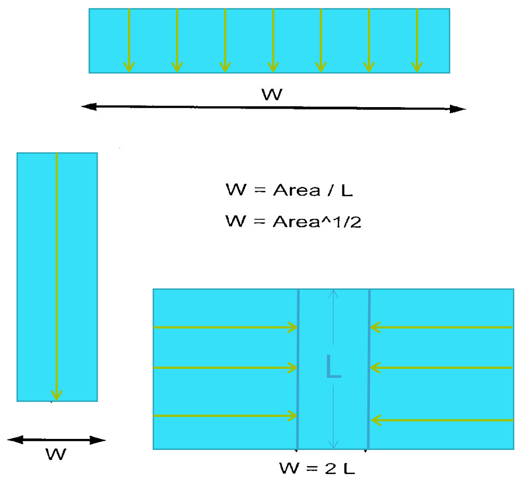


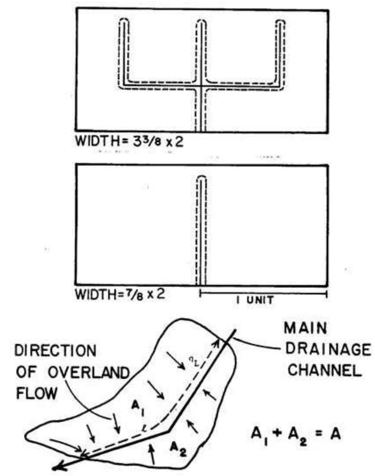
- A2: Pervious Area
- A1 & A3: Impervious Area
  - A1 with depression storage
  - A3 without depression storage
- Slope and Width same
- Depression Storage and roughness are unique

Innovyze®

32

Subcatchment Width





The subcatchment or watershed width parameter is a key parameter in calibrating peak flow and total flow volume.

Innovyze®

33

Infiltration Options

- Horton
  - optional cumulative max infiltration (i.e. wetland)
- Green-Ampt (best continuous simulation choice)
- Uniform Loss
  - Proportional Loss
  - Initial and Continuing Loss
  - Initial and Proportional Loss
- SCS
  - Fraction Initial Abstraction
  - Fixed Depth Initial Abstraction

Innovyze®

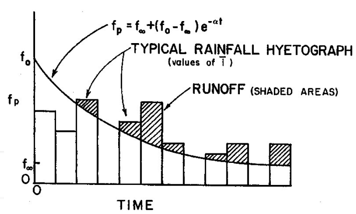
34

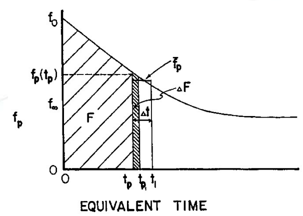
Horton Infiltration

Infiltration capacity as a function of time:

fp = fc + (fo - fc) e -kt

where:
fp = infiltration capacity into soil, ft/sec,
fc = minimum or ultimate value of fp (WLMIN, ft/sec,
fo = maximum or initial value of fp (WLMAX), ft/sec,
t = time from beginning of storm, sec, and
k = decay coefficient (DECAY), sec -1.





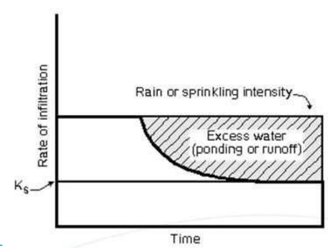
Innovyze®

35

Green-Ampt Equation

For F < Fs: f=i and Fs=Su IMD/(i/Ks-1) for i>Ks
and no calculation of Fs for i<Ks

For F > Fs: f = fp and fp=Ks(1+Su IMD/F)



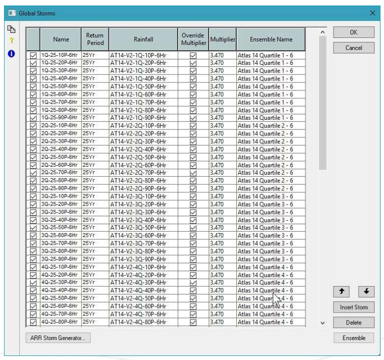
Where:
f = infiltration rate, ft/sec,
fp = infiltration capacity, ft/sec,
i = rainfall intensity, ft/sec,
F = cumulative infiltration volume, this event, ft,
Fs = cumulative infiltration volume causing surface saturation, ft,
Su = average capillary suction (SUCT), ft water,
IMD = initial moisture deficit for this event (SMDMAX), ft/ft, and
Ks = saturated hydraulic conductivity of soil, (HYDCON) ft/sec.

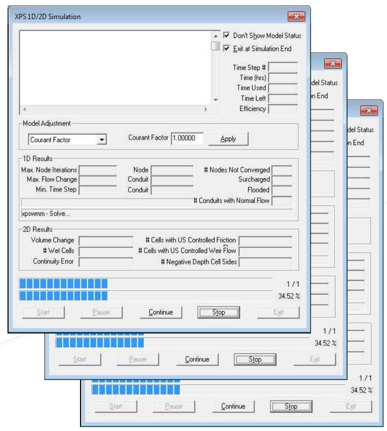
Innovyze®

36

Global Storms

- Storms grouped as Ensembles
- Use Solve Manager for concurrent simulations
- Atlas 14 Rainfall



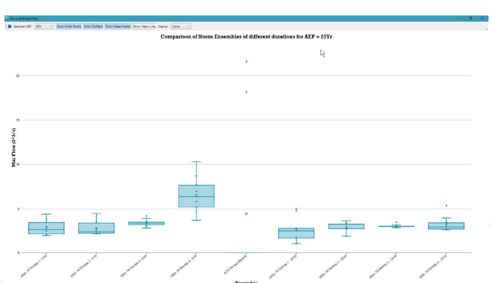


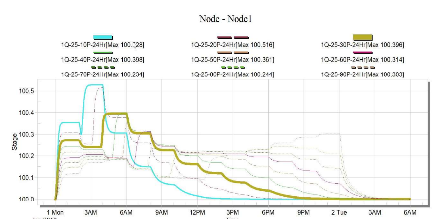
Innovyze®

37

Global Storm Comparisons

- Multiple results in tables and graphs
- Specialized Box and Whisker Plot



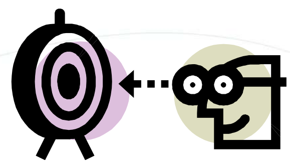


Innovyze®

38

Workshop Example Preview

- Interface skill development adding
  - CAD, Aerial Images and DTM layers
  - Import Node, Catchments and Links from GIS
- Derive Hydrologic data and catchment connections
- Use SCS and SWMM Hydrologic Methods
- Solve and Review Runoff analysis
- Use Global Storms



Innovyze®

39

Innovyze®

Empowering water experts

40

Surface Water Hydrology

The purpose of this training module is to teach users how to use XP's tools to create a hydrologic model from GIS files. You will learn how to use the Soil Conservation Service (SCS) and the SWMM Runoff methods for generating runoff from rainfall. Finally, hydrologic model results will be reviewed graphically and in tabular form.

In the Runoff mode, rainfall, infiltration, evaporation and depression storage for each subcatchment is simulated and the runoff is assigned to a collection node. Up to 5 subcatchments are allowed for each node. A variety of hydrologic methods are available to generate runoff hydrographs. Methods details are described in additional workshop modules and in the program documentation.

Objectives

A stormwater collection network can be developed in the graphical interface using a variety of methods. In this example, we will create a collection system network using the direct import of nodes, links and catchment polygons from shape files with GIS background images. Specifically, you will learn how to:

1. Use a template that provides basic Global Database items
2. Add and georeference an Aerial Photo
3. Add a corresponding CAD file to the project and control layer display
4. Create a surface or Digital Terrain Model (DTM)
5. Create and import data for nodes, and catchments from shape files
6. Use graphical tools to calculate subcatchment areas
7. Connect subcatchments to runoff nodes
8. Display and Derive Catchment Parameters from GIS
9. Create and assign SCS design storms in the Global Database
10. Manage Runoff Job Control settings
11. Run the hydrologic analysis
12. Use the global database to manage infiltration data
13. Define the SWMM runoff parameters in a subcatchment using xptables
14. Use graphical tools to obtain data for catchment parameters
15. Review results using XP Tables and the output file

Data files to be used are:

stormwater_template.xpt
TCD06ortho263.jpg
TCD06ortho263.jgw
Stormwater.dwg
spot_elevations.xyz (to create TIN)
Nodes.shp (GIS files for nodes)
Catchments.shp (GIS files for catchments)
stormwater.xls
Landuse_CN.shp
stormwater4a.xp
stormwater_SCS.xp
stormwater_completed.xp

Build a new model using a template

An xpswmm or xpstorm model includes a series of input and output files. The main files are the .xp database file that holds the data and results for the model and references other external files needed, such as background images or DTM surface data; the .mdb file which holds scenario information; and the .json file which contains rainfall data. A very efficient way to build a new model is to use a template file (.xpt). A template file is essentially an xp database that already has important information such as global database records that may be used for the new file.

Create new file from Template

1. Open xpswmm or xpstorm (xp).

2. At the opening dialog, select **Create From Template…**

3. In the Windows Explorer dialog, navigate to the **xps\workshop\M02-Hydrology** folder and name the file **stormwater.xp**. A file with the default extension (.xp) will be created.

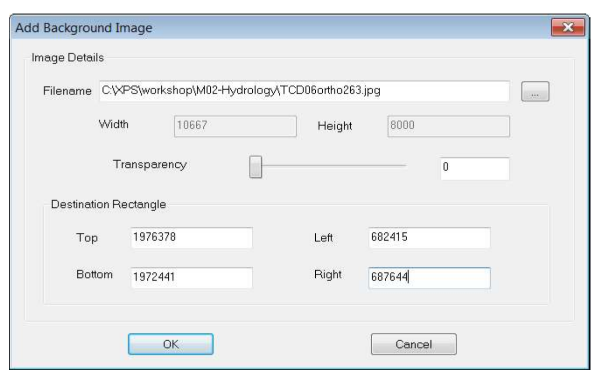
4. In the subsequent Windows Explorer dialog, select the provided template **XPS\workshop\M02-Hydrology\stormwater_template.xpt** and click on **Open**.

5. The template has **File->Properties** set as well as **Configuration->Global Data…->Rainfall** and **XP Tables**.

Add background image and CAD file

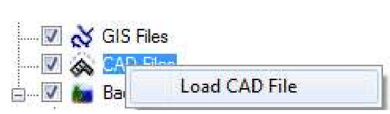
1. Using the menu choose **View->Background Images->Add Background Image** or use the icon on the toolbar for **Add Picture**. 

2. Select the file **XPS\workshop\M02-Hydrology\TCD06ortho263.jpg.** The coordinates **1976378**, **682415**, **1972441** and **687664** for the Top, Left, Bottom and right respectively are read automatically from a world file if present. These coordinates can also be directly entered.



3. Locate the **CAD Files** row on the Layer Control Panel.

4. Right click on the **CAD Files** layer and select **Load CAD File** from the popup menu as shown to the right.

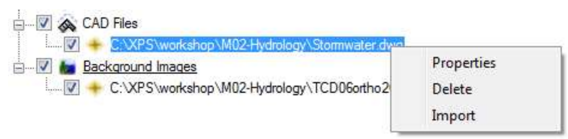


5. In the dialog, select the file **Stormwater.dwg**.

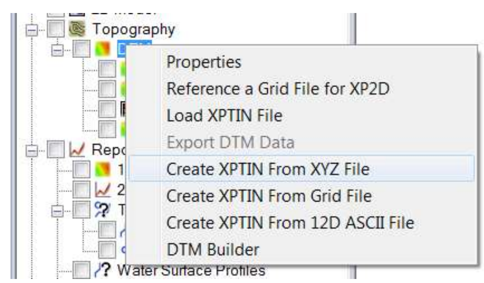
6. Click on **Open** to display the file on the network view. This file is georeferenced so that its x and y coordinates are coincident with the photo and proposed drainage network.

7. Pan the project site by depressing and holding the mouse wheel or right mouse button and then dragging the cursor. A hand appears next to the cursor to pan around the image. Use the mouse wheel to zoom in and out by rolling the wheel forward and backwards, respectively. A second option is to use the pan icon in the navigation panel.

8. To adjust the CAD layer display, right click on the **Stormwater.dwg** layer. Choose **Properties** from the popup menu. You can modify the display of layers in the CAD file at your discretion. Then click **OK** to view the drawing. For example, choose **Properties** again, **uncheck** the **UNITS** layer and view the drawing. You will now see the drawing without these layers of text.



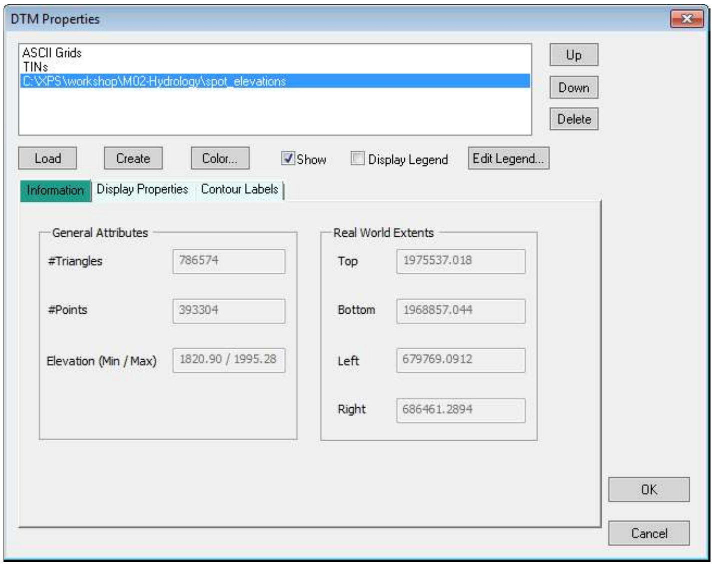
Build DTM



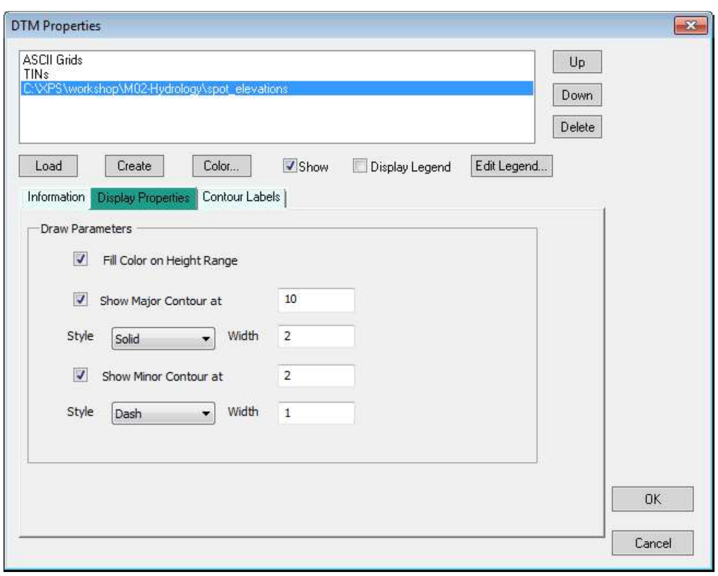
1. To build a digital terrain model (DTM) right-click the **DTM** layer in the Topography section of the Layer Control Panel. From the popup menu, select **Create XPTIN From XYZ File** and **Open** the file **spot_elevations.xyz**.

   Notes: The status bar will display the progress of the triangulation process. Also the S column in a XYZS is used for break lines. When consecutive values have the same S value those points are treated as a break line in the triangulation process.

2. **Right-click** on the line corresponding to the newly created layer. Select **Properties** from the popup menu. Click on **Color** to open the Gradient Colors dialog and set the transparency to about **0.4** using the Transparent slide bar or by directly entering the value. Click on **OK**.

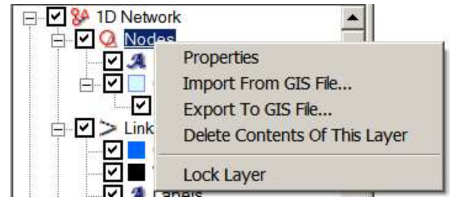


3. Select the **Display Properties** tab and then select the **Show Major Contour at** and **Show Minor Contour at** boxes. Adjust the display properties to show major contours at **10** and minor contours at **2** ft intervals as indicated in the figure above. If the **Fill Color on Height Range** box is unchecked, check this box. Click on **OK**.

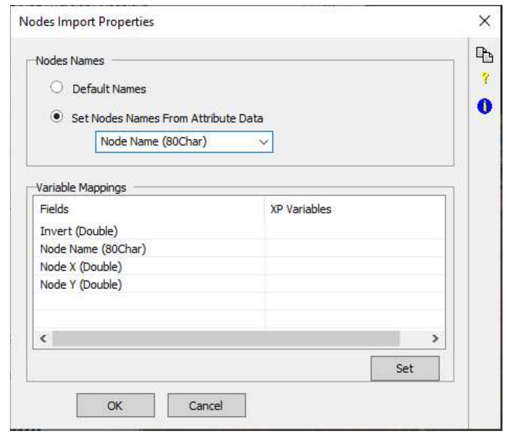


Importing Nodes, and Catchments from GIS (.shp) File

1. We are now ready to import nodes from GIS and overlay them over the optional supporting topographic and background layers. Ensure the Nodes layer is visible and unlocked. Right–click on the **Nodes** layer and choose **Import From GIS File...** Find and **Open** the file **Nodes.shp** and click on **Import**.



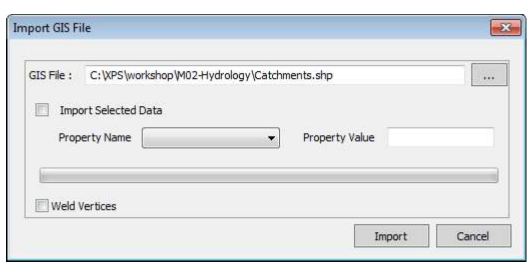
2. Click on the **Set Node Names From Attribute Data** radio button. Select **Node Name** field from the drop list. Other fields of data could be mapped if available in the Variable Mappings. Click on **OK**. A Data Import Information dialog box will report the results of the import and show that 10 objects were read. Click **OK** to close the message window.



3. The network view should show 10 nodes in the upper right-hand corner. **Zoom** to include those 10 objects in the lower left quadrant of the plan view. Note: The nodes may be more visible if the Background Image is turned off.

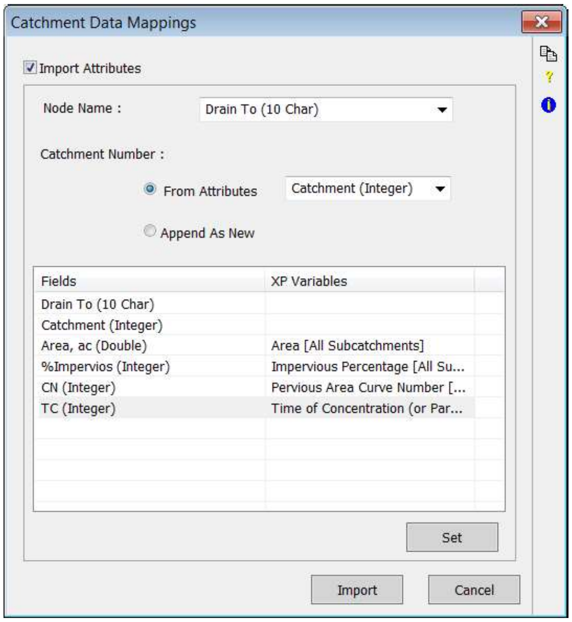
4. Save the View to be able to return to it easily in the future. Use **View->Save View** and enter **Network**. To return to that saved view use **View->Restore Views->Network**.

5. Load catchments from GIS. Make sure the Catchments layer is visible and unlocked. Right-click and choose **Import From GIS File...** from the pop-up menu.



Find and select the file **Catchments.shp**. Click on **Import**.

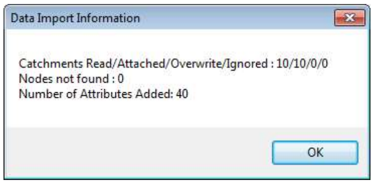
6. Catchments imported from GIS can be assigned to nodes and to a specific catchment position. In addition, hydrologic data can be imported by mapping fields. In the Catchment Data Mappings dialog choose **Drain To (10 Char)** for the **Node Name** field, and **Catchment (Integer)** for the **From Attributes** field. Lastly, in this dialog we will map the hydrologic fields.



7. Map the hydrologic fields. Select the **Area** field and **Set** it to **Node Data and Results->Runoff Node->RNF Node Data->Area** and click **OK**. Repeat this step for the 3 remaining fields. The Impervious Percentage will be found in the list below Area while the CN and TC fields should be mapped to:

   CN: **Node Data and Results->Runoff Node->RNF Node Data->Sub Catchment Data->SCS Hydrology->Pervious Area Curve Number**

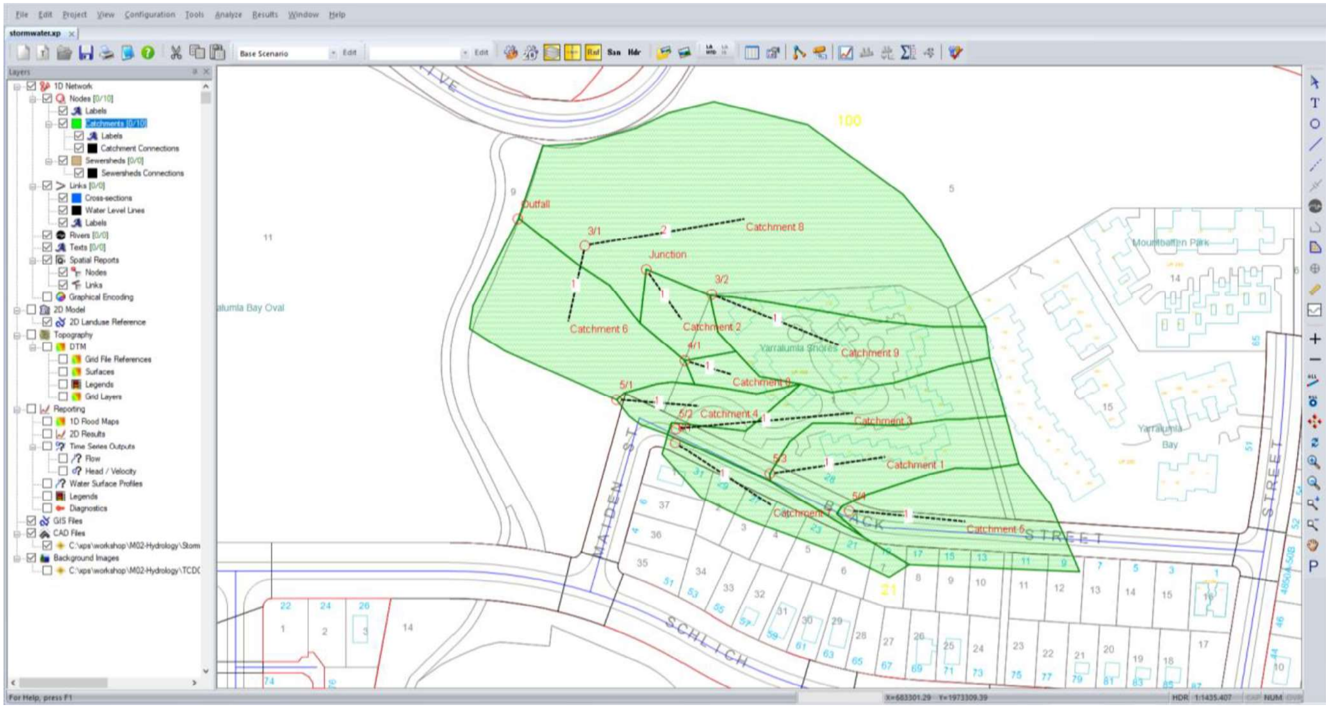
   TC: **Node Data and Results->Runoff Node->RNF Node Data->Sub Catchment Data->Time of Concentration (or Parameter 2)**



   Click on **Import**. A notification dialog box will report the import.

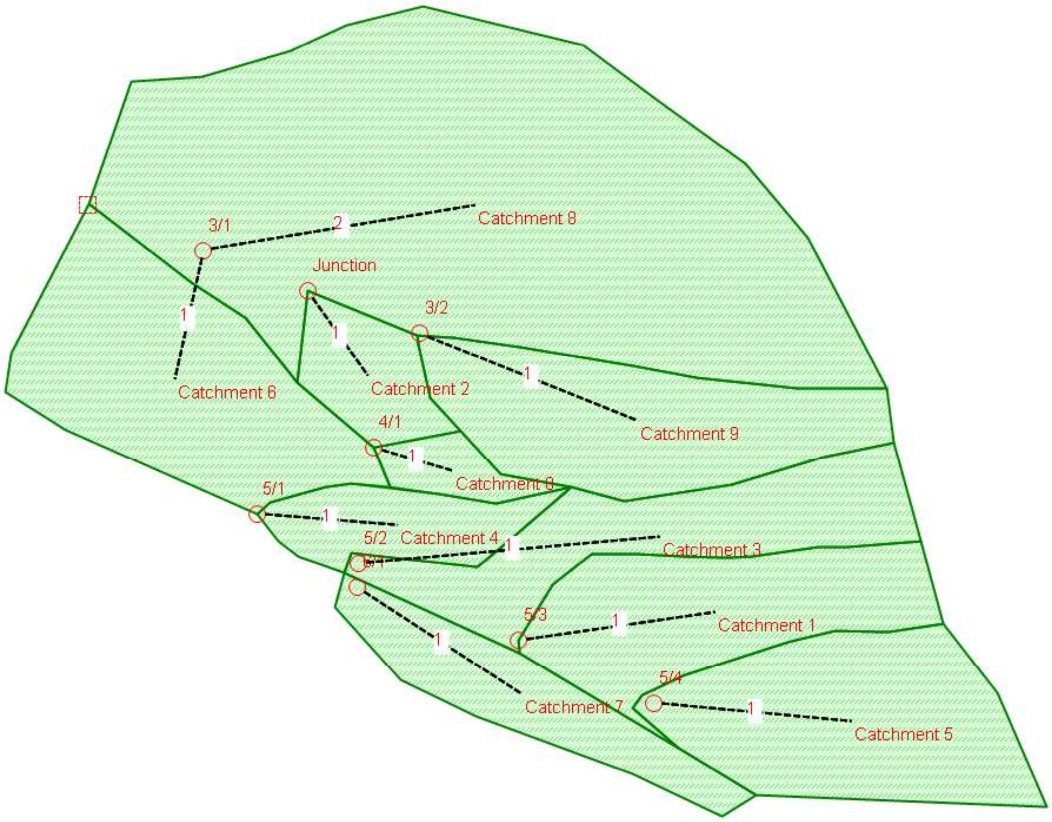
8. Turn off the **Background Image** and **Surface** to visualize the network and catchments.

9. The catchment polygons with the connections are displayed in the network view, as shown below. **Right-click** on the **Catchments** layer in the Layer Control Panel and select **Properties** to adjust the display (color, pattern, line size) of the catchments.

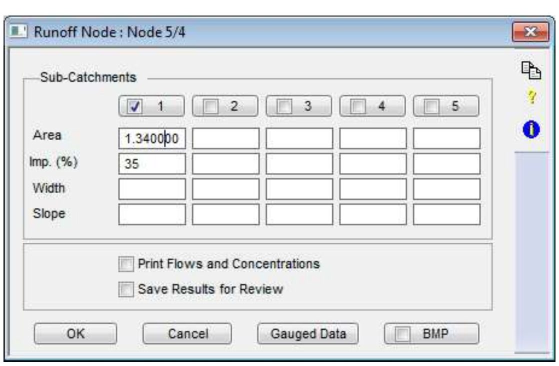


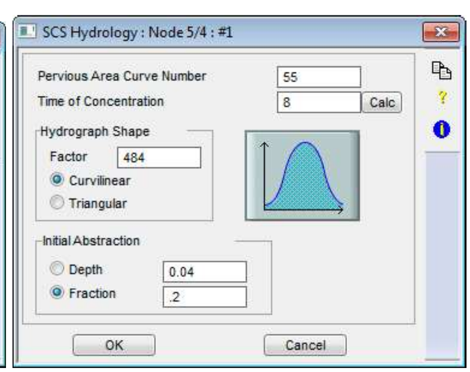
Verify Catchment Data Import and Calculate Areas

1. Verify Catchment Nodes in Runoff. Navigate to the Runoff mode by clicking on the Rnf **Rnf** button in the toolbar if it is not already highlighted. Having imported the nodes to the Runoff Layer they are inactive in the Runoff layer. To make the catchment nodes active, **click** on the **Select All Nodes** tool or <**Ctrl>+A**. Then click on the **"+"** tool or **+** key on the keyboard. This action will make all selected Nodes in the network active in the current mode. Then **click** on white space of the network view to deselect the nodes. Next **click** on the node **Outfall** then click on the **"-"** tool. This will leave only nodes active in the Runoff Mode with catchments. The network should appear as shown on the next page. Note: multiple objects can be selected using the **<Ctrl>** with click.

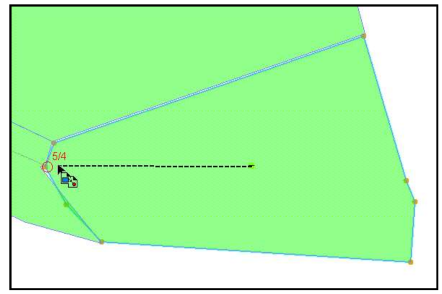


2. **Double-click** the catchment polygon connected to node **5/4**. The imported data can be seen in the Runoff Node dialog, and in the SCS Dialog found within the Sub-Catchment dialog. In the Runoff Node dialog, **Width** and **Slope** values are required, and an error message will display if they are left blank. Enter a value of **1.0** for both to make the error message go away. The **Width** and **Slope** are not used for the SCS routing method, and we will go over these parameters in more detail when we discuss the **RUNOFF** routing method.



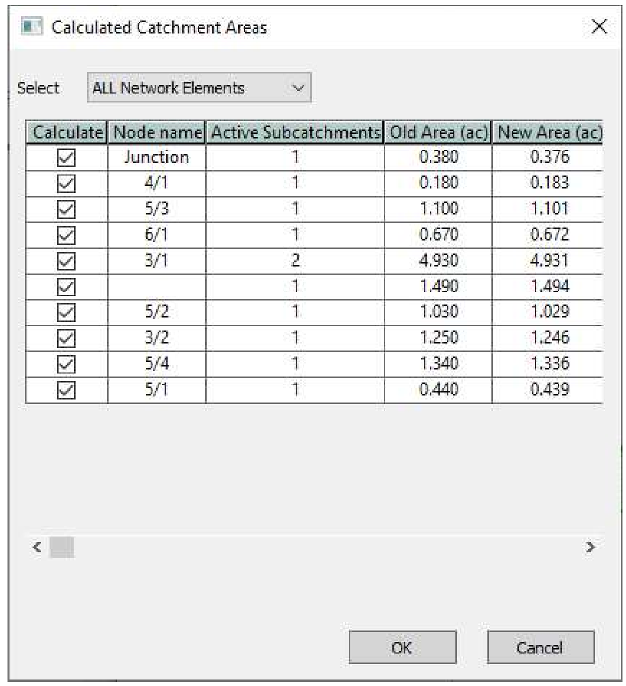


Note: To connect catchment polygons to nodes: Select the polygon. Move the cursor over the centroid, it will appear as shown to the right indicating a catchment to node drawing process. Hold the left button down and draw a line to the node. Once snapped to a node the cursor will appear as a cross. Release the button and a menu will appear. Choose the drain to subcatchment positions.



3. Calculate catchment areas. On the Tools menu select **Calculate Node** and then **Catchment Areas**.

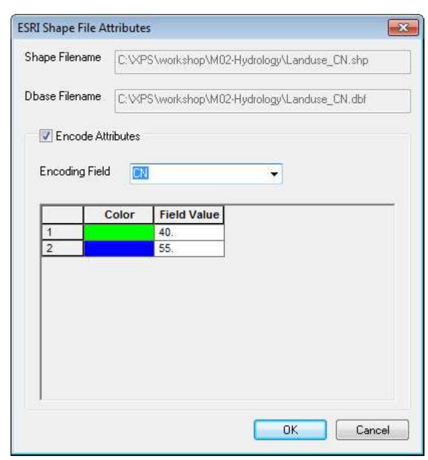
   A dialog will be displayed showing the imported values and new (calculated from the subcatchment polygons) areas. The new areas may be edited to override the calculation. Click on **OK** to accept the new values. This data is added to the model database. A dialog box will show that the calculation was completed successfully. Click **OK**.



4. Save your file as **stormwater2.xp** by selecting the **File** menu and using the **Save As…** selection in the list.

Display and Derive Catchment Data From GIS Files

1. **Right-click** on the **Background Images** layer in the Layer Control Panel and choose **Add Background Image**. Select the file using the ellipsis **(…) Landuse_CN.shp** and press **Open**.

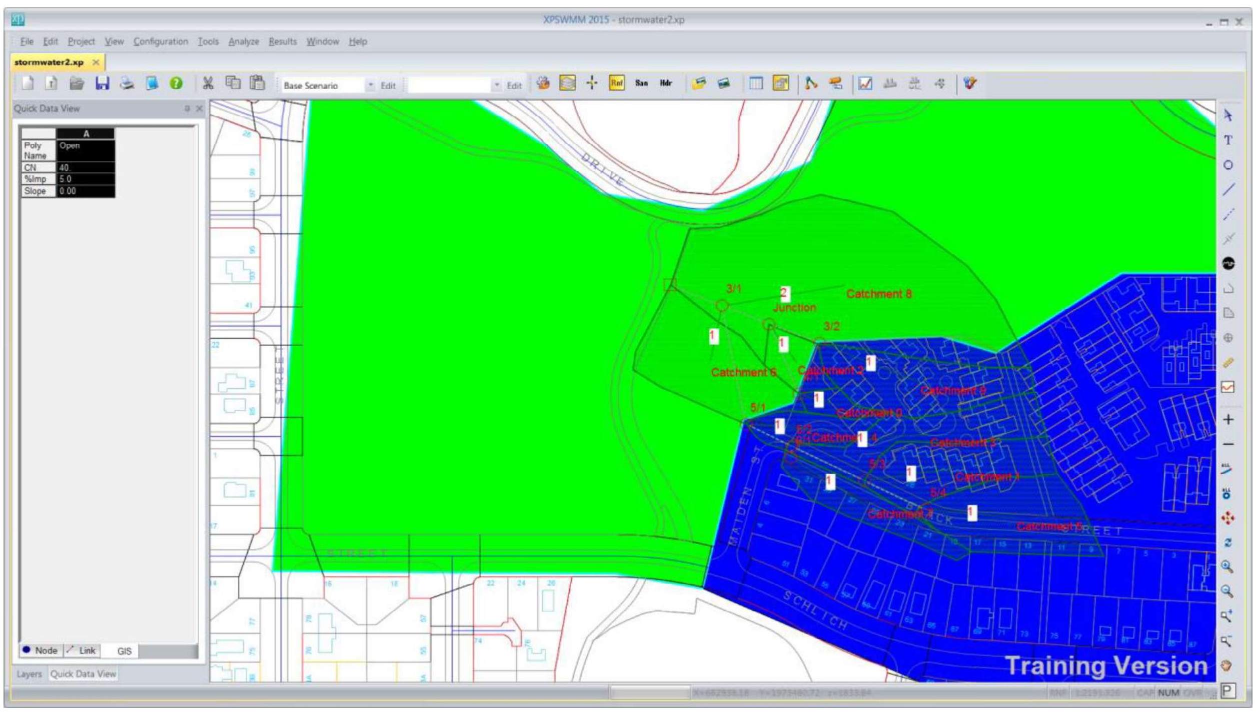


2. In the ESRI Shape File Attributes dialog choose **Encode Attributes** and select **CN** as the Encoding Field. The colors may be altered or accept green for a value of 40 and blue for a value of 55. Click **OK**. This will color the polygons in the plan view based on the CN field. Shapefiles of polylines and points can also be displayed in this manner. **Restore** the **Network** view.

3. Load a GIS File. Data from GIS files can also be displayed in the plan view. **Right-click** on the **GIS Files** layer in the Layer Control Panel and select **Load GIS File**. Select **Landuse_CN.shp** and press **Open**. The polygons are traced with a black line in the Plan View. Color, fill and line color can be specified to the newly added GIS layer, by right-clicking the new layer where the GIS file is loaded, and then selecting **Properties**. All objects will have the same color in this layer.

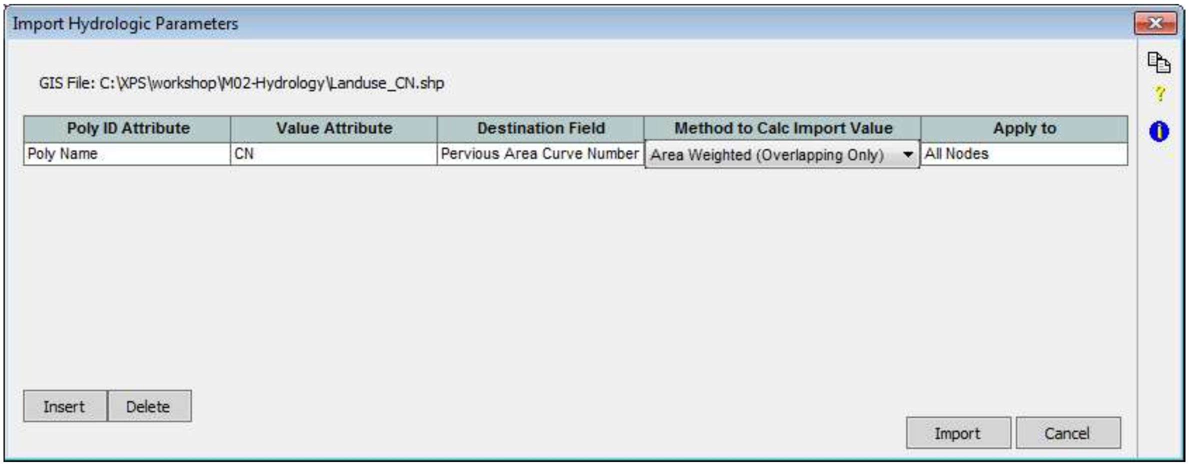
4. Turn on the Quick Data View. Select the **Quick Data View** icon  to enable the quick data view window.

5. With the pointer tool **select** the various **polygons** of the highlighted GIS Files layer. Using **<Ctrl>** and mouse click more than one object can be selected.



6. Import Hydrologic Parameters. A specialized tool has been developed to generate some hydrologic parameters by intersecting the linked catchments with GIS files. **Right-click** on the **GIS Files->Landuse_CN.shp** file layer and choose **Import Hydrologic Parameters...**

7. Complete the table as shown below by Selecting **Poly Name**, **CN**, **Pervious Area Curve Number**, **Area Weighted (Overlapping Only)** and **All Nodes** from left to right.



8. Select **Import**. A report should appear in the default editor with each of the polygon intersects reported as a percentage and the final derived value. These values are now assigned to the nodes. **Save** the model.

Questions

Please answer the following questions and we will review the answers together.

1. In regards to the DTM used in this exercise, what are the:

   Number of suvery points _____

   Minimum elevation _____ ft

   Maximum elevation _____ ft

2. Open the File menu, select **Properties** and click on **Job Statistics**. In the current column, what are the number of:

   _____ links

   _____ nodes

   _____ maximum number of nodes

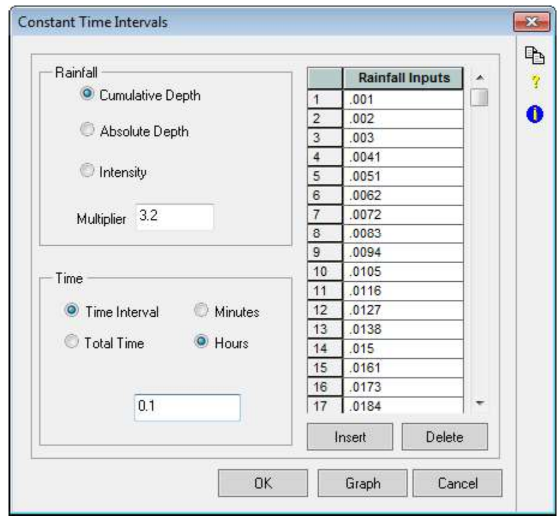
3. How many catchment polygons are in the stormwater model? _______________ catchments

4. Which node and subcatchment number has the largets area? _________________

5. How many ctachments can be assigned to a node? ____________________________

Adding Design Storms with SCS Hydrology

In xp, design storms and rainfall hyetographs can be imported by a variety of methods. In the United States, a set of commonly used design storms are the SCS 24-hour rainfall distributions. This section demonstrates how SCS rainfall distributions are scaled into a design storm. Our template already has the SCS rainfall distributions but these could also be imported to any model file using an XPX text file. These files allow the import of external data into the XP database.



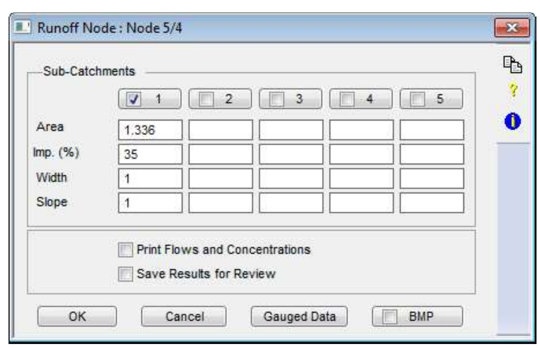
SCS Hydrology

1. Creating the 3.2 inch – 24 hour design storms from the SCS Type II distribution. On the **Configuration** menu, select **Global Data**. In the left panel select **(R) Rainfall**. In the right panel select **SCS Type II**. Click on **Duplicate**. A new storm "**SCS Type II.1**" is created. In the editing box below the record list, change the name to **5yr-24Hr SCS Type 2**. Click on **Rename**.

2. Click on the **Edit** button to open the (R) Rainfall dialog. Select **Constant Time Intervals**. Enter **3.2** as the **Multiplier** and click on **Graph** to view the rainfall hyetograph.

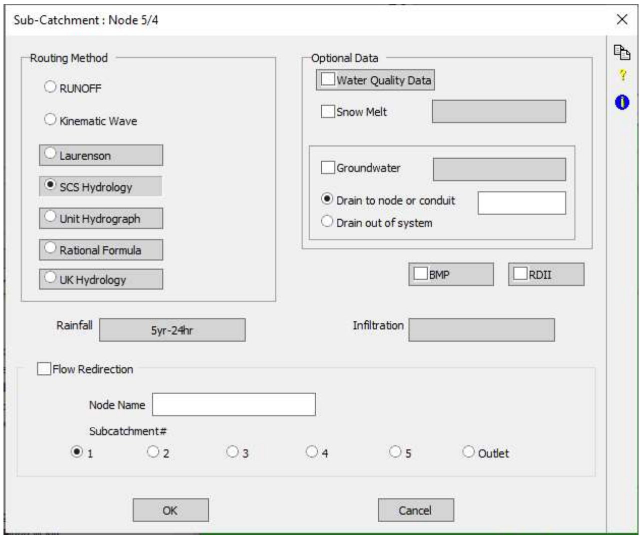
   Note: The graph always displays intensity vs. time.

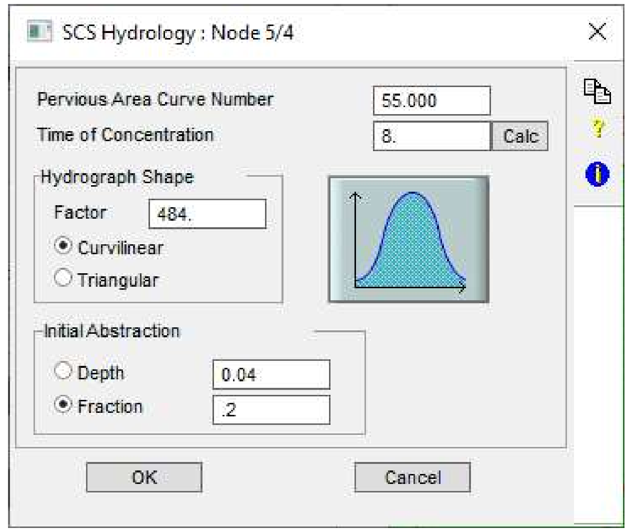
   Click on **Close** and then **OK** three times to return to the network view.



3. Enter the remaining Hydrology data. Double-click on node **5/4** or its catchment polygon to open the Runoff Node dialog. The catchment area and percent impervious were previously imported. The Width and Slope are not used in SCS Hydrology. However, xp requires that these fields have positive nonzero values. We will use the value of **1** and **1** for both the **Width** and **Slope**.

4. **Double-Click** on the **Sub-Catchments 1** flag to activate the subcatchment and advance to the Subcatchment dialog. This dialog allows the selection of the Hydrologic Routing Method and supporting parameters and the Rainfall record to be applied to this catchment.

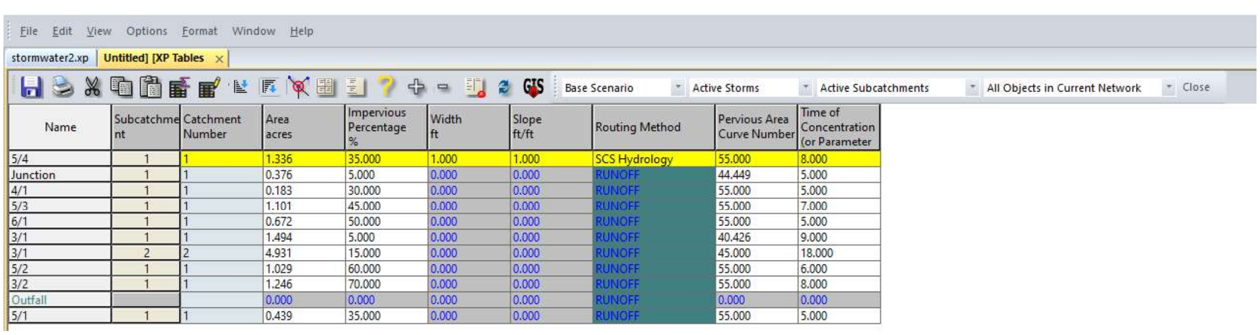




5. Click on the **SCS Hydrology** button. The values of **55** and **8** respectively for the Pervious Area Curve Number and Time of Concentration were imported and derived from GIS attributes. Click **OK** to use the Hydrograph Shape Factor default value of 484 and other defaults such as initial abstraction values. Click on **OK**.

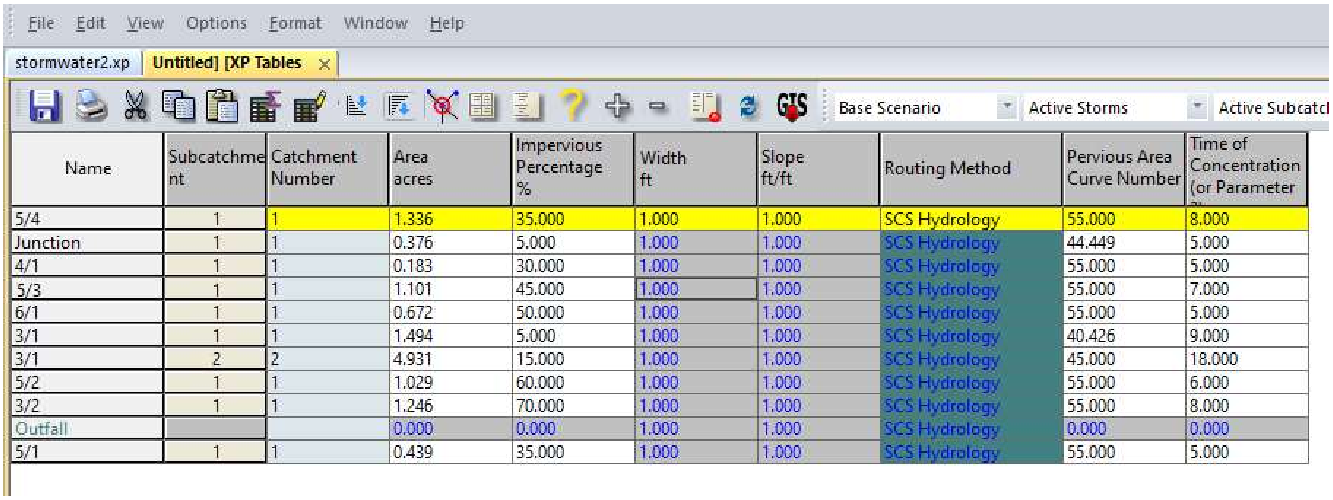
6. Click on the **Rainfall** button. Select the **5yr-24hr** storm from the Global Database and click on the **Select** button. Copy the record using the Copy Icon in the upper corner of the dialog. Click on **OK** two times to return to the network view. Select all the nodes and then Paste so they all have the rainfall record.

7. We could repeat the selection of each node and following the above procedure, enter the data in the table below for the remaining runoff nodes. However, using **XP Tables** data can be entered more efficiently. Click on the **XP Tables** icon or press **F2**. Select the **Physical Hydrology** table (using tabs at the lower right-hand corner). Use the All Objects in Current Network combo box option as Opposed to Selected to see all object rows.



Name	Subcatchment	Catchment Number	Area acres	Impervious Percentage %	Width ft	Slope ft/ft	Routing Method	Pervious Area Curve Number	Time of Concentration (or Parameter
5/4	1	1	1.336	35.000	1.000	1.000	SCS Hydrology	55.000	8.000
Junction	1	1	0.376	5.000	0.000	0.000	RUNOFF	44.449	5.000
4/1	1	1	0.183	30.000	0.000	0.000	RUNOFF	55.000	5.000
5/3	1	1	1.101	45.000	0.000	0.000	RUNOFF	55.000	7.000
6/1	1	1	0.672	50.000	0.000	0.000	RUNOFF	55.000	5.000
3/1	1	1	1.494	5.000	0.000	0.000	RUNOFF	40.426	9.000
3/1	2	2	4.931	15.000	0.000	0.000	RUNOFF	45.000	18.000
5/2	1	1	1.029	60.000	0.000	0.000	RUNOFF	55.000	6.000
3/2	1	1	1.246	70.000	0.000	0.000	RUNOFF	55.000	8.000
Outfall			0.000	0.000	0.000	0.000	RUNOFF	0.000	0.000
5/1	1	1	0.439	35.000	0.000	0.000	RUNOFF	55.000	5.000

8. Select **Active Subcatchments** in the subcatchment pull down above the top of the table. Now type, or copy (ctrl-c) and paste (ctrl-v), values of **1** for all **Widths** and **Slopes**. Ensure all **Subcatchment Flags** except for node Outfall are ON, select **SCS Hydrology** and the Rainfall Reference **5yr-24hr.** When you are finished, click on the **Save** button and then on the **Close** button.



Name	Subcatchment	Catchment Number	Area acres	Impervious Percentage %	Width ft	Slope ft/ft	Routing Method	Pervious Area Curve Number	Time of Concentration (or Parameter
5/4	1	1	1.336	35.000	1.000	1.000	SCS Hydrology	55.000	8.000
Junction	1	1	0.376	5.000	1.000	1.000	SCS Hydrology	44.449	5.000
4/1	1	1	0.183	30.000	1.000	1.000	SCS Hydrology	55.000	5.000
5/3	1	1	1.101	45.000	1.000	1.000	SCS Hydrology	55.000	7.000
6/1	1	1	0.672	50.000	1.000	1.000	SCS Hydrology	55.000	5.000
3/1	1	1	1.494	5.000	1.000	1.000	SCS Hydrology	40.426	9.000
3/1	2	2	4.931	15.000	1.000	1.000	SCS Hydrology	45.000	18.000
5/2	1	1	1.029	60.000	1.000	1.000	SCS Hydrology	55.000	6.000
3/2	1	1	1.246	70.000	1.000	1.000	SCS Hydrology	55.000	8.000
Outfall			0.000	0.000	1.000	1.000	SCS Hydrology	0.000	0.000
5/1	1	1	0.439	35.000	1.000	1.000	SCS Hydrology	55.000	5.000

9. Save your file as **stormwater3.xp.**

Questions

Please answer the following questions; we will also review the answers together.

1. In regards to the 5yr-24 storm used in this exercise, what is the:

   Total rainfall ____ in.

   Maximum intensity ____ in/hr

   Time of Maximum intensity ____

2. Must the rainfall to be the same over the entire network? Yes or No

3. Can xptables show only 1 index of subcatchments such as Number 2? Yes or No

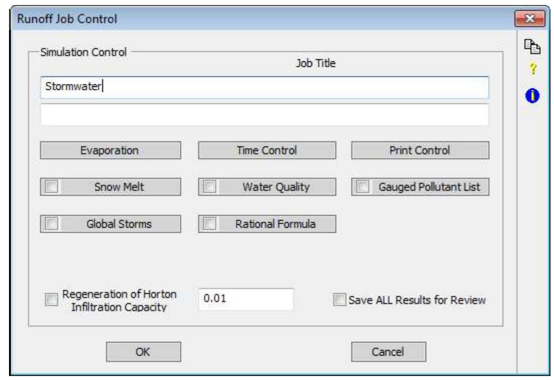
Job Control Settings & Running the Model

Settings for running the model are managed in the Job Control dialog. This section reviews some of the Job Control settings for the Runoff mode.

Job Control Settings

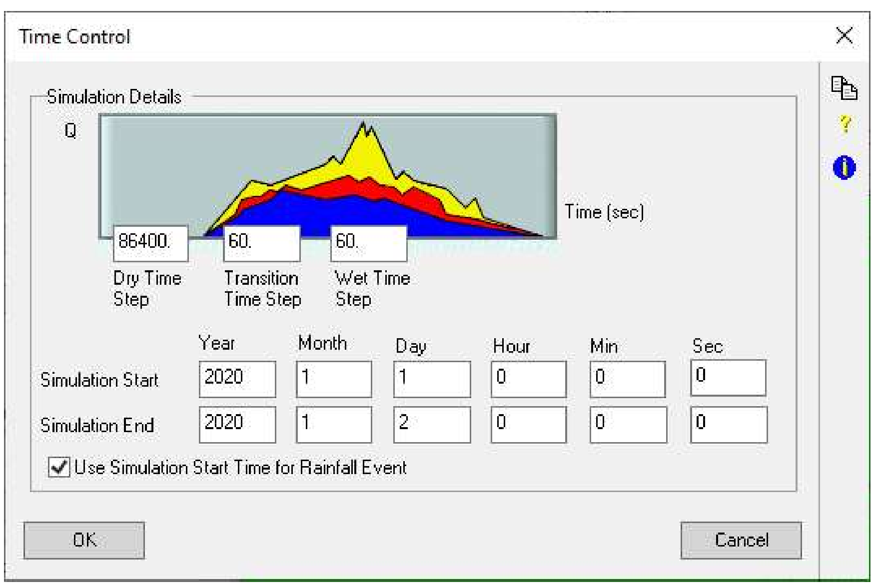
1. Open the file **stormwater3.xp** if not already open.

2. On the Configuration menu, select **Job Control → Runoff**. Enter the optional text in the **Job Title** field.



3. Click on **Evaporation**. Evaporation data can be entered as either daily or monthly values, or a default value of 0.1 inch/day. Select the default value of **0.1** inch/day. Click on **OK**.

4. In the Runoff Job Control dialog, click on **Time Control**. Check the box next to **Use Simulation Start Time for Rainfall Event**. Click on **OK**. We will use the default dates and duration.



5. Use the respective default time steps of **86400**, **60** and **60** seconds for Dry Time Step, Transition Time Step and Wet Time Step.

6. Save your file as **stormwater4.xp**.

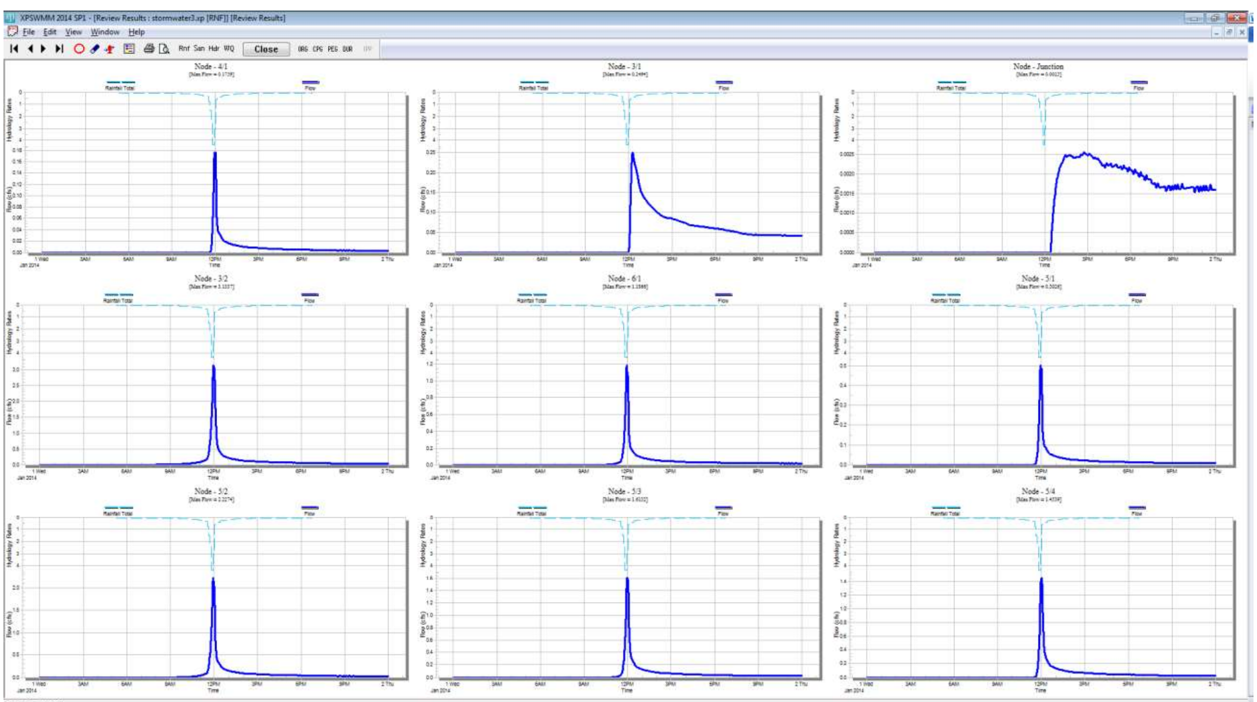
Running the analysis and Reviewing results

1. Run the analysis. Click on the solve icon or select **Analyze->Solve** from the menus or press the **F5** key. A default name for the output (**stormwater4.out**) will be produced. The analysis engine will launch if there are no errors. Errors will be displayed in the assigned default text editor of the program. The engine will display a dialog indicating the status of the calculation such as percent complete. When the simulation is completed, user control returns to the main user interface.

Note: If there are errors they will need to be fixed before a "Solve" can be performed. Following the above procedures should produce 1 error related to node 3/1 subcatchment 2 does not have selected rainfall. Fix the error in order to proceed.

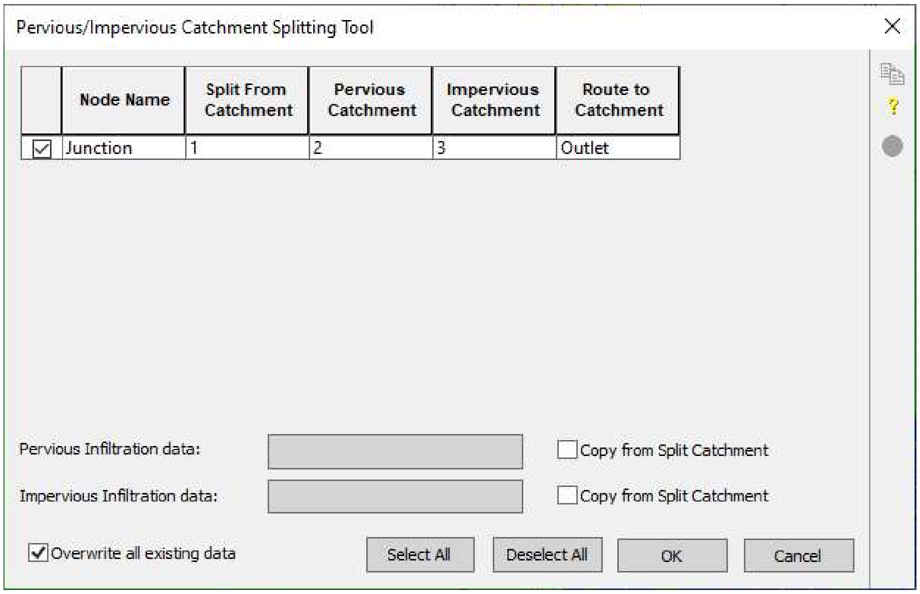
2. Select all the nodes with subcatchments by holding down the left mouse button and dragging a box around the active nodes. Click on the **Review Results** tool. The program will display graphs of rainfall and runoff for each of the selected nodes.

3. **Save** your file.

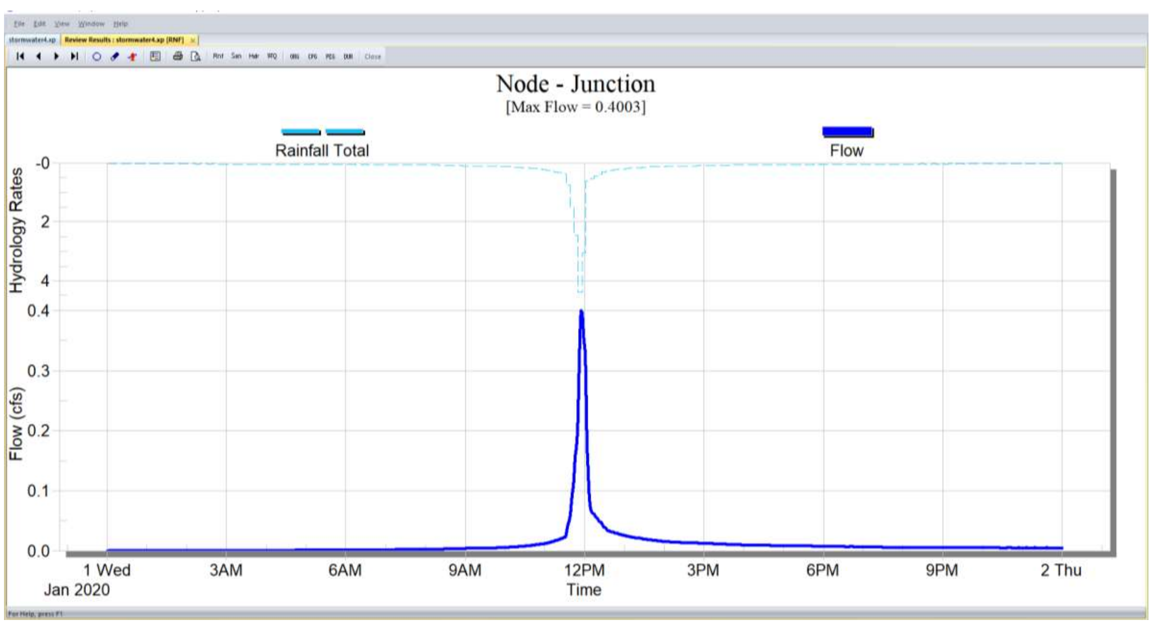


4. At the Junction node almost all the rainfall becomes infiltrated, but it is 5% directly connected impervious. A fault of the SCS method is that all subcatchments are homogenous and an area weighted curve number is derived using CN=98 for the impervious portion. In order to model directly connected impervious with SCS the impervious and pervious parts of the subcatchment should be split.

5. **Select** only the **Junction** node. Use the **Tools->Calculate Node->Split Catchments Into Perv/Imperv, check** the **box** for the row and for **Overwrite existing data.**



6. Make this change or **Open** the file **stormwater4a.xp**. **Solve** and **Review Results** to see the new hydrograph representing some directly connected impervious area.



Questions

Please answer the following question and we will review the answers together.

1. Many other nodes do not show flow until just before 12PM, but it had been raining for the previous 12 hours, can you explain why?

Using SWMM Hydrology

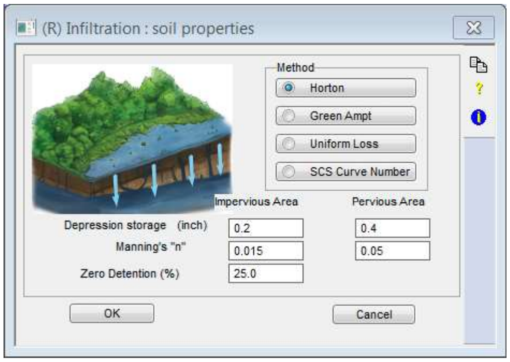
Another popular routing procedure is the EPA SWMM non-linear runoff method. Overland flow hydrographs are generated by a routing procedure using Manning's equation and a lumped continuity equation. Surface roughness and depression storage for pervious and impervious area parameters further describe the catchment. The subcatchment width parameter is related to the collection length of overland flow and is easily calculated based on the watershed properties. The method can include infiltration modeled with the Horton or Green-Ampt equations, using a uniform loss rate, or SCS loss method.

Using SWMM Hydrology

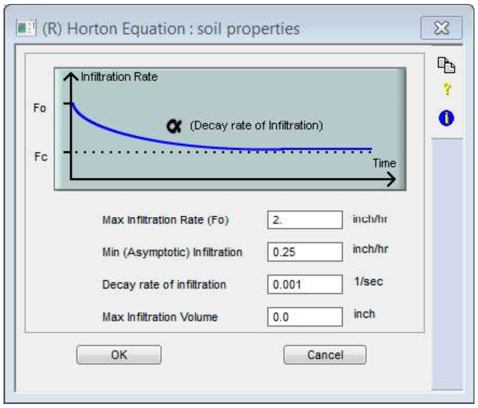
1. Open the file **stormwater4a.xp**

2. In the network view double-click on **Node 5/4** to open the data dialog. Enter **300** ft for the width and **0.02** ft/ft for the slope in the Subcatchment 1 column. **Double-click** on the **1** button to open the Subcatchment dialog. In **Routing Method**, select the **Runoff** radio button.

3. Click on the **Infiltration** button to open the Infiltration Global Database list. Type **soil properties** in the blank box and then click on **Add**. With the soil properties record highlighted, click on **Edit**. Enter the Impervious (Depression storage = **0.2** in, Manning's n = **0.015**, and Zero Detention = **25**%) and Pervious Area (Depression storage = **0.4** in and Manning's n = **0.05**) data as shown.



4. Click on the **Horton** button and in the Horton dialog enter Maximum Infiltration Rate = **2** in/hr, Minimum rate = **0.25** in/hr and Decay Rate = **0.001** 1/sec. Click on **OK** twice.

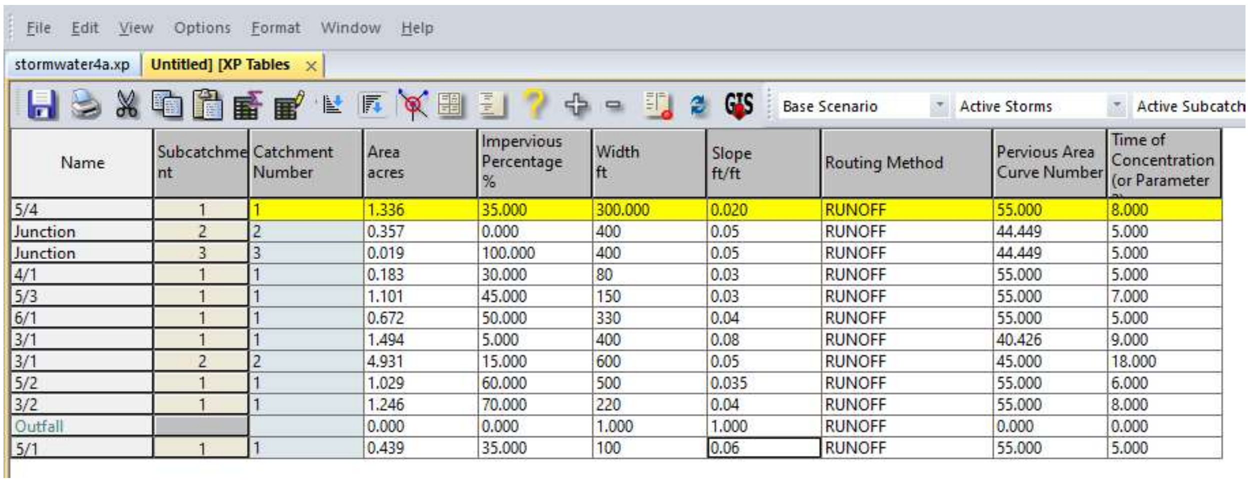


5. In the Global Database list for Infiltration, highlight **soil properties** and click on **Select**, returning to the Subcatchment dialog with **soil properties** listed on the Infiltration button. Click **OK** twice to return to the network graphic.

6. Select the **XP Tables** icon and check that you are viewing the **Physical Hydrology** table. Then change the routing method to **RUNOFF** using Copy/Paste or the Block Edit tool.

7. Enter Width and Slope data for all catchments. Pay attention to the node name and the data order to ensure the data is entered in the correct node. When you are finished, click on the **Save** button and then on the **Close** button. You can also copy and paste from the supplied Excel file the parameters listed below. The file is **stormwater.xls**.

Node	***Catchment***	***Width, ft***	***Slope, ft/ft***
5/4	1	300	0.02
Junction	2	400	0.05
Junction	3	400	0.05
4/1	1	80	0.03
5/3	1	150	0.03
6/1	1	330	0.04
3/1	1	400	0.08
3/1	2	600	0.05
5/2	1	500	0.035
3/2	1	220	0.04
Outfall	n/a	n/a (1)	n/a (1)
5/1	1	100	0.06



Name	Subcatchment	Catchment Number	Area acres	Impervious Percentage %	Width ft	Slope ft/ft	Routing Method	Pervious Area Curve Number	Time of Concentration (or Parameter
5/4	1	1	1.336	35.000	300.000	0.020	RUNOFF	55.000	8.000
Junction	2	2	0.357	0.000	400	0.05	RUNOFF	44.449	5.000
Junction	3	3	0.019	100.000	400	0.05	RUNOFF	44.449	5.000
4/1	1	1	0.183	30.000	80	0.03	RUNOFF	55.000	5.000
5/3	1	1	1.101	45.000	150	0.03	RUNOFF	55.000	7.000
6/1	1	1	0.672	50.000	330	0.04	RUNOFF	55.000	5.000
3/1	1	1	1.494	5.000	400	0.08	RUNOFF	40.426	9.000
3/1	2	2	4.931	15.000	600	0.05	RUNOFF	45.000	18.000
5/2	1	1	1.029	60.000	500	0.035	RUNOFF	55.000	6.000
3/2	1	1	1.246	70.000	220	0.04	RUNOFF	55.000	8.000
Outfall			0.000	0.000	1.000	1.000	RUNOFF	0.000	0.000
5/1	1	1	0.439	35.000	100	0.06	RUNOFF	55.000	5.000

8. Select the **Rain + Infiltration** table. **Block Edit** the string "soil properties" for Infiltration Reference for all catchments. You could also simply **Copy** and **Paste**.

9. Save your file as **stormwater5.xp**. **Solve** the model. On the **Analyze** menu, select **Solve**. If errors or warnings are produced, they will be displayed. If no errors or warnings are produced, the error log does not appear. The engine dialog will appear, and the model will be solved.

10. After a successful solve **Save**. This ensures model results are permanently in the .xp database.

Using graphical tools to obtain data from catchment parameters

1. Using graphical interface tools to measure horizontal and vertical distances and areas. To measure a distance select the **Ruler** tool . Click to begin a measurement. Click to add a vertex and line segment. Double-click to end the measurement. The current distance is the length of the last line segment. Total Distance indicates the length of the polyline. To measure an area, draw a closed polygon by selecting the first point. Slope is also indicated along the segment and the total line.

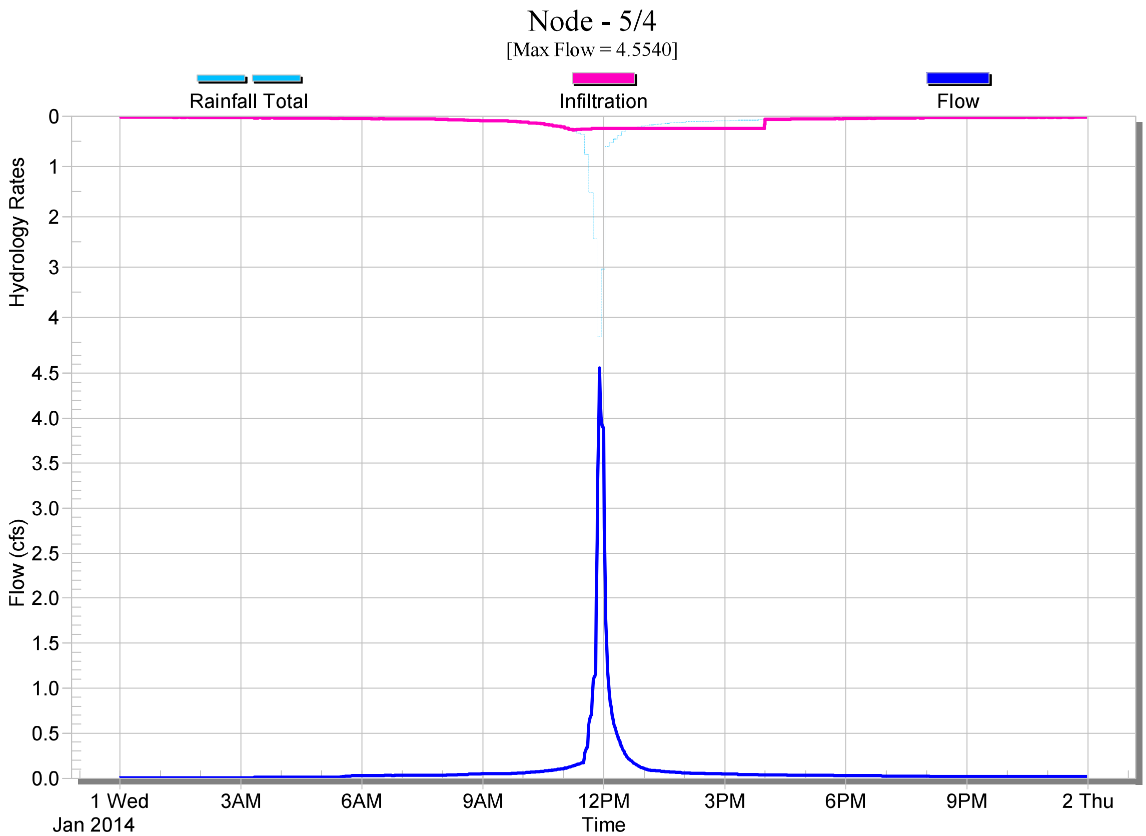
2. To visualize the subcatchment slope, use the **Ruler** tool or Section **Profile** tool . The cursor will show a drawing polyline. Move the cursor to the start point. Click to begin. Move the cursor to a new location. Click to add a vertex. Double click to end. The program displays the cross section of the drawn polyline.

3. To determine the subcatchment width, one can either measure the average width using the Ruler tool or measure the drainage area and the longest flow path. The first method would be an estimate of the width at the average distance using a visual estimate of the location. The second method would be the calculated area divided by the measured flow path or the measured flow path times 2 if surface flows are captured on both sides.

Review results

1. **Select** node **5/4**. Right-click and choose **Review Results** or the icon . A set of hydrographs will display the rainfall and runoff for this node. Click on the **Properties tool** to open the properties dialog. In the Show section, select **1 Graph per Page** from the drop list. Check **Infiltration** in the Hydrology Rates section. Click on **OK**.

2. With the cursor anywhere on the graph grid, **right-click** to reveal a popup menu for graph customization and export options.



Questions

Please answer the following questions and we will review the answers together.

1. In regards to the results for Node 5/4, what is the

   Maximum infiltration rate _____ in/hr

   Maximum rainfall intensity _____ in/hr

2. Why does infiltration peak before the maximum rainfall?

   ____

3. In this simulation the impervious and pervious surfaces generate individual hydrographs that are summed together and presented as the node flow. This is very different than the SCS in that there is a small amount of flow during the low intesity rainfall period before 12PM. Can you think of a way to accomplish this with SCS?

   ____

Reviewing the Model using XP Tables and the Output File

A variety of tools are available for examining model results. We have already looked at the review results portion. Let's look at XP Tables and the output file.

Reviewing a model using XP Tables

1. Open the file **stormwater5.xp** if not already opened.

2. Since the model was solved earlier we do not need to solve it again to generate results.

3. Click the XP Tables icon. Use the arrows at the lower left corner of the screen to navigate to the **Rain + Infiltration** sheet. Cells with light grey shading cannot be edited. Data in cells with a white background is input data.

   Note: The rainfall reference and infiltration can be edited as a drop list from the Global Database.

4. Use the arrows at the lower corner of the screen to navigate to other tables. Select the **Subcatchment Results** table. Data with grey shading are results which cannot be edited. Data with a white background is input data. In the Name column, nodes that are inactive are displayed with a grey text.

5. To sort the table by highest flow, right-click on the Max Flow cfs column and select the sort descending. This will order nodes based on flow. **Close** the XP Tables window.

Reviewing the Output File.

1. On the **Analyze** menu, select **Show Output Logs->1D Log** to open **stormwater5.out** in the default text editor. This file is generated every time the model is solved. It contains information regarding the settings, input data and results. This information is useful for debugging, calibrating and obtaining detailed model results. Selected sections are described here.

2. The beginning section presents information about the software version and the input data file. Users are encouraged to use the latest version of the program.

```
*==============================================*
|                   xpswmm                      |
|     Storm and Wastewater Management Model     |
|         Developed by XP Software Inc.         |
|==============================================|
|                                              |
| Last Update      : June, 2014                |
| Interface Version: 2012                      |
| Engine Version   : 12.0                      |
| Data File Version: 12.6                      |
|                                              |
|                                              |
*==============================================*
```

3. The tables produced by a runoff analysis are listed below.

```
*============================================================*
|        RUNOFF TABLES IN THE OUTPUT FILE.                   |
|  These are the more important tables in the output file.   |
|  You can use your editor to find the table numbers,        |
|  for example: search for Table R3 to check continuity.     |
|  This output file can be imported into a Word Processor    |
|  and printed on US letter or A4 paper using portrait       |
|  mode, courier font, a size of 8 pt. and margins of 0.75   |
|                                                            |
| Table R1  - Physical Hydrology Data                        |
| Table R2  - Infiltration data                              |
| Table R3  - Raingage and Infiltration Database Names       |
| Table R4  - Groundwater Data                               |
| Table R5  - Continuity Check for Surface Water             |
| Table R6  - Continuity Check for Channels/Pipes            |
| Table R7  - Continuity Check for Subsurface Water          |
| Table R8  - Infiltration/Inflow Continuity Check           |
| Table R9  - Summary Statistics for Subcatchments           |
| Table R10 - Sensitivity analysis for Subcatchments         |
*============================================================*
```

4. Table R5 contains the continuity check and basin wide results for various runoff parameters.

```
************************************************************
* Table R5. CONTINUITY CHECK FOR SURFACE WATER             *
*       Any continuity error can be fixed by lowering the  *
*       wet and transition time step. The transition time  *
*       should not be much greater than the wet time step. *
************************************************************
                                                       Inches over
                                          cubic feet   Total Basin
 Total Precipitation (Rain plus Snow)     1.487661E+05     3.200
 Total Infiltration                       7.528257E+04     1.619
 Total Evaporation                        4.648941E+03     0.100
 Surface Runoff from Watersheds           6.673215E+04     1.435
 Total Water remaining in Surface Storage 2.171484E+03     0.047
 Infiltration over the Pervious Area...   7.528257E+04     2.310
                        --------
 Infiltration + Evaporation +
 Surface Runoff + Snow removal +
 Water remaining in Surface Storage +
 Water remaining in Snow Cover.........   1.488351E+05     3.201
 Total Precipitation + Initial Storage.   1.487661E+05     3.200

  The error in continuity is calculated as
   ***************************************
   * Precipitation + Initial Snow Cover  *
   *      - Infiltration -               *
   *Evaporation - Snow removal -         *
   *Surface Runoff from Watersheds -     *
   *Water in Surface Storage -           *
   *Water remaining in Snow Cover        *
   *-------------------------------------*
   * Precipitation + Initial Snow Cover  *
   ***************************************
```

```
Percent Continuity Error..............          -0.0464
```

5. Table R9 shows detailed runoff data for each node.

```
##################################################
# Table R9. Summary Statistics for Subcatchments #
##################################################

Note:      Total Runoff Depth includes pervious & impervious areas.         Pervious  and
Impervious Runoff Depth is only the runoff from       those two areas.
           For catchments receiving redirected flow, this flow will only     be  shown  if
the flow is not directed directly to the outlet.      Flow that is getting redirected is
also listed with the    original subcatchment.

  Subcatchment...........          4/1#1           3/2#1
  Area (acres)...........        0.18300         1.24600
  Percent Impervious.....       30.00000        70.00000
  Total Rainfall (in)....        3.20000         3.20000
  Max Intensity (in/hr)..        4.38720         4.38720

  Pervious Area
  -------------
  Total Runoff Depth (in)        0.80311         0.80392
  Peak Runoff Rate (cfs).        0.47910         1.42150

  Total Impervious Area
  ---------------------
  Total Runoff Depth (in)        2.94843         2.94202
  Peak Runoff Rate (cfs).        0.24263         3.81999

  Impervious Area with depression storage
  ---------------------------------------
  Total Runoff Depth (in)        2.17382         2.16902
  Peak Runoff Rate (cfs).        0.18197         2.86500

  Impervious Area without depression storage
  ------------------------------------------
  Total Runoff Depth (in)        0.77461         0.77300
  Peak Runoff Rate (cfs).        0.06066         0.95500

  Total Area
  ----------
  Total Runoff Depth (in)        1.44670         2.30059
  Peak Runoff Rate (cfs).        0.72173         5.24150

  Rational Formula
  ----------------
  Pervious Tc. (mins)....        0.00000         0.00000
  Perv. Intensity (in/hr)        0.00000         0.00000
  Pervious C ............        0.00000         0.00000
  Impervious Tc. (mins)..        0.00000         0.00000
  Imp. Intensity (in/hr).        0.00000         0.00000
  Impervious C ..........        0.00000         0.00000
  Partial Area (Ha)......        0.00000         0.00000
  Partial Area Tc........        0.00000         0.00000
  Partial Area Intensity.        0.00000         0.00000
```

The output file ends with notes indicating that the calculation ended successfully.

```
===> Runoff simulation ended normally.
===> XP-SWMM Simulation ended normally.
===> Your input file was named  : C:\XPS\workshop\M02-Hydrology\stormwater5.DAT
===> Your output file was named : C:\XPS\workshop\M02-Hydrology\stormwater5.out

*===============================================================*
|             SWMM Simulation Date and Time Summary             |
*===============================================================*
| Starting Date... December   29, 2014  Time...   15:14:53:64  |
|   Ending Date... December   29, 2014  Time...   15:14:54:56  |
|  Elapsed Time...    0.01533 minutes or      0.92000 seconds  |
*===============================================================*
```

Questions

Please answer the following questions and we will review the answers together.

Review the output file (**stormwater5.out**) to answer the following questions.

1. In the network, what are the areas:

   Total catchment _____ ac
   Impervious area _____ ac
   Pervious area _____ ac

2. For the storm event, what are the volumes of

   Rainfall _____ ft^3
   Runoff _____ ft^3
   Evaporation _____ ft^3
   Surface storage _____ ft^3

3. Which subcatchment had the highest peak runoff rate?

   Subcatchment _____
   Peak runoff rate _____ ft^3/s

4. Circle if it was the Pervious or Impervious Area with the largest contribution to peak flow?

5. Did that pattern hold true for all nodes? Circle Yes or No.

6. Can you explain why? ____

Advanced Surface Water Hydrology

The purpose of this section is to teach users how to use xpswmm's tools for advanced hydrologic components. You will learn how to model multiple storm events with global storms.

In Runoff mode, xpswmm simulates the rainfall, infiltration, evaporation and depression storage, for each subcatchment, and calculates the runoff to a collection node. This allows xpswmm to model many difficult hydrologic elements.

Add Global Storms

Global storms allow users to run multiple storms in both runoff and hydraulics. The user can also review multiple storms on one graph using review results. An xpswmm model includes a series of input and output files. The main file is the .xp file that holds the data for the model and references other files needed, such as background images or DTM data. The .mdb file stores the global storm data.

Open Existing Model

6. Close any current model and open the file **Stormwater_Completed.xp**.

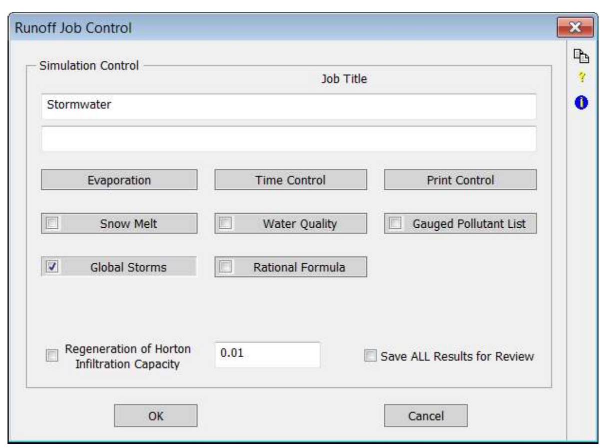
7. Save as **Stormwater6.xp**.

Set up Global Storms

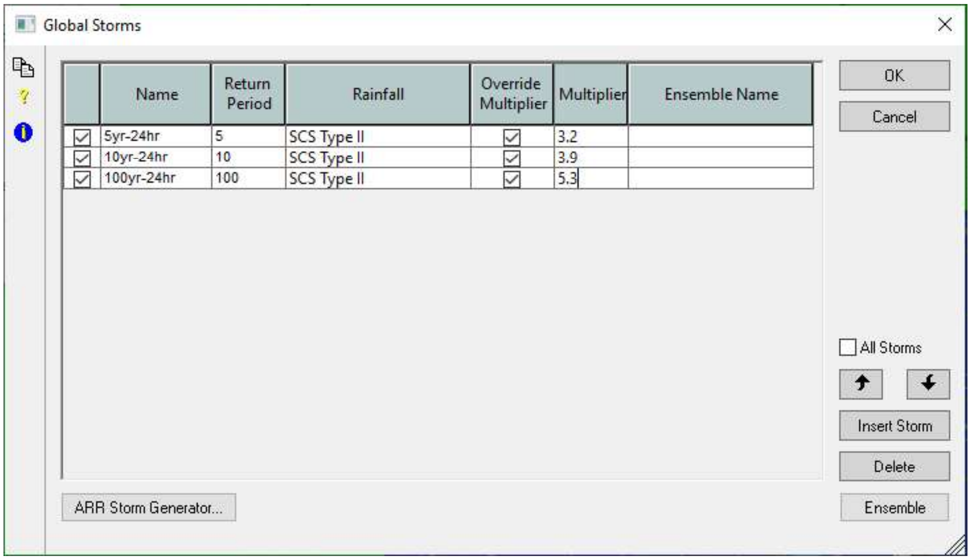
1. Confirm the Mode is **Runoff** by clicking on the icon in the Job, Layer and Mode Control tool bar.

2. Select **Configuration** in the pull down menu. Then select the **Job Control** menu followed by the **Runoff** menu item. Another option is to left click on the **Job Control** icon.

3. Select the **Global Storms** button in the Runoff Job Control dialog as shown to the right.



4. In the global storms dialog, insert three rows using the **Insert Storm** button.



5. On the first row, within the **Rainfall** column using the pull down list, select the **SCS Type II** rainfall. Repeat for the second and third row.

6. The next step defines the rainfall data for the three storms. We will be modeling the 5-year, 10-year and 100-year recurrence storm events. Type the data shown above for all three lines.

7. After the global storm data has been typed in, click on the **OK** button twice to return to the project window.

8. Note that the global storms now appear 

in the global storms tool bar. Users can choose to run one, all or any number of the listed storms.

Run Model(s) with Global Storms

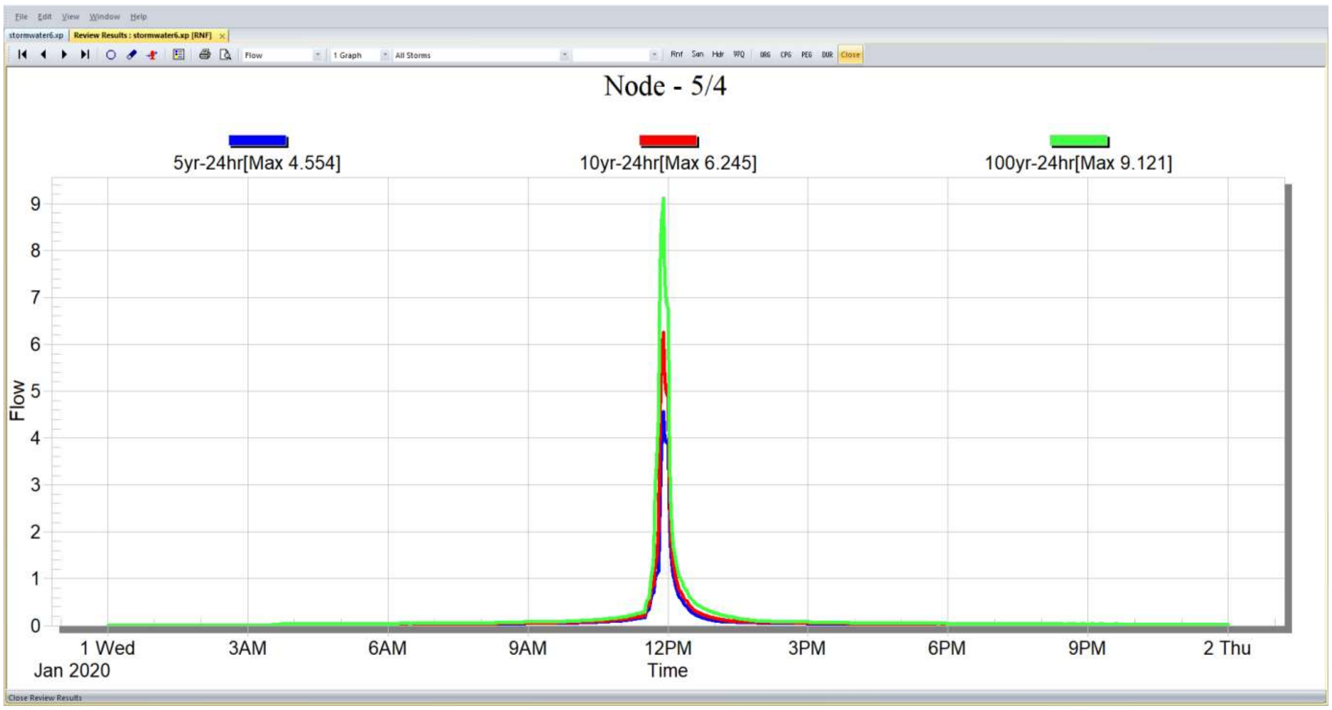
1. In Workshop No 2, the job control settings were set for the model. These settings will work for the global storm runs also. Save your file.

2. Solve the model. On the Analyze menu, select Solve Manager to allow all storms to run concurrently depending on the number of CPU cores.

3. When the models have completed choose the **Load Results** button.

Review Model with Global Storms

11. Close the Solve Manager and Engine Viewer dialogs once the Solve Manager is complete. In the **Results** menu, select **Results ->Reload 1D Results**. Within the Reload 1D Results dialog click on **Load Results**. Select node 5/4. Click on the **Review Results** tool. **xpswmm** will display the graphs of runoff for the three storms for this node. By selecting the Flow pull down, users may select other graphs for this node, such as rainfall, infiltration, evaporation, etc. In the same way, users may select one of the storms or all of the storms. Using these pull downs, select and review different data for other nodes. Close the review results by clicking on the **Close** button.



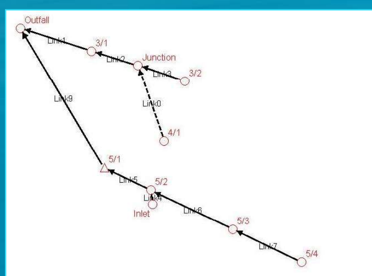
Questions

Please answer the following questions and we will review the answers together.

1. What was the highest peak runoff for Node 3/2 for the 5 year storm? ________
2. What was the highest peak infiltration rate for Node 4/1 for the 10 year storm? ________
3. What was the overall continuity error for the Yarra_24HR- 100 year model? ________

Hydraulic Analysis

Link/Node Network Flow Routing

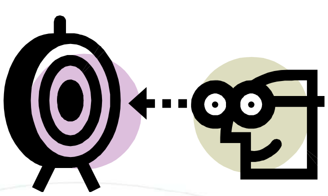


Innovyze®
Empowering water experts

1

Objectives

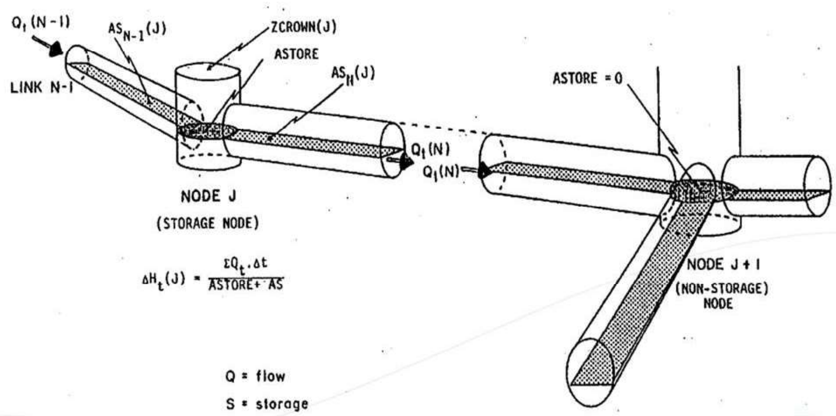
- Review SWMM (Link and Node) Hydraulics Theory
- Create a pipe model for 1D Hydraulics
- Import data from external databases
- Run a combined Runoff and Hydraulics model
- Effectively review hydraulic model results



Innovyze®

2

Link-Node Representation

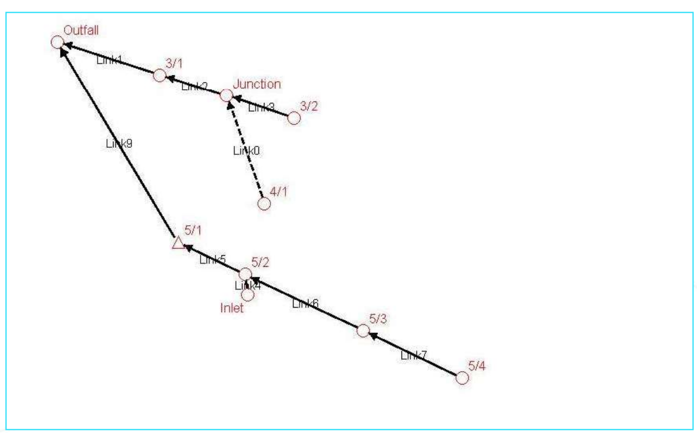


Innovyze®

3

Nodes and Links

- Nodes
  - Manhole
  - Inlet
  - Storage Node
  - Catch Basin
  - Outlet
  - Junction
- Links
  - Pipe (closed)
  - Channel (open)
  - Pump
  - Orifice
  - Weir
  - Rating Curve
  - Bridge & River Links

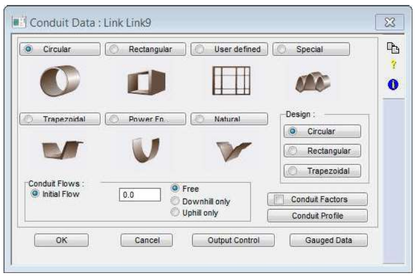


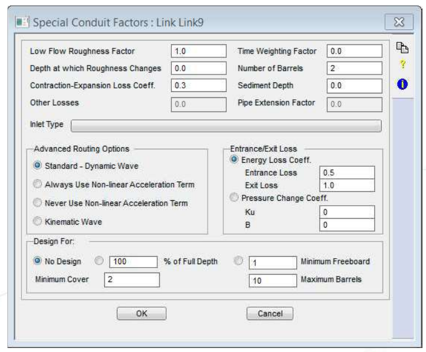
Innovyze®

4

Links – Shapes & Options

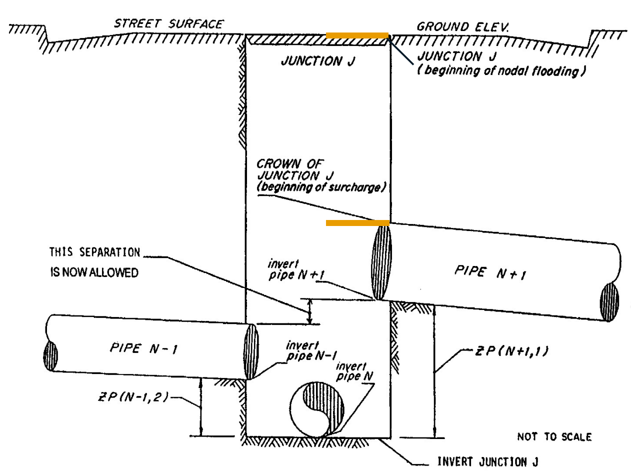
- All shapes open and closed are possible
- Special Conduit Factors such as Minor Losses, Barrels, sediment depth and Culvert Entrance





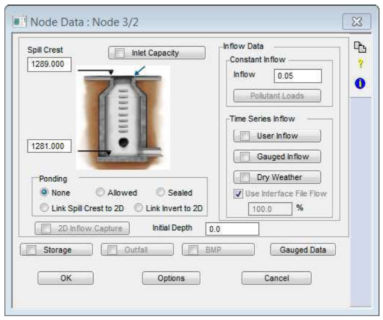
Innovyze®

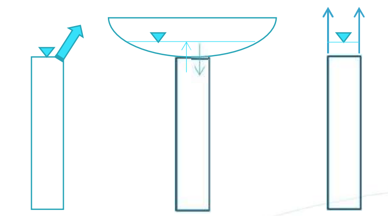
Definition of Junction Elevations



Innovyze®

Node Ponding Options





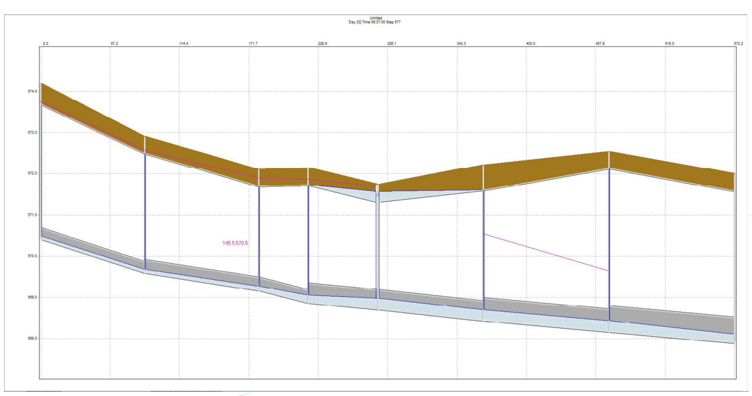
None (Lost) | Allowed (Stored) | Sealed (Pressure)

Innovyze®

7

Dual Drainage

- Simulate Major and Minor Drainage in Parallel (multilink)
- Account for restriction of curb openings and inlet grate etc.



Innovyze®

8

Dynamic Models

- CONCEPT: Use the full St. Venant equations to route flows.
- Data Needs:
  Conduit Geometry
  Conduit Inverts (slope not required)
  Node Rims and Inverts
- Limitations:
  Model Stability (sometimes)
  Smaller time step (Courant Condition some schemes)
  Geometric simplification to tackle above issues is becoming rare

Innovyze®

9

St. Venant Equations

Continuity Equation (Conserves Mass)

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0$$

Momentum Equation (Conserves "Energy")

$$\frac{\partial Q}{\partial t} + \frac{\partial \left(\frac{Q^2}{A} \right)}{\partial x} + gA\frac{\partial y}{\partial x} + gA(S_e + S_c + S_f - S_o) = 0$$

Innovyze®

10

Momentum Equation

$$\frac{\partial Q}{\partial t}+\frac{\partial\left(\frac{Q^2}{A}\right)}{\partial x}+gA\frac{\partial y}{\partial x}+gA\left(S_e+S_c+S_f-S_o\right)=0$$

Kinematic Wave

Diffusive Wave

Dynamic Wave

Local Acceleration | Convective Acceleration | Pressure Force | Friction Force | Gravity Force

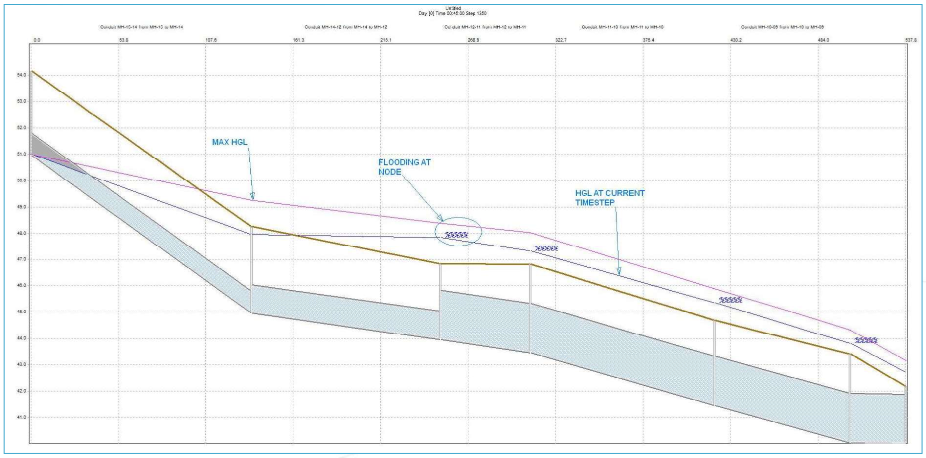
11

Hydraulic Results

- Hydraulic Grade Line (Max HGL and animation of HGL)
- Node time series:
  - Water Surface Elevations
  - Overflow
- Link Time Series:
  - Velocity and Flow
  - Upstream and Downstream Water Surface Elevation
  - Volume
- Many single valued results:
  - Losses, Freeboard, d/D, Qmax/Qdes…

12

Hydraulic Results: (HGL)

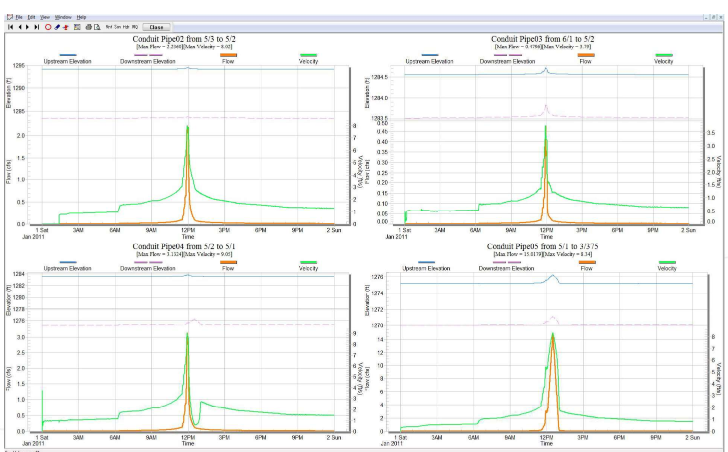


Innovyze®

13

Hydraulic Results: Levels & Flows

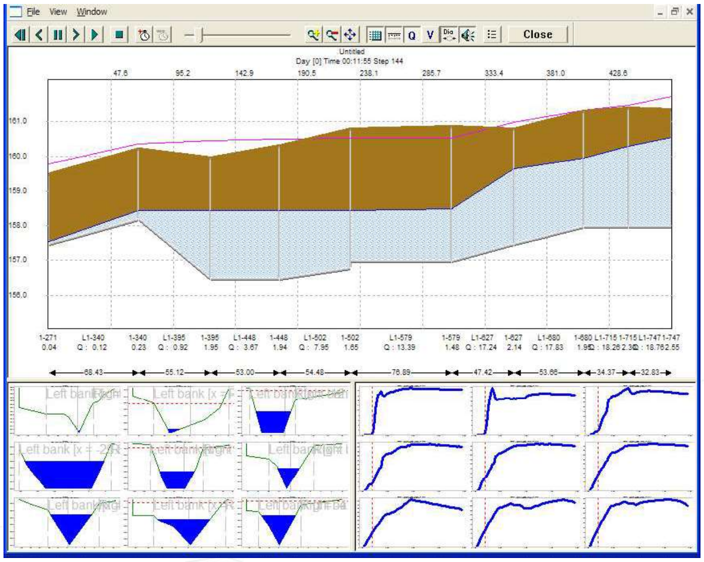
- Full Control of the plotting i.e. series, display, quantity



Innovyze®

14

1D Dynamic Results



Innovyze

15

Hydraulics: Network Creation

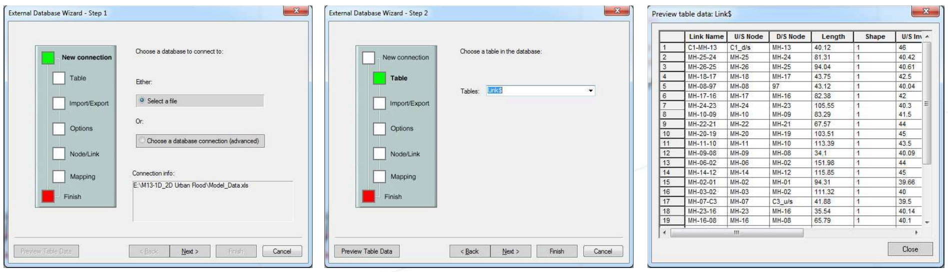
- Digitize
  - Use Node tool in the Hydraulics Layer
  - Use Link tool in Hydraulics Layer (connect nodes or nodes will be created as you create links)
- Import
  - Create nodes/links from CAD file (dwg or dxf)
  - Import LandXML or XPX or CSV text files
  - Import nodes/links from ODBC database (Shapefile or Geodatabase, Spreadsheets, databases etc.)
- Activate
  - Select Nodes/Links from Runoff Layer and activate (+ tool)
  - Links should only be active in one layer – not both!
  - Nodes maybe active in both Layers

Innovyze

16

Import/Export External Databases

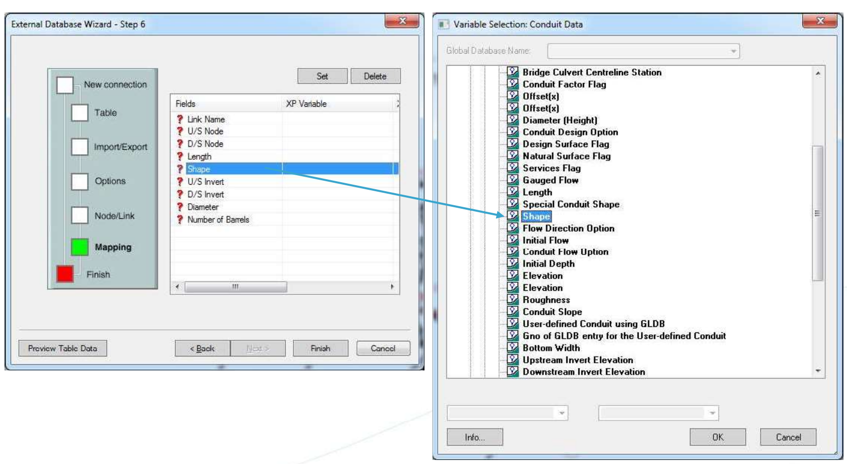
- Import and Export from Excel, Access, Dbase
- Map source fields to target fields in xp database



Innovyze®

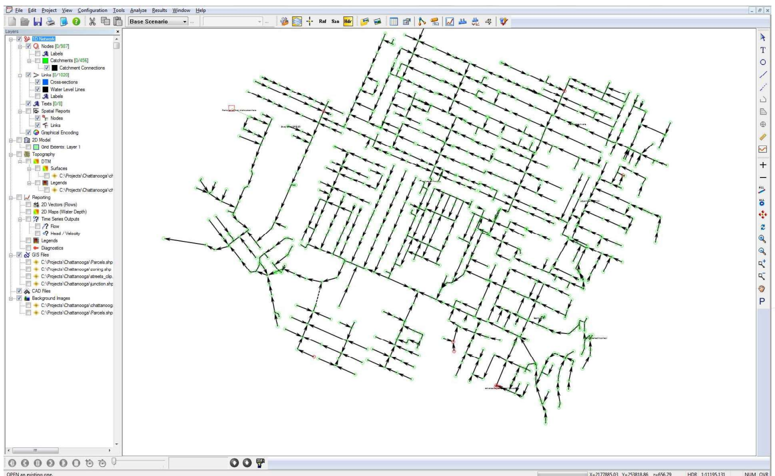
External Database Wizard

- Map data fields to XP Database definitions



Innovyze®

Network Created!



Innovyze®

19

Workshop

20

Stormwater Pipe Hydraulics

The purpose of this training module is to teach users how to use XP's tools to simulate the storage and transport or routing water through a stormwater network. Users will learn how to layout the nodes and links a in a stormwater collection network. Where possible, XP tools are used to extract data from GIS files. Dialog boxes will also be used to add data to other objects.

This workshop begins with the runoff model developed in Chapter 2 (**Surface Water Hydrology)** workshop.

Objectives

A stormwater collection network can be developed in the graphical interface using a variety of methods. In this example, users will learn how to utilize tools to build on the model started in Workshop Example 2 using various tools to add data to the model. You will learn how to:

1. Layout links and nodes in Hydraulics mode
2. Derive ground elevations and inverts from a DTM
3. Calculate pipe lengths from the model coordinate system
4. Enter required data for links and nodes in dialogs
5. Enter the basic configurations settings for solving in Hydraulics
6. Obtain results from the output file and the Review Results tool

Data files to be used are:

stormwater_completed.xp (from Chapter 2)
stormwater_hydraulics.xls
stormwater_network.xml

Adding Objects to Hydraulic Network

An existing model can be opened and changed with new or modified data. In this Module, we will start with the runoff model created in Workshop Module 2 and add the required data to route the created runoff flows through the pipe network.

Open existing model

1. Open xpswmm/xpstorm.
2. At the opening dialog, select Browse.

3. In Windows Explorer, navigate to the **XPS\Workshop\Module 3** folder and select the file named **stormwater_completed.xp**.
4. Optionally toggle the visible box for the DTM and GIS layers to improve visibility.
5. Uncheck the visible boxes for the catchments and the catchment connections layers.
6. Set the Mode to **Hydraulics** by clicking on the  icon. In Chapter 1, the nodes were created in the RNF Mode. Therefore, the nodes are all inactive. Select the nodes then press **+** to make the nodes active in HDR mode.

7. In Chapter 1 we imported nodes from GIS. Links can also be imported this way but we will use LandXML to introduce you to a way to import network data from CAD. Use **File->Import/Export Data->Import LandXML...** and select the file **stormwater_network.xml**. Click **OK** on both of the message dialogs.

8. Navigate to the **RNF** mode, select all the links and choose **–** to make them inactive. Only nodes with catchment data should be active in Runoff.

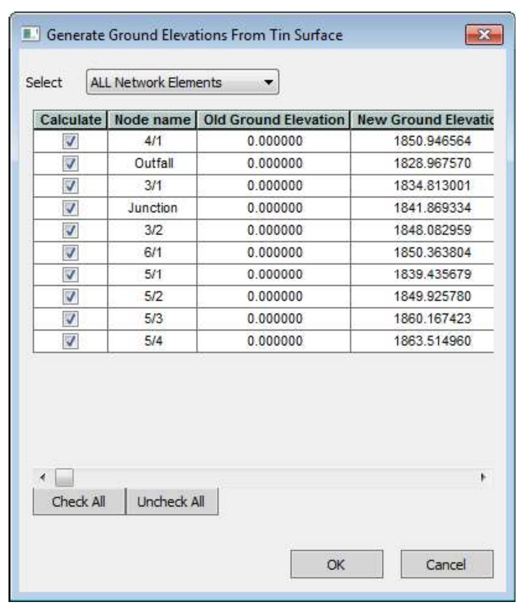
9. Use **File->Save as...** and use the new name **stormwater7.xp**.

Adding data to existing Objects

After creating the basic model, adding the data for the hydraulics mode is necessary. The data can be added in many ways; importing directly from GIS and CAD files, copying and pasting from a spreadsheet, using a .txt or .csv file or typing in the data in dialog boxes.

Generating node and link data from DTM

1. On the **Tools** menu, select **Generate Ground Elevations From TIN**.
2. Select **All Network Elements** and click on **OK**. Note: the old and new computed elevations are shown. After clicking **OK** the new elevations are added to the database.



3. To set invert elevations first select all the objects using the tools on the toolstrip or using **<Ctrl>+A** and **<Ctrl>+L** or draw a box around all the objects.

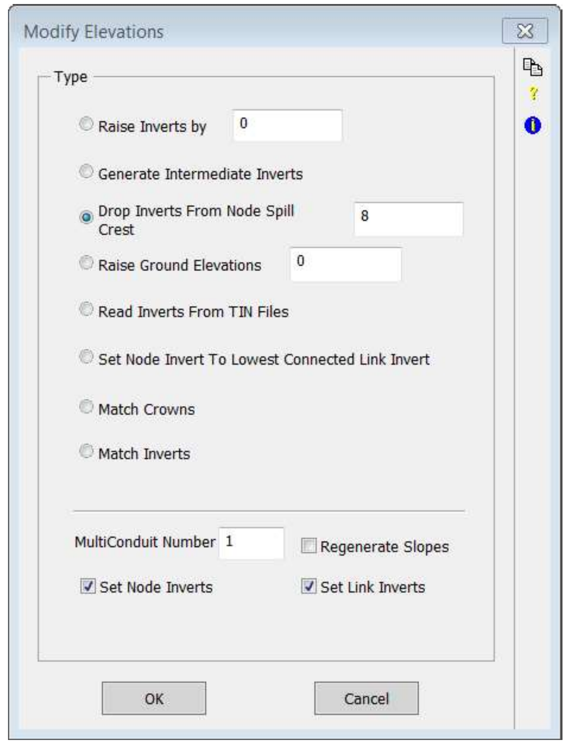
4. Click on the **Tools ->Modify Elevations**.

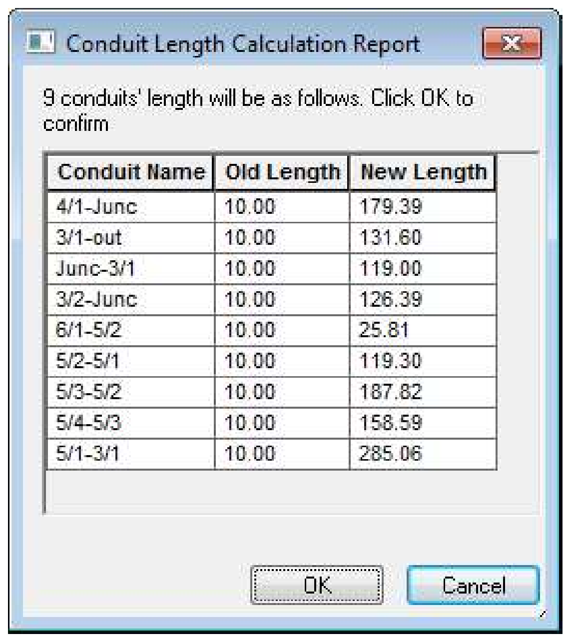
5. In the dialog box, check **Drop Inverts From Node Spill Crest** and enter **8** feet. Make sure that the check boxes for **Set Node Inverts** and **Set Link Inverts** are both active.

6. Click on **OK**. The number of links and nodes with modified inverts will be reported. Click **OK**.

7. Click on the Tools menu again and select **Calculate Conduit->Lengths**. Click on the **All** radio button to select all the conduits. Click on the **Calculate** button. See the new lengths to the right and select **OK** to accept.

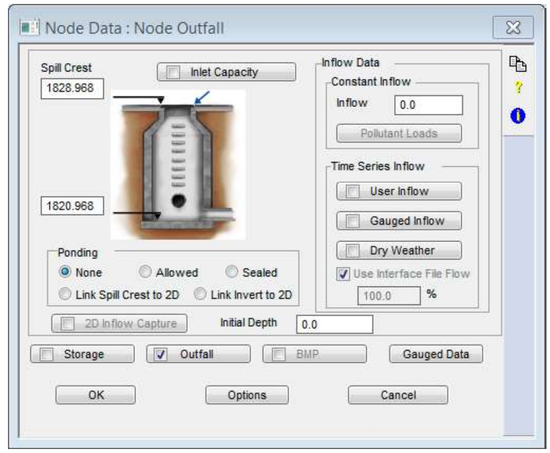
8. With inverts and link set we can now calculate conduit slopes. Conduit slopes can be calculated individually by selecting a conduit and Conduit Profile (**F3**) or for all Conduits. **Tools->Calculate Conduit->Slopes** then choose **All** and **Calculate**. Calculated slopes are shown in a similar report.





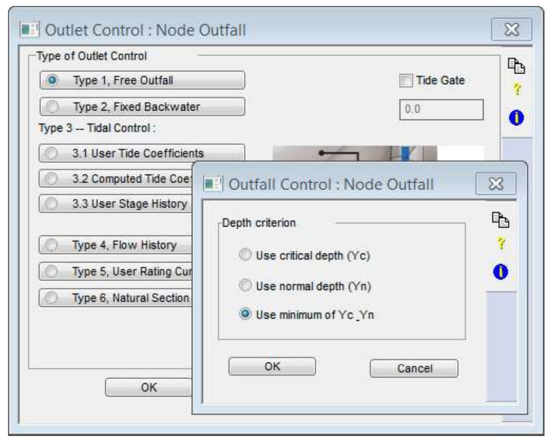
Conduit Name	Old Length	New Length
4/1-Junc	10.00	179.39
3/1-out	10.00	131.60
Junc-3/1	10.00	119.00
3/2-Junc	10.00	126.39
6/1-5/2	10.00	25.81
5/2-5/1	10.00	119.30
5/3-5/2	10.00	187.82
5/4-5/3	10.00	158.59
5/1-3/1	10.00	285.06

Entering node and link data in Dialog Boxes

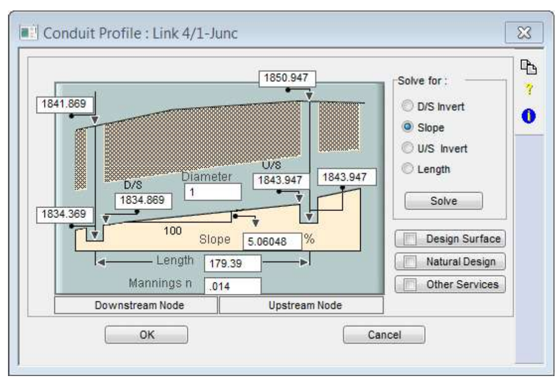


1. **Double-click** on node **Outfall** to open the node data dialog. Click on the **Outfall** button to convert the node to an outfall.

   Note: Only a terminating node in a network can be made an outfall. A disabled button means the minimum criteria for an outfall has not been met.



2. Select outlet control **Type 1, Free Outfall** and then **Use minimum of Y_c** and $\mathbf{Y_n}$ for the depth criterion. This selection of free outfall applies only the critical depth or the normal depth whichever is lower for backwater calculations in the upstream conduit. As you can see much more sophisticated boundaries are possible. Click **OK** 3 times to return to the plan view.

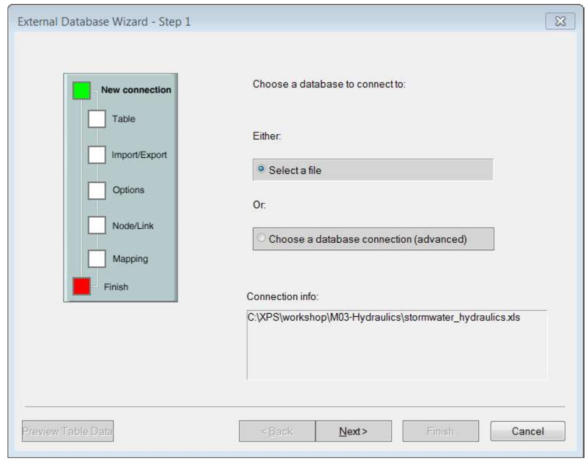


3. Double click on link **4/1-Junc**. Click on **Conduit Profile** or **F3** with the link selected. The upstream and downstream inverts and the length should have been imported in previous steps.

4. Enter **1** ft as the pipe diameter. In the Solve for: section, select **Slope** and click on **Solve**. Note: this step is not required but serves as a check. Click on the **OK** button twice.

5. In Chapter 1, we used XP Tables to input data to the model. Another way to input data is to import the data from a excel spreadsheet using Import/Export External Database procedure. We will use this to obtain conduit inverts, shape and diameter.

6. To connect to external databases to import node and link data, click on the **Tools->GIS Link...** or select **File->Import/Export Data->Import/Export External Databases**. Select **New** to launch the External Database Wizard.

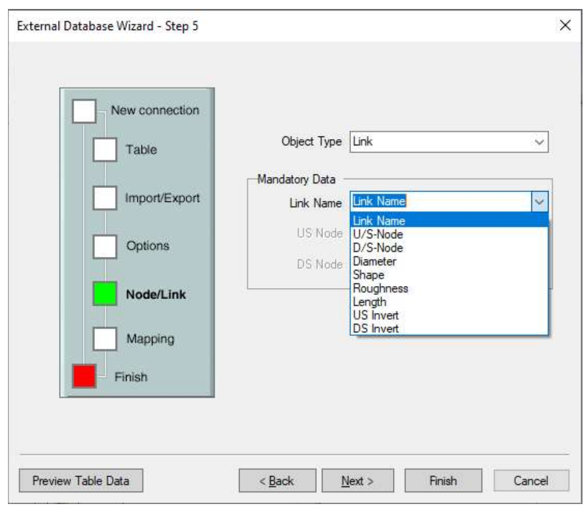


7. Click on **Select a File**. In the Open dialog, highlight **conduits.dbf** and click on **Open**. Click on **Next**. In the next screen of the wizard (step 2), select **conduits** in the Tables dropdown list. Select **Next**.

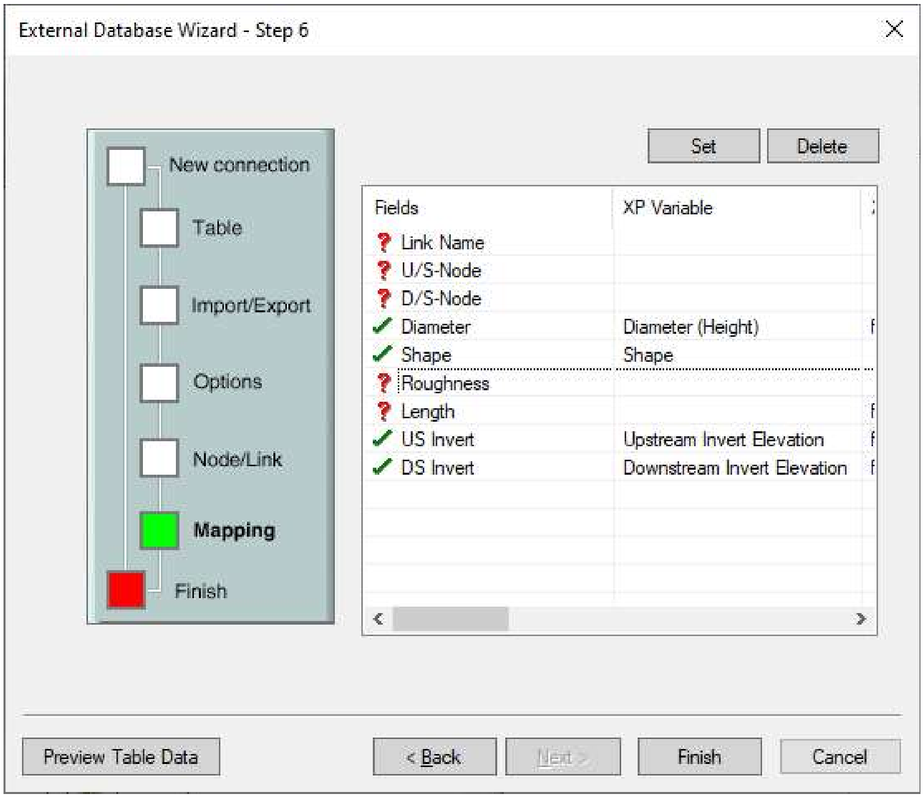
8. In step 3 of the wizard, select **Import Data Only** and select **Next**.

9. Select **Update Existing Objects only** from the drop list in step 4 and click on **Next**.

10. In step 5 of the wizard, change the **Object Type** to **Link** and then select the **Link Name** as the source field for the Link Name as shown to the right. Click on **Preview Table Data** to view the data in the table. Click on **Next**.



11. The final steps involve mapping or assigning the columns of data to variables in xp. Highlight one field at a time and select **Set** to get the **Variable Selection: Conduit Data** screen to show. Since we are importing Hydraulics mode data, open **Link**, then **Link Data**, then **Conduit Data** to find the data to select. Choose the variable that describes the Field (e.g. **Diameter, ft** → **Diameter (Height)**). Set the XP Variables for **Shape**, **US** & **DS Inverts** and **Diameter**, ft, as shown below. Select **Finish** to exit Wizard Step 6.



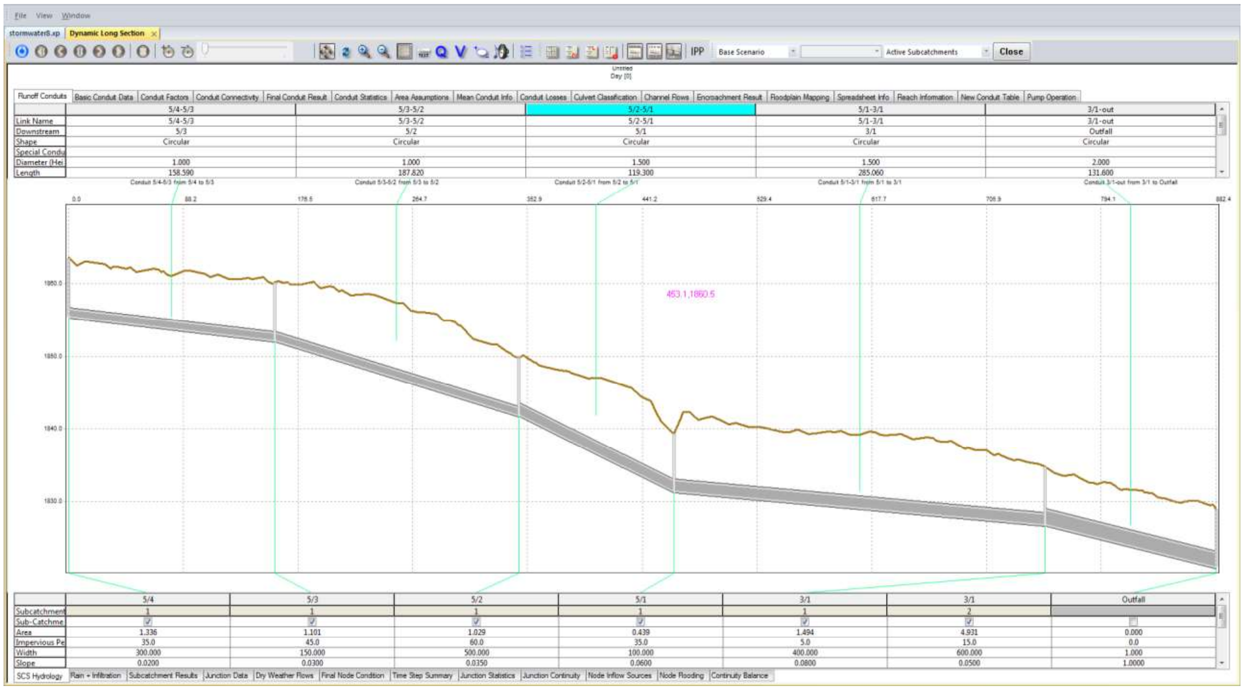
12. With **conduit** highlighted, select **Import**. A preview table will be displayed. Data may be viewed but not edited here. After reviewing the data, click on **OK.**

13. The import summary table will show that no nodes or links were created but does show that 36 variables were imported into the model database. Click on **OK**. Then click on **Close.**

14. Use **File->Save as…** and use the new name **stormwater8.xp**.

15. Use Dynamic Long Section to view the model profile. **Click** on node **5/4**. While holding the **Shift** key **click** on the node **Outfall**. This action selects a continuous set of links without branches. This is important because a selection of links including branches cannot be shown in a profile view.

16. Choose **Results->Dynamic Long Section** or **F9**. In addition, the Dynamic Section View tool from the toolstrip can also be used.



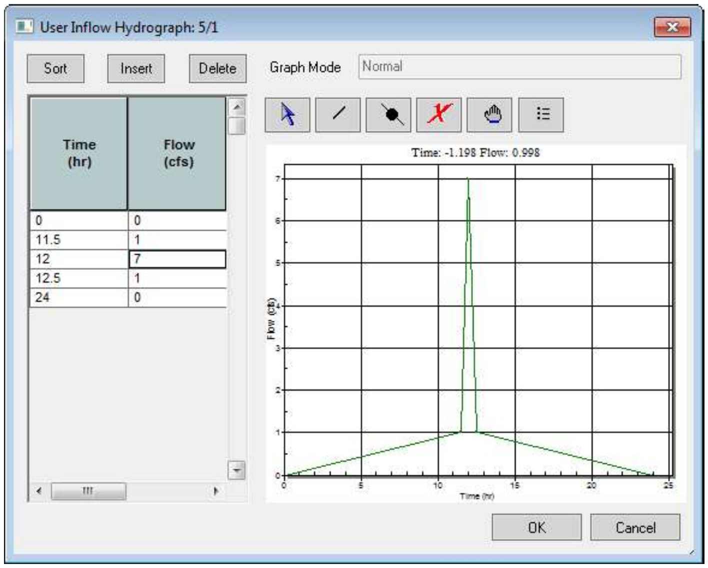
17. From the view above we can see that the inverts are matched. Using **Tools -> Modify Elevations…** we could automatically **Match Crowns** and also **Match Inverts**.

18. We imported the link upstream and downstream inverts and entered the node inverts from the DTM, it is possible that the link inverts may be lower than the node inverts. To ensure that this is not the case, we can use a tool to set the node invert to the lowest connected link invert. Using the **Select all Nodes** tool, highlight all of the nodes in the model. Then using **Modify Elevations…**, under the **Tools** pull-down menu, select the **Set Node Invert to Lowest Connected Link Invert** button and click on the **OK** button. You will see a dialog box showing the 10 Nodes were shifted.

Add a user inflow to nodes

1. Double click on node **5/1** to open the node data dialog.
2. **Click** on the **User Inflow** button in the Time Series Inflow section.
3. **Click** on the **Insert** button 4 times to add 4 additional blank data rows.

4. Input the hydrograph data as shown in the figure below. This data represents inflow from areas outside of the catchments in the model.



5. **Save** the model.

Questions

Please answer the following questions and we will review the answers together.

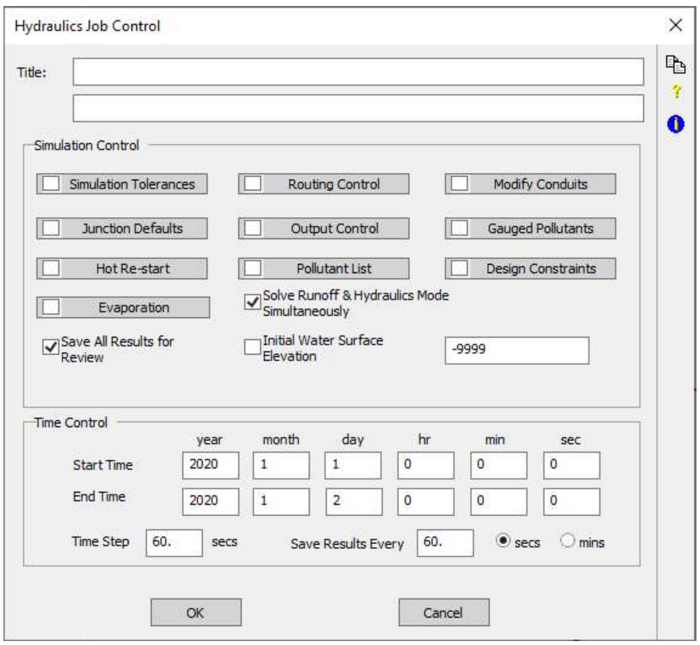
1. What is the ground elevation of Node 5/1? _____
2. What is the invert elevation of Node 3/2? _____
3. What is the length of Link 6? _____ ft
4. What is the peak user inflow at Node 5/1? _____ cfs

Job Control Settings & Running the Model

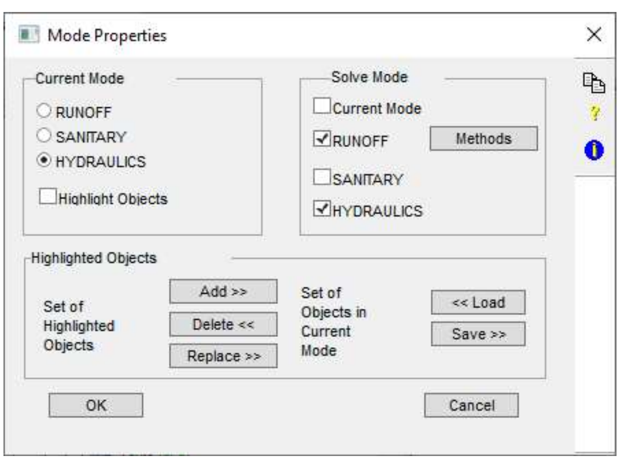
Settings for running the model are managed in the Job Control dialog. This part reviews some of the Job Control settings in the Hydraulic mode.

Changing configuration and application settings

1. On the **Configuration** menu, select **Job Control** and then **Hydraulics**. Tick the box for the option **Run Hydrology/Hydraulics Simultaneously**. This option eliminates the need for Interface Files. Set the **Start** and **Stop** times equal to the values used in Runoff. Click on **OK**.



2. On the Configuration menu, select **Mode Properties**. Under the Solve Mode, turn off **Current Mode** and check both **Runoff** and **Hydraulics**. Then click **OK**.



3. **Save as** your model with the name **stormwater9.xp**.

Solving the Model

1. Either click on the solve icon, press **F5** or select **Solve** in the **Analyze** menu. A default name for the model output file will be assigned by concatenating the model name and using and extension .out (stormwater9.out). A dialog indicating the status of the calculation will be displayed. When the simulation is completed, the network view will reappear.

2. The error.log file will open with a warning message telling us that one of our pipes is less than 32.81 feet or 10 meters. If the error.log file includes a warning, the model will run. If the error.log file includes a error, the model will not run until the error is fixed. The message in this error file is shown below:

```
WARNING: HDR: Links '6/1-5/2': Conduit Length (25.81) less then minimum
length for Analysis (32.81): Set Configuration Parameter MINLEN=


0 Error(s) and 1 Warning(s) were encountered

Data Export Completed Successfully
```

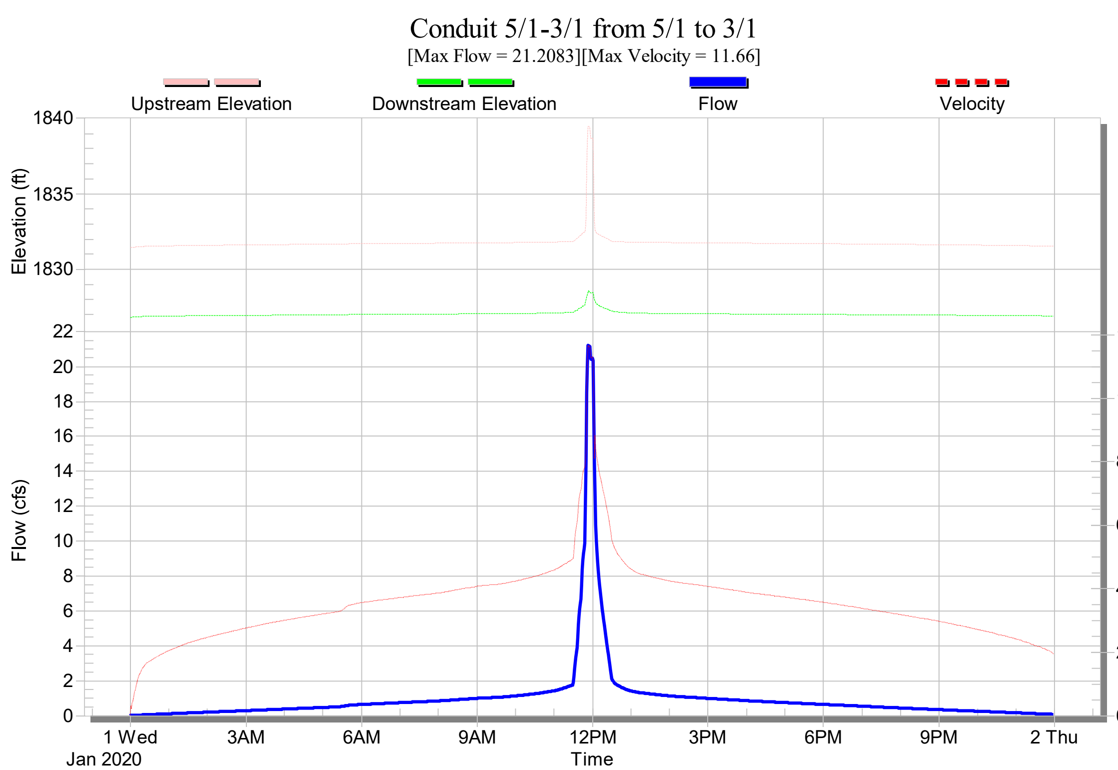
   We can proceed with the warning at this point. The warning means that all short pipes less than the default minimum will be simulated using the minimum length. As the warning indicates the default minimum can be altered using a configuration parameter.

Reviewing the Model

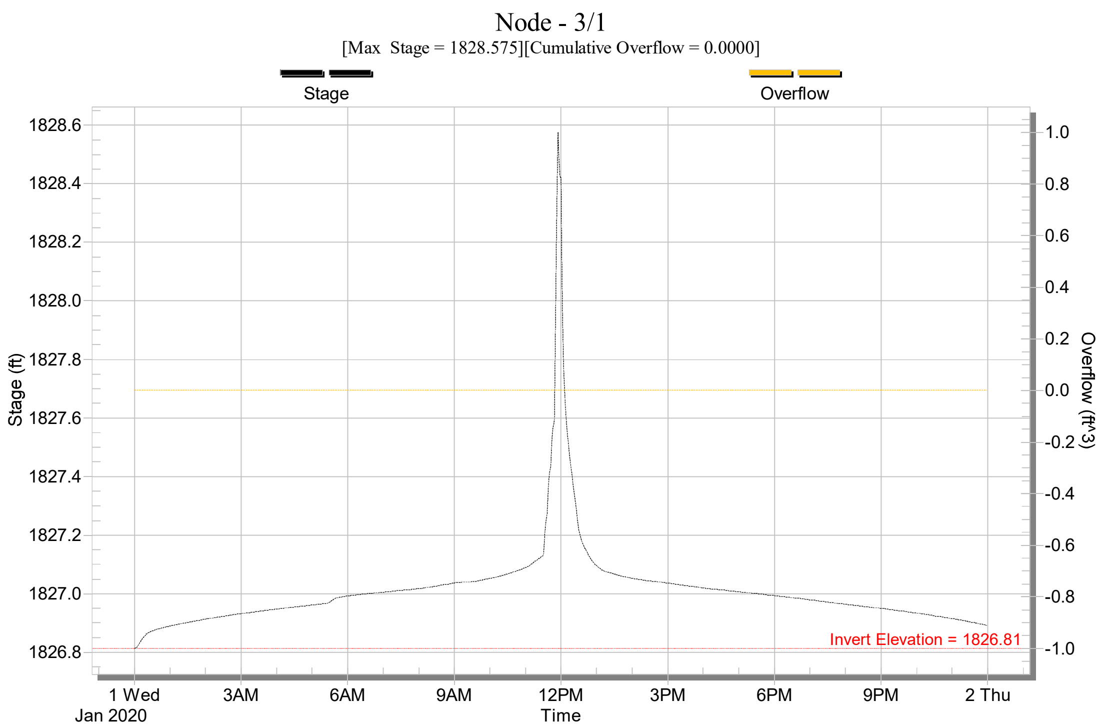
A variety of tools are available for examining model results. In Workshop Module 2, we explored the output from the Runoff Mode. In this example, we will explore output options in the Hydraulics Mode.

Examining graphical results

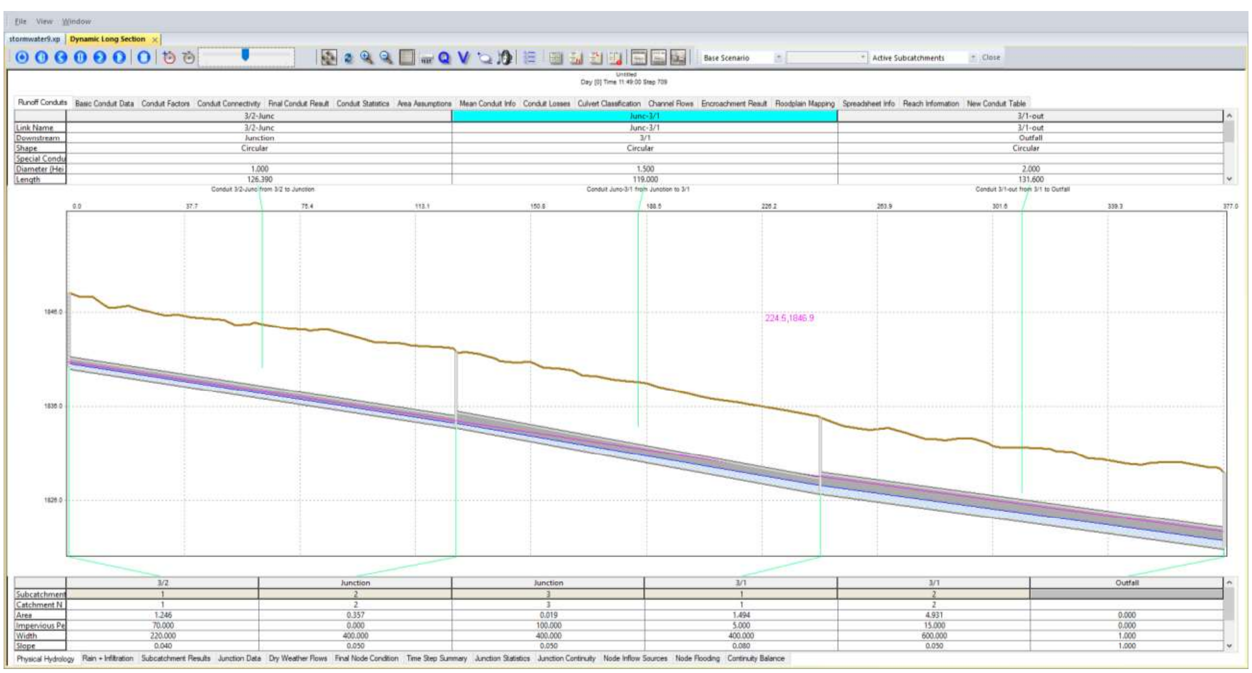
1. To get graphical results, you can right click on a node or link and select review results or you can left click to select the node or link and select the review results icon, or with the node or link selected, click on the review results line or you can use the **F7** button on the keyboard. Now select **5/1-3/1** and select **Review Results** using one of the methods mentioned above.
2. In the Conduit 5/1-3/1 output graphic, the Hydraulic Gradeline in the upstream and downstream part of the pipe is shown. Also, the flow and velocity are shown for January 1, 2020. Then **Close** the graphic.



3. Now select Node 3/1 and review results. In the Node 3/1 output graphic, the hydraulic gradeline (HGL) in the node is shown. This gradeline may not coincide with the pipe grade line, if the node is a drop manhole.



4. Another graphical output is the **Dynamic Long Section**. This allows the user to see a dynamic view of the flow and associated hydraulic gradeline. **Right-click** on node **3/2** and click on **Select Downstream Objects.** Then click on the **Dynamic Long Section** icon or in the results pull down menu. The magenta line shows the maximum hydraulic gradeline and the blue line shows the grade line at this time step.



Examining the Output File

1. To review the comprehensive text output, either select the Browse File icon, and open the stormwater9.out file or from the menu choose **Analyze->Show Output Logs->1D** Log. The latter will directly open the output file for the current storm and scenario into the programs text editor.

2. Open the results file **stormwater9.out**. A set of 30+ tables are generated in the output file. A list of the important tables is shown below. It is strongly recommended that users review all of these tables to determine the tables that are most useful for various analyses. Note: most of the information contained here can be viewed in xptables.

```
*============================================================*
|        HYDRAULICS TABLES IN THE OUTPUT FILE                |
|  These are the more important tables in the output file.   |
|  You can use your editor to find the table numbers,        |
|  for example: search for Table E20 to check continuity.    |
|  This output file can be imported into a Word Processor    |
|  and printed on US letter or A4 paper using portrait       |
|  mode, courier font, a size of 8 pt. and margins of 0.75   |
|                                                            |
| Table E1   - Basic Conduit Data                            |
| Table E2   - Conduit Factor Data                           |
| Table E3a  - Junction Data                                 |
| Table E3b  - Junction Data                                 |
| Table E4   - Conduit Connectivity Data                     |
| Table E4a  - Dry Weather Flow Data                         |
| Table E4b  - Real Time Control Data                        |
| Table E5   - Junction Time Step Limitation Summary         |
| Table E5a  - Conduit Explicit Condition Summary            |
| Table E6   - Final Model Condition                         |
| Table E7   - Iteration Summary                             |
| Table E8   - Junction Time Step Limitation Summary         |
| Table E9   - Junction  Summary  Statistics                 |
| Table E10  - Conduit   Summary  Statistics                 |
| Table E11  - Area assumptions used in the analysis         |
| Table E12  - Mean conduit information                      |
| Table E13  - Channel losses(H) and culvert info            |
| Table E13a - Culvert Analysis Classification               |
| Table E14  - Natural Channel Overbank Flow Information     |
| Table E14a - Natural Channel Encroachment Information      |
| Table E14b - Floodplain Mapping                            |
| Table E15  - Spreadsheet Info List                         |
| Table E15a - Spreadsheet Reach List                        |
| Table E16  - New Conduit Output Section                    |
| Table E17  - Pump Operation                                |
| Table E18  - Junction Continuity Error                     |
| Table E19  - Junction Inflow & Outflow Listing             |
| Table E20  - Junction Flooding and Volume List             |
| Table E21  - Continuity balance at simulation end          |
| Table E22  - Model Judgement Section                       |
*============================================================*
```

3. The first table that we will look at in this module is Table E22. This table provides an overall review of the model continuity. We expect that the overall error would be + or – 2% and that the worst nodal error would be + or – 5%. If the errors exceed those values, Table 18 should be reviewed to identify the nodes that have a significant continuity error.

```
###################################################
# Table E22. Numerical Model judgement section    #
###################################################

Overall error was (minimum of Table E18 & E21)            -0.0161 percent
Worst nodal error was in node 3/1                 with    -0.0046 percent
Of the total inflow this loss was                          0.0093 percent
Your overall continuity error was                 Excellent
                                                 Excellent Efficiency
Efficiency of the simulation                      2.45
Most Number of Non Convergences at one Node       1.
Total Number Non Convergences at all Nodes        1.
Total Number of Nodes with Non Convergences

*===============================================================*
|         XPSWMM/XPSTORM Simulation Date and Time Summary      |
*===============================================================*
| Starting Date... February   17, 2020  Time...  21:29:13.862  |
|   Ending Date... February   17, 2020  Time...  21:29:19.879  |
|  Elapsed Time...    0.07786 minutes or     4.67188 seconds  |
*===============================================================*
```

The second table is Table E18. This table is a continuation of Table E22. It shows the continuity error for each node in the model.

```
 *===========================================================================*
|    Table E18 - Junction Continuity Error.   Division by Volume added 11/96  |
|                                                                             |
|  Continuity Error = Net Flow + Beginning Volume - Ending Volume             |
|                     -------------------------------------------------       |
|                     Total Flow + (Beginning Volume + Ending Volume)/2       |
|                                                                             |
|  Net Flow   = Node Inflow - Node Outflow                                   |
|  Total Flow = absolute (Inflow + Outflow)                                  |
|  Intermediate column is a judgement on the node continuity error.          |
|                                                                             |
|  Excellent  < 1 percent    Great  1 to  2 percent  Good  2 to  5 percent   |
|  Fair  5 to 10  percent    Poor  10 to 25 percent  Bad  25 to 50 percent   |
|  Terrible > 50  percent                                                    |
 *===========================================================================*
```

Junction Name	<------Continuity Error -------> Volume	% of Node	% of Inflow	Remaining Volume	Beginning Volume	Net Flow Thru Node	Total Flow Thru Node	Failed to Converge
Junction	-1.4538	-0.0058	0.0012	3.0548	0.0000	1.6010	25230.5885	0
4/1	0.0836	0.0043	0.0001	0.5542	0.0000	0.6377	1921.3633	0
5/3	0.2834	0.0010	0.0002	2.7703	0.0000	3.0537	29031.6898	0
6/1	-0.4361	-0.0048	0.0004	0.6441	0.0000	0.2080	9161.5611	0
3/1	-11.3717	-0.0046	0.0093	10.0188	0.0000	-1.3529	244686.5697	0
5/2	2.6034	0.0048	0.0021	3.6136	0.0000	6.2171	53806.8005	0
3/2	-0.4752	-0.0023	0.0004	1.3317	0.0000	0.8565	20808.9359	0
Outfall	-3.8414	-0.0016	0.0031	3.5500	0.0000	-0.2913	244724.5203	1
5/4	-1.7041	-0.0114	0.0014	1.4411	0.0000	-0.2630	14987.5424	0
5/1	-3.4468	-0.0020	0.0028	7.4995	0.0000	169.4451	170172.7040	0

```
 The total continuity error was  -19.759     cubic feet
 The remaining total volume was   34.478     cubic feet
 Your mean node continuity error was   Excellent
 Your worst node continuity error was  Excellent
```

Table E8 shows the number of iterations required to solve each node and if the node did not converge. Non convergence means that the maximum iterations of 500 was reached. In that case the solution is solved one more time and that result is used as the answer for that time step.

```
 *=========================================================*
|     Table E8 - Junction Time Step Limitation Summary    |
*=========================================================*
|   Not Convr = Number of times this junction did not     |
|               converge during the simulation.           |
|   Avg Convr = Average junction iterations.              |
|   Conv err  = Mean convergence error.                   |
|   Omega Cng = Change of omega during iterations         |
|   Max Itern = Maximum number of iterations              |
*=========================================================*
```

Junction	Not Convr	Avg Convr	Total Itt	Omega Cng	Max Itern	Ittrn >10	Ittrn >25	Ittrn >40
Junction	0	1.54	13212	0	9	0	0	0
4/1	0	1.14	9784	0	5	0	0	0
5/3	0	1.53	13145	0	8	0	0	0
6/1	0	1.36	11704	0	287	3	2	2
3/1	0	2.29	19685	0	343	1	1	1
5/2	0	1.75	15055	0	8	0	0	0
3/2	0	1.44	12399	0	7	0	0	0
Outfall	1	9.72	83425	0	501	156	155	153
5/4	0	1.29	11097	0	7	0	0	0
5/1	0	2.38	20438	0	9	0	0	0

```
Total number of iterations for all junctions..    209944
Minimum number of possible iterations.........     85840
Efficiency of the simulation.................       2.45
                                                 Excellent Efficiency
```

4. The last table we will look at in this module is Table E7. Table E7 helps us decide on a reasonable largest time step for each model.

```
 *=============================================================*
|          Table E7 -  Iteration Summary                     |
*=============================================================*

Total number of time steps simulated...........       1440
Total number of passes in the simulation........     12830
Total number of time steps during simulation....      8584
Ratio of actual # of time steps / NTCYC.........     5.961
Average number of iterations per time step......     1.495
Average  time step size(seconds)................    10.065
Smallest time step size(seconds)................     1.000
Largest  time step size(seconds)................    60.000
Average minimum Conduit Courant time step (sec).    12.822
Average minimum implicit time step (sec)........    11.049
Average minimum junction time step (sec)........    11.049
Average Courant Factor Tf.......................    11.049
Number of times omega reduced...................         0
```

Questions

1. What is the total outflow at node Outfall? (Hint: Table E19) ________ ft^3

2. What is the maximum flow in link 3/1-out? (Hint: Table E15a) _______ cfs

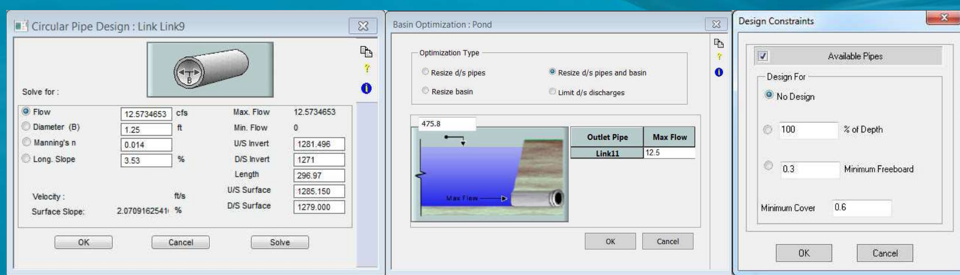
3. What node exceeded 10 iterations in this run? ________

4. What is the peak velocity for link 5/1-3/1? ______ fps

5. What is the maximum stage in node 5/1? ______

Advanced Hydraulic Modeling

Sizing/Design Tools and Dual Drainage

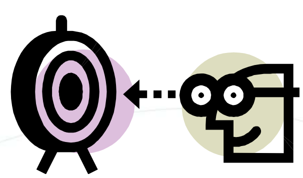


Innovyze®
Empowering water experts

1

Workshop No. 11 Objectives

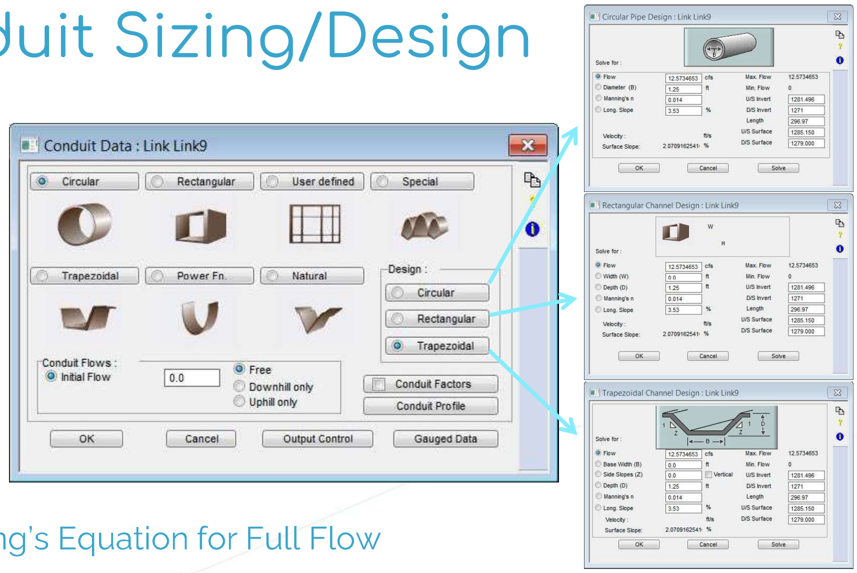
- Automated Conduit Sizing
- Modify Elevations
- Ponding Options
- Dual Drainage
- Drainage Inlets
- Pond Optimization



Innovyze®

2

Conduit Sizing/Design



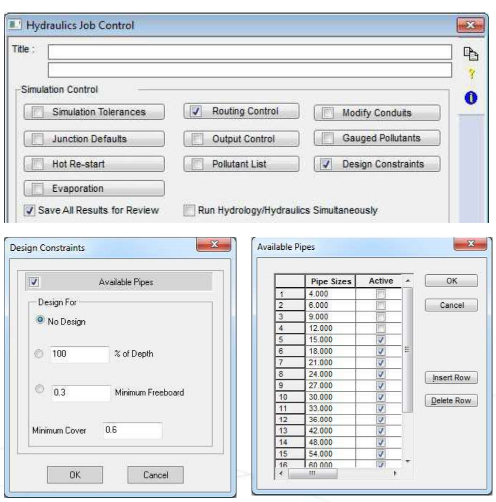
- Manning's Equation for Full Flow

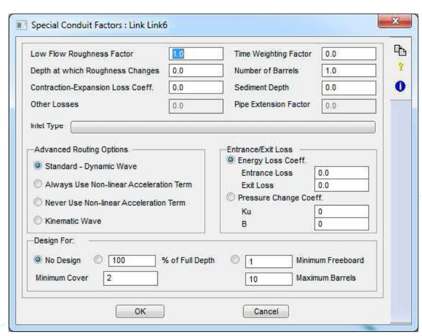
Innovyze®

3

Automated Conduit Design

- Job Control: All Conduits
- Conduit Factors : Individual Conduit(s)





Innovyze®

4

Automated Conduit Design

Overview of Algorithm

1. Identify undersized pipes
2. Increase pipe size
   i. Increase by one nominal size
   ii. If already at maximum, (no cover) then increase number of barrels by one
3. Restart Simulation
4. Repeat until all pipes meet design constraints

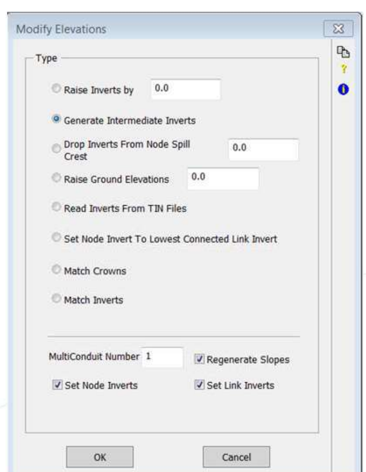
Note: Step 3 can be disabled with Configuration Parameter DESIGN_RESTART=OFF

Innovyze®

5

Modify Elevations

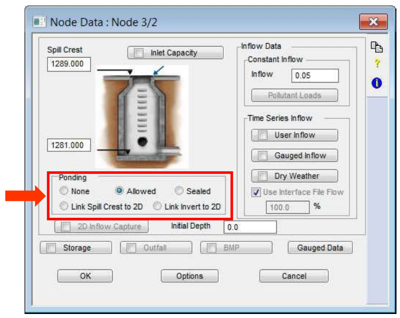
- For selected objects:
  - Raise or lower inverts
  - Generate intermediate inverts
  - Drop Inverts from ground
  - Raise or lower ground elevations
  - Read inverts from TIN files
  - Set node level to lowest link invert
  - Match Crowns
  - Match Inverts

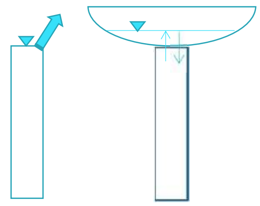


Innovyze®

6

Node Ponding Options





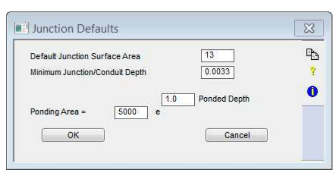
None (Lost) | Allowed (Stored) | Sealed (Pressure)

Innovyze

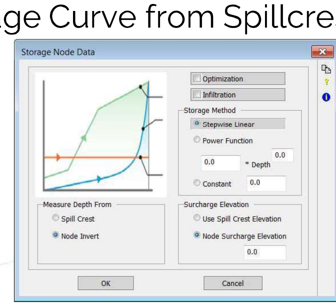
7

Ponding Allowed

1. Global Equation in HDR Job Control



2. Local Storage Curve from Spillcrest



Innovyze

8

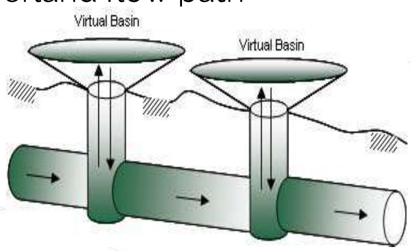
Traditional Surcharge Approach

Ponding Allowed in Flooded Node

- Overflowed water can be stored in virtual basin
- Returns back into the system when 1D system regains capacity.

Limitations:

- How to define the basin dimension
- No overland flow path



Innovyze®

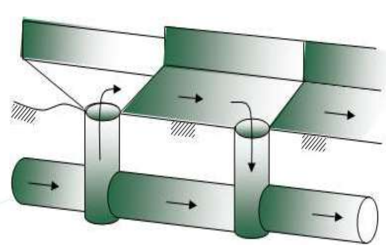
9

Dual Drainage

- Overflowed water can be routed through the predefined street channels (up to defined street cross-section)
- Capture to closed system can simulated with inlet capacity

Limitations:

- Extra effort is necessary
- No flood extents beyond the defined street geometry

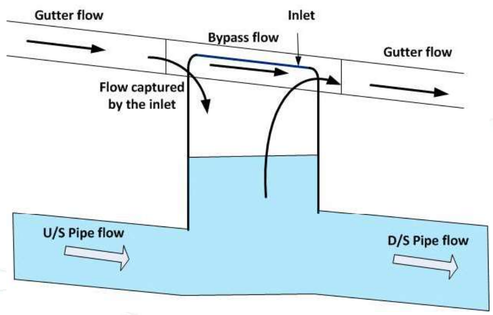


Innovyze®

10

Dual Drainage

- Gutter flow and pipe flow can be simulated together
- Multilink can be used for modelling
- Gutter flow modeled using natural or trapezoidal shape
- Inlet Capacity should be specified at the nodes

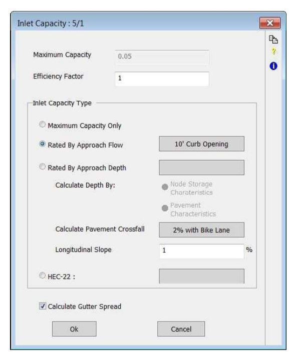


Innovyze®

11

Drainage Inlets

- Inlet Capture
  - Maximum capacity
  - Rates by approach flow or depth
  - HEC-12/22
- Gutter flow, spread, and velocity

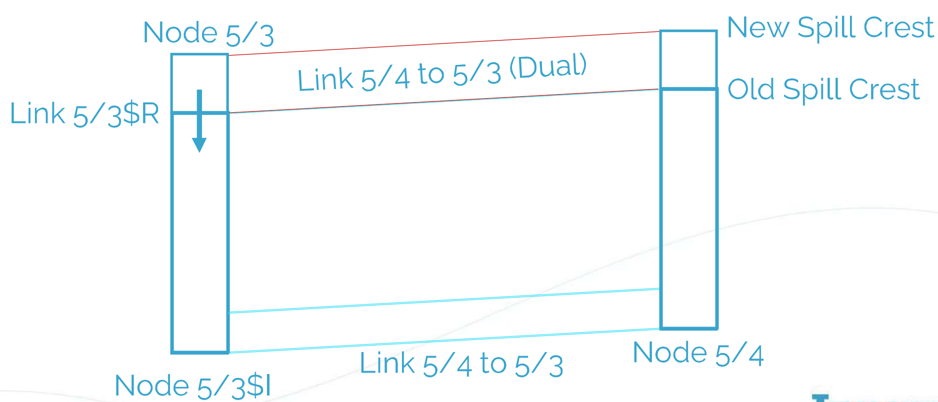


Innovyze®

12

Inlet Capacity – Internal Nodes and Links

- When using inlet capacity an internal (computational) node and link are created at each node

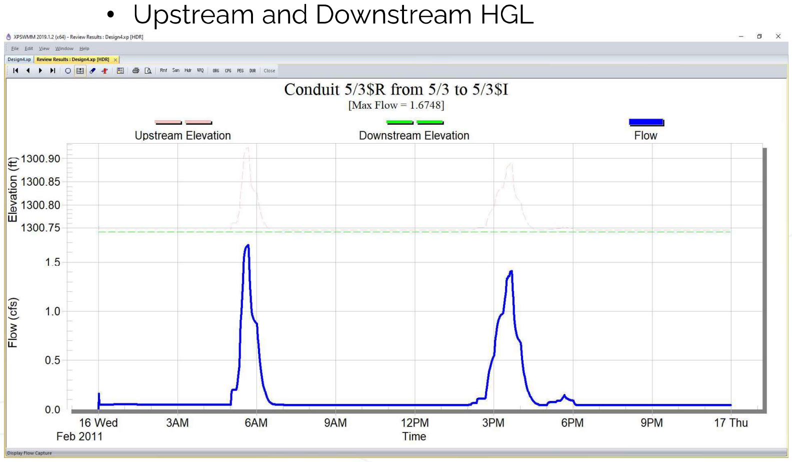


Innovyze®

13

Review Results: Inlet Capacity

- Troubleshoot Inlet Flows, Inlet time series include:
  - Inlet Flows (+/– flow direction)
  - Upstream and Downstream HGL

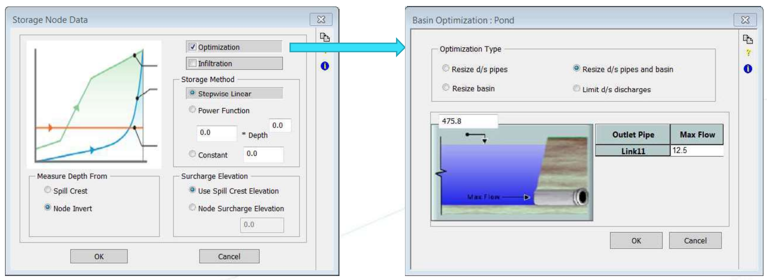


Innovyze®

14

Pond Optimization

1. Resize downstream pipes [Max HGL]
2. Resize basin [Max HGL]
3. Resize downstream pipes & basin [Max HGL | Max Flow]
4. Limit downstream discharges [Max Flow]



Innovyze®

15

Workshop Example Preview

- Conduit Sizing
- Storage
  - Pond Storage
  - Ponding Allowed
- Dual Drainage and Inlet Capacity
- Pond Design
  - Restrict downstream discharges
  - Restrict downstream discharges and maximum water surface

Innovyze®

16

Innovyze®

Empowering water experts

17

Advanced Stormwater Modeling Tools

The purpose of this training module is to teach users how to use xpswmm/xpstorm's design tool for sizing system and other tools for determining missing elevations, ponding options and pond optimization. In addition, the use of dual drainage and advanced conduit factors will be covered.

Objectives

Using model features, users will learn how to utilize xp's tools for sizing, optimize pond elevations or outflow, and simulate dual drainage. You will learn how to:

1. Determine optimal pipe diameters.
2. Use node storage modeling options.
3. Model parallel street and storm sewer (dual drainage).
4. Optimize pond storage and outflow.
5. Model pond structures.
6. Use conduit factors.

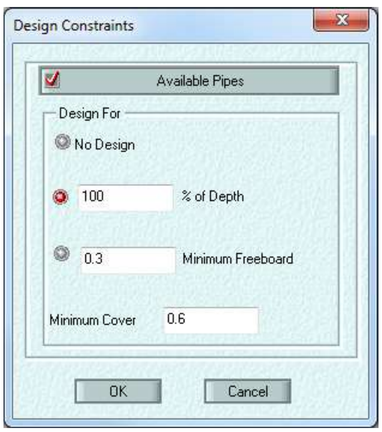
Data files:		
	YARRA_area.dwg	yarra_network_pt.shp
	yarra_link_arc.shp	Design_Dbase.xls
	Design.xp	Rational_method.xp
	Design1.xp	Design2.xp
	Design3.xp	Design4.xp

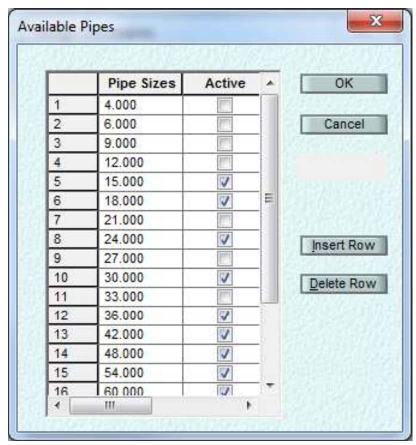
Calculating Pipe Diameters

Determining pipes sizes are often done using a trial and area approach. Using a hydrology method, the peak flow is determined and the pipe size is calculated using Manning's equation. In xp, a tool can be used to automate this design process. As can be shown, the 1-foot diameter pipes are not able to convey the flows created from the 10-year storm. We will use the pipe sizing design function to size all the pipes in our model.

1. Open the file **Design1.xp** and save the file as **My_Design2.xp**

2. Open the **HDR: Job Control** and **click** on the **Design Constraints**. Select **Design For 100**% of Depth. Click on the **Available Pipes** button and turn off the active check box for 21, 27 and 33-inch Pipe Sizes. Click the **OK** button three times.





3. **Solve** the model. The pipe length warning will be displayed. The model status will start over as it solves the model and increases the pipe sizes. Note: Using the configuration parameter, DESIGN_RESTART=OFF, will stop the restarting. However, the model should be rerun in such a case to ensure that the changing pipe diameters do not affect the final design.

4. Right click on Node **5/5** and pick **Select Downstream Objects**. Click on the **Dynamic Section View** and then click File pull down and select **Properties**. Using the icons in the toolbar turn on the **Text** and the **Diameter Info**. Note that some of the pipes increase as you move downstream and that the maximum HGL is below the crown. Click on the **Close** Button. Do the same thing for Node **3/2**.

5. The results of the Hydraulic design routine final results is shown in the output file, after Table 5a.

	<-------- Original	-------->		<--------- Designed	-------->	
Conduit Name	Height	Width	Barrels	Height	Width	Barrels
Link1	1.0000	1.0000	1.0000	1.2500	1.2500	1.0000
Link2	1.0000	1.0000	1.0000	1.2500	1.2500	1.0000
Link5	1.0000	1.0000	1.0000	1.2500	1.2500	1.0000
Link9	1.0000	1.0000	1.0000	1.2500	1.2500	1.0000
Culvert	1.0000	1.0000	1.0000	2.0000	2.0000	1.0000
Link11	1.0000	1.0000	1.0000	1.5000	1.5000	1.0000

6. Click on the **Job Control** button and click on the **Design Constraints** to turn off the pipe design feature.

7. **Save** the model.

Node Modeling Options and Dual Drainage

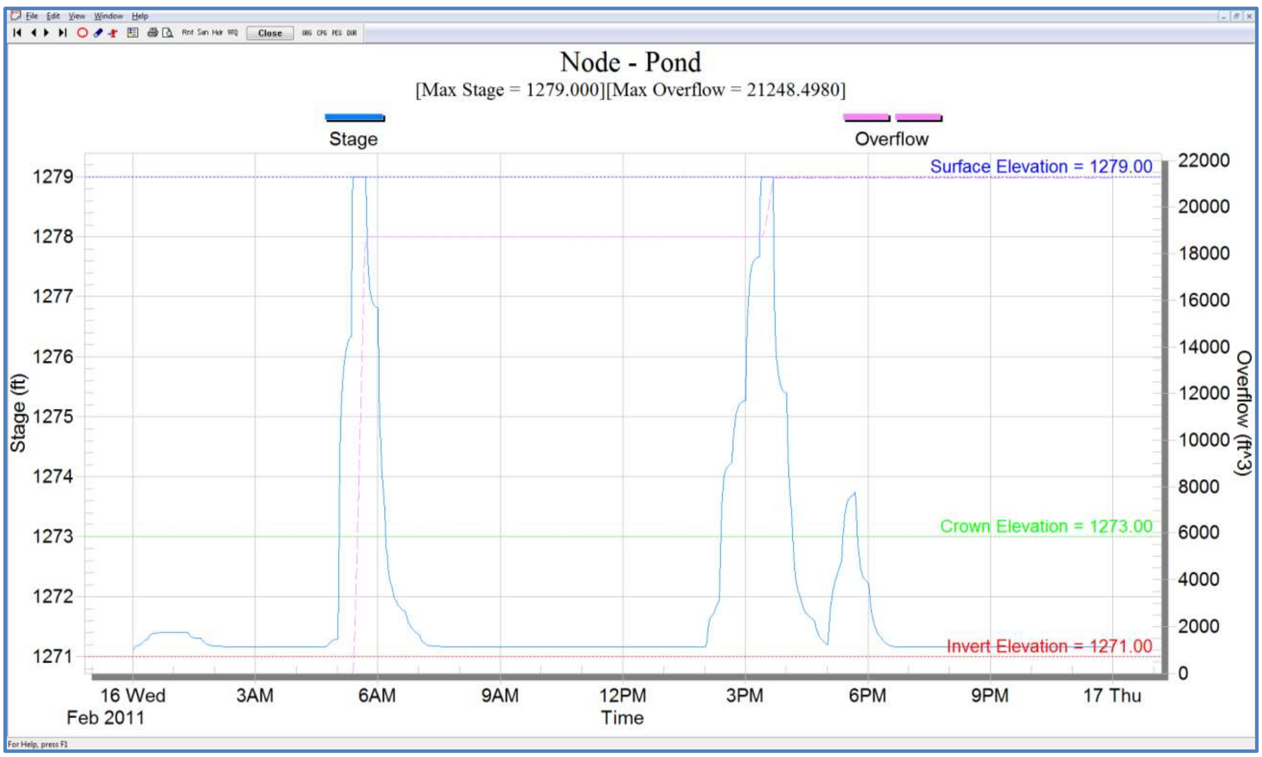
In xp, nodes can be modeled with inlet capacity and 5 different options for ponding alternatives. In this portion of the workshop, we will investigate these options and their impacts on model performance. In addition, the modeling of both sewers and streets will be shown.

Ponding Options

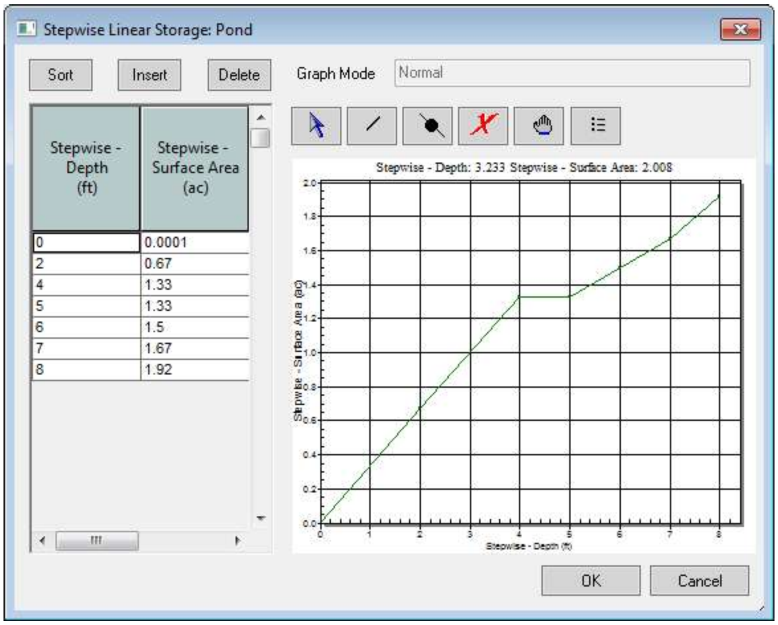
1. Open the file **Design2.xp** and save the file as **My_Design3.xp**. This version of the model is using the Runoff method, a rainfall and an infiltration global database.

2. **Solve** the model.

3. Review Results for the Node **Pond**. Notice that the HGL goes above the Crown Elevation which means that node was surcharged, and it goes to the ground elevation which means it has flooded and water is lost from the model. This is the default condition for nodes in xp. In the output file (Table E20) summarizes the duration of surcharge and flooding as well as the water lost.

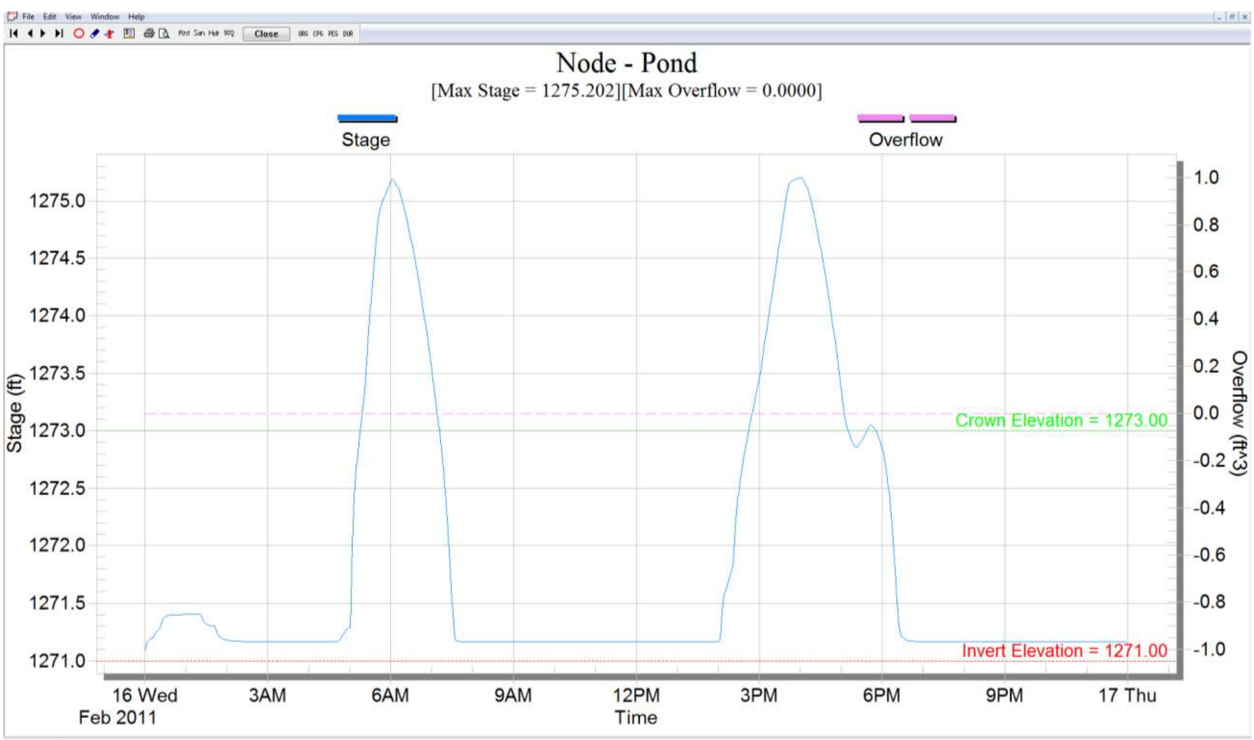


4. Convert the Pond node from a default manhole area to a Storage node. Double-click on Node **Pond**. Click on the **Storage** button. Three storage methods are available as shown. This pond will be defined using the stepwise linear method. Open the file **Design_Dbase.xls** to retrieve the stage storage information found in the worksheet called Pond. Click on the **Stepwise Linear** Button and click on the **Insert** Button to add one row. Then copy the data from worksheet Pond and paste it into Stepwise Linear Storage. Then click **OK** three times to return to the project window.



5. **Solve** the model.

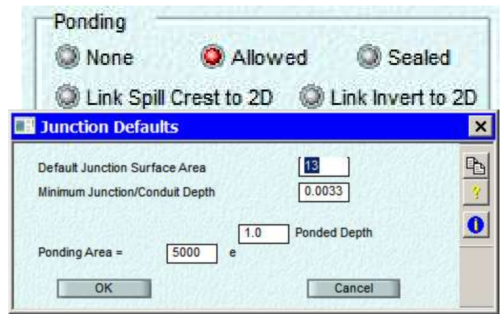
6. Review Results for the Node **Pond**. Notice that the HGL only slightly exceeds the Crown Elevation. By adding the storage data, the Pond is still surcharging but no water is lost from this node. However, the maximum volume in the node is now 26,700 cubic feet. Using the Dynamic Long Section see that other nodes are flooding.



7. Right Click on Node **5/5** and choose **Select Downstream Objects**. Click on the **Dynamic Long Section** icon and click on the **Play** button. Notice that all nodes upstream from **5/1** have flooding occurring during the run. Repeat the above starting at node **3/2**. Notice that node **Junction** is in a sag location.

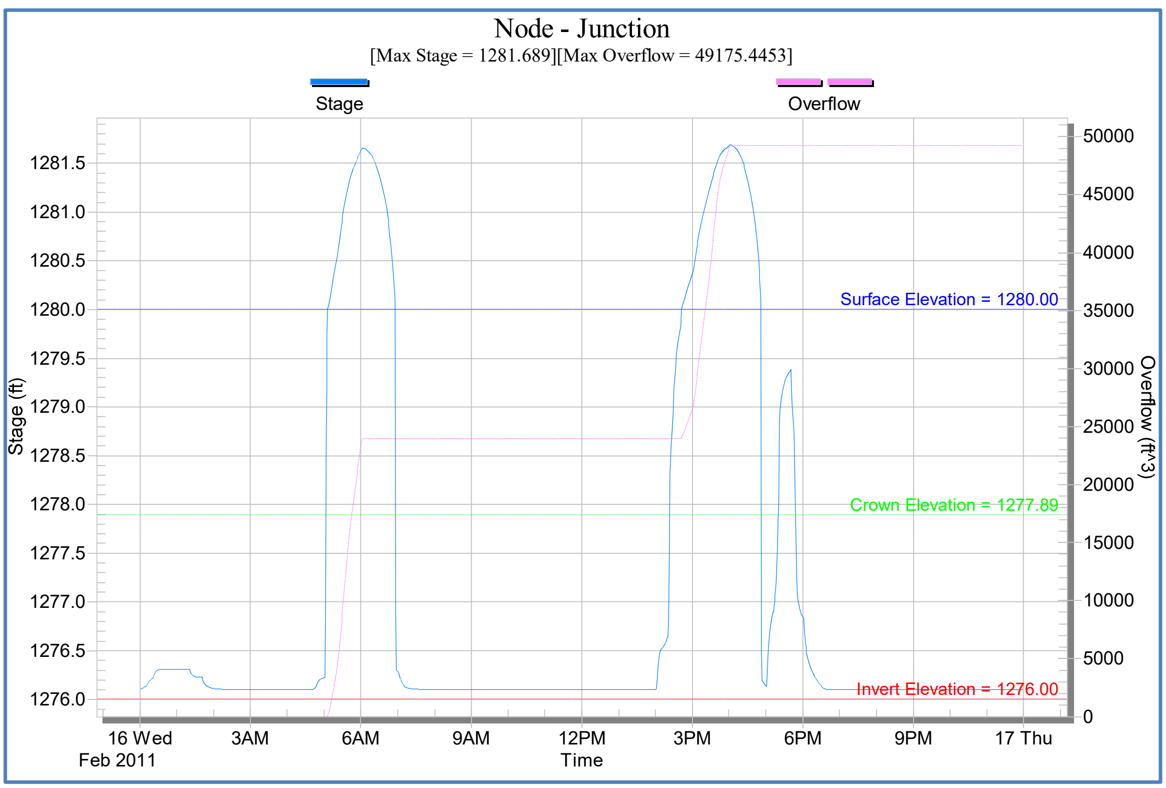
8. Review Results for the Node **Junction**. Notice that the HGL goes up to the Surface Elevation and then goes flat. Water was lost as described earlier.

9. Double-click on Node **Junction**. Ponding options are located in the middle of the node dialog. Since this node is in a sag location, click on the **Allowed** button. The ponding area will be determined from the Junction Defaults in the HDR Job Control as shown to the right.

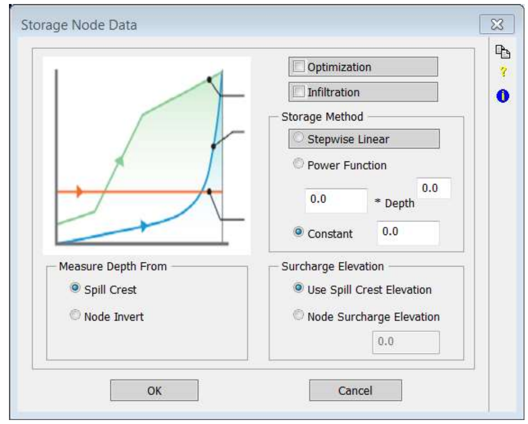


10. **Solve** the model.

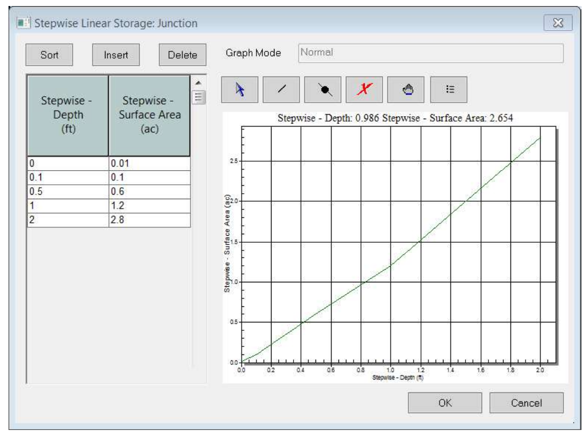
11. Review Results for the Node **Junction**. Notice that the HGL now exceeds the Surface Elevation by more than 1.5 feet. By allowing ponding, **Junction** is still flooding, but no water is lost from this node and the water above the node returns to this same node during the recession limb.



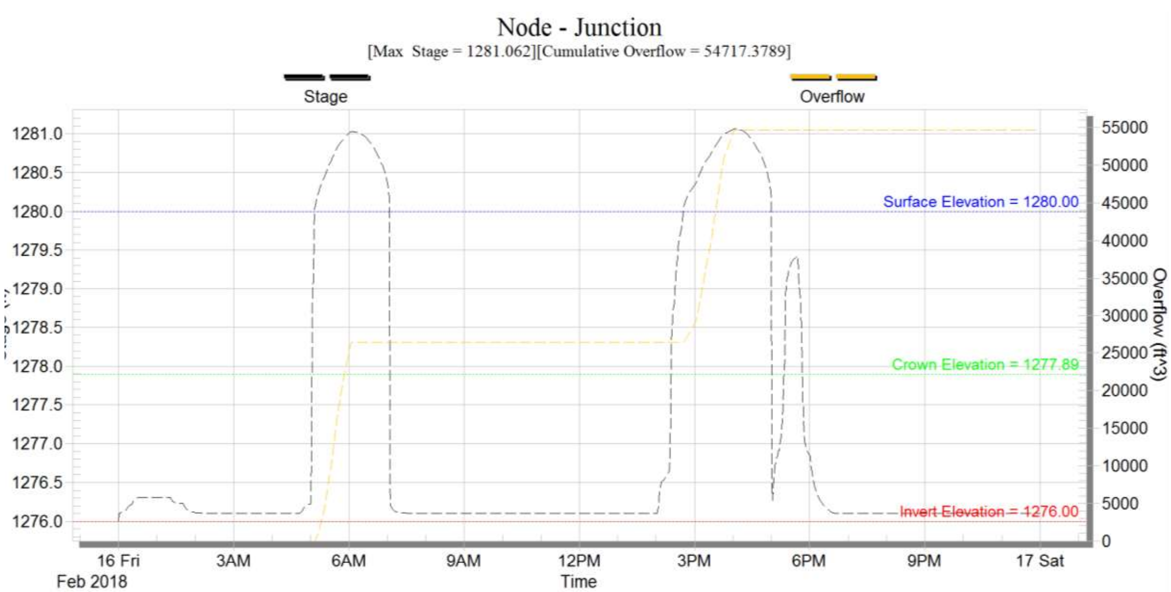
12. The next step is to define specific storage data for this node. Double-click on **Junction**. Click on the **Storage** button. Three storage methods are available as shown to the right. This pond will be defined using the stepwise linear method. Under Measure Depth From: Click on the **Spill Crest** Button.



13. Open the source data file **Design_Dbase.xls** which is where the node ponding information is located. Click on the **Stepwise Linear** Button and click on the **Insert** Button to add one row. Then copy the data from worksheet **Junction** and paste it into Stepwise Linear Storage. Then click **OK** three times to return to the project window.



14. **Solve** the model and compare the results at Junction. Note that elevation is now lower as compared to the previous simulation. The storage has a big influence on the HGL as demonstrated in the last few models.



15. **Save** the model database.

Inlet Capacity and Dual Drainage

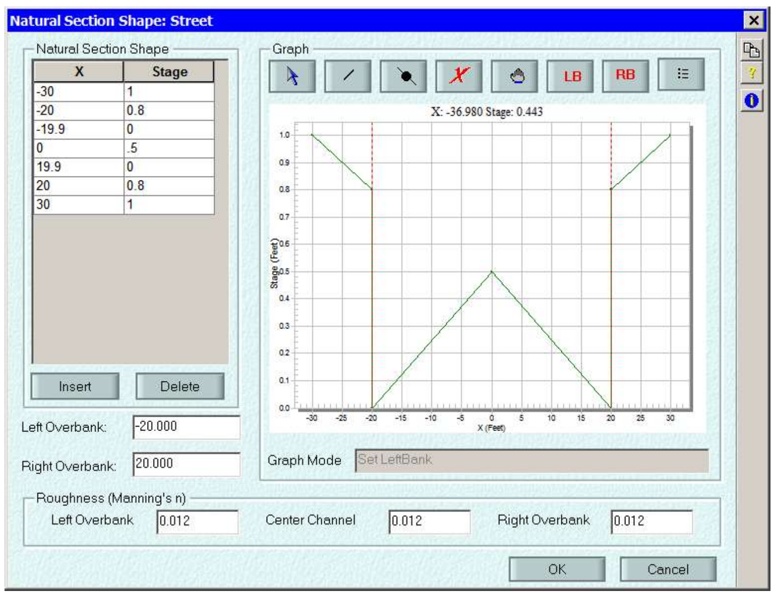
1. Open the file **Design3.xp** and save the file as **My_Design4.xp**.

2. In order to represent flows in the street and the pipe Link8 needs to been changed to a multilink and display as a dashed line. Right-click on **Link8** and change the single link to a multi-link.

3. Double-click on **Link8** which will take you to the Multiple Link dialog box. Enter Conduit Names for Conduits 1 and 2 as **SewerL8** and **StreetL8**.

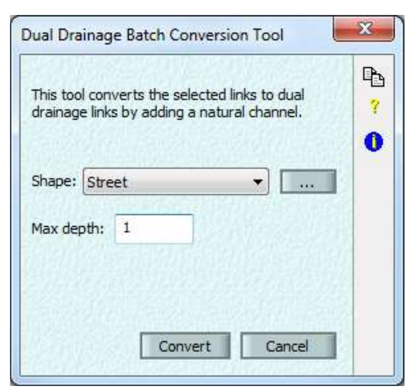
4. Double-click on the checkbox for **SewerL8** (Conduit 1). Click on **Conduit Profile**. This is the original closed conduit of the multilink. Then click **OK**.

5. Double-click on the checkbox for **StreetL8 (**Conduit 2). Click on the **Natural** Button and then click on the **Edit** button to view the Natural Section Shapes global database used to represent the overland flow.



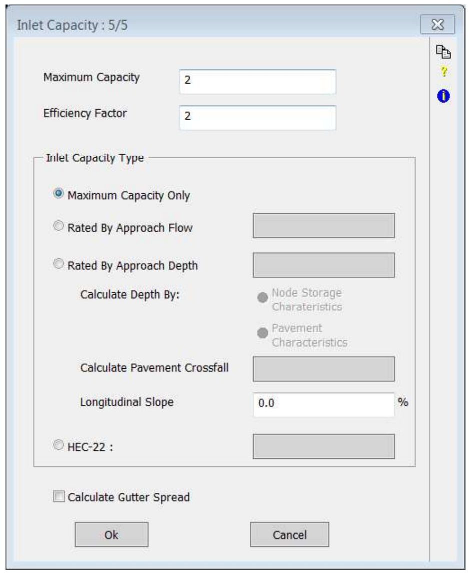
6. Click **OK**, then **Select**, then click **OK** three times to return to the project window.

7. Select **Link7** and **Link6** using the CTRL key. Then choose the **Tools->Dual Drainage Batch Converter…** menu item.

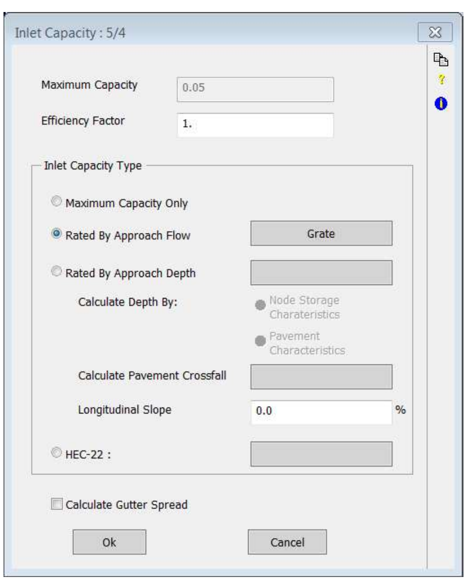


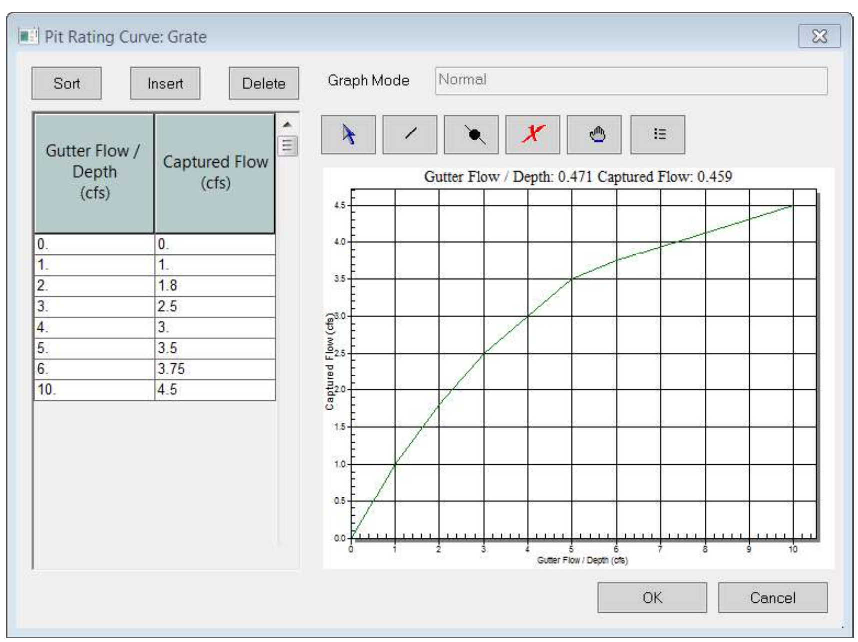
8. Select the **Street** shape already in the Global Database. Add Max depth of **1** so that the new open conduit has top of bank equal to the new adjusted spillcrest elevations. Select the **Convert** button to perform the conversion.

9. The next step is to use inlet capacity to control the amount of water in the sewer and on the street. Double click on Node **5/5** and click on the **Inlet Capacity** button. Type in a Maximum Capacity of **2** cfs and an Efficiency Factor of **2** (two inlets of 2cfs each).



10. Then double click on Node **5/4** and click on the **Inlet Capacity** button. Under **Inlet Capacity Type:** Click on the global database box next to **Rated by Approach Flow.** Select **Grate** and then click on **Edit** button to look at the rating curve. Click **OK**, then **Select** and then **OK** twice to return to the project window.



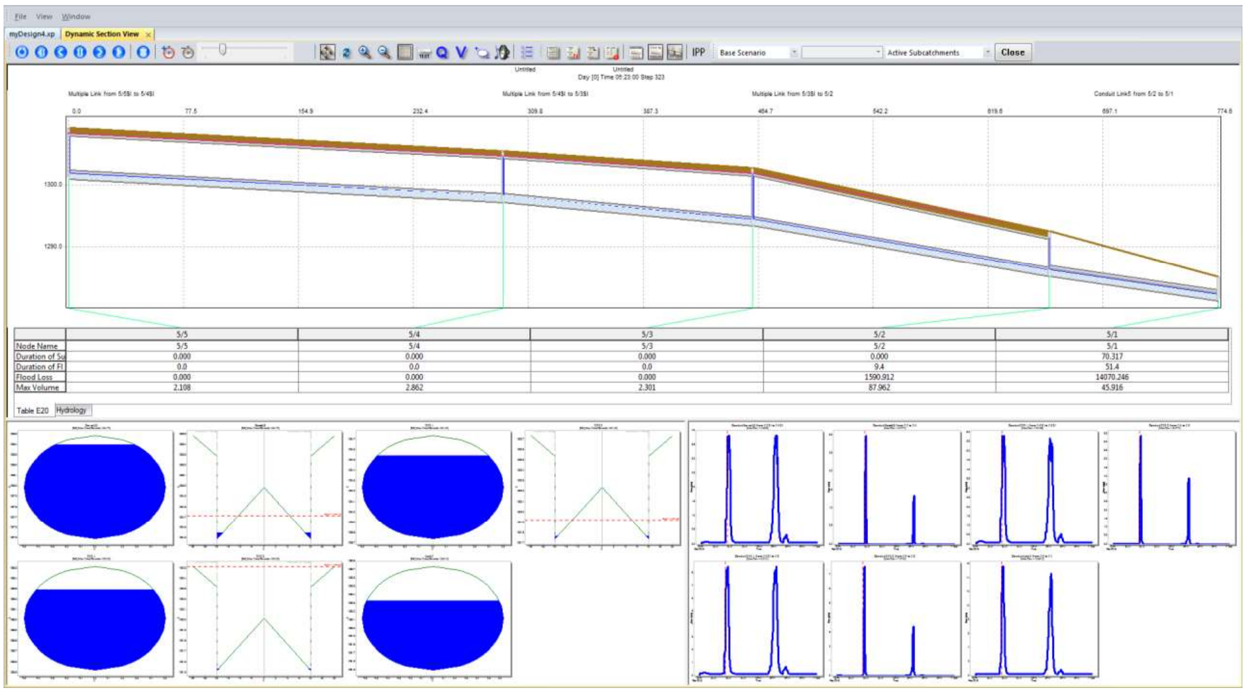


11. Repeat for Node **5/3** and select the **Grate** curve for inlet capacity.

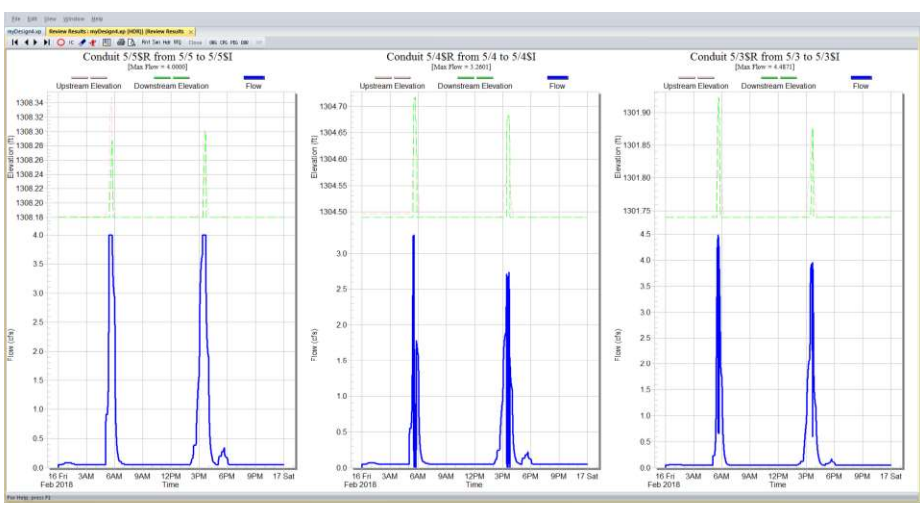
12. Save the model by using the Save button or by using the File Pull-Down or **Ctrl-S**.

13. **Solve** the model.

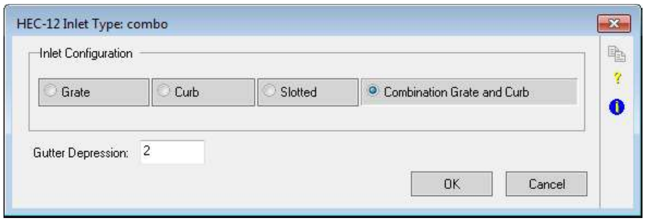
14. Left Click on Node **5/5** and then press Shift and Click on Node **5/1**. Click on the **Dynamic Section Views** icon and click on the **Play** button. Notice that how the model puts water in both the sewer and the street. Use review results to look at different nodes and links associated with dual drainage. **Close** the view.

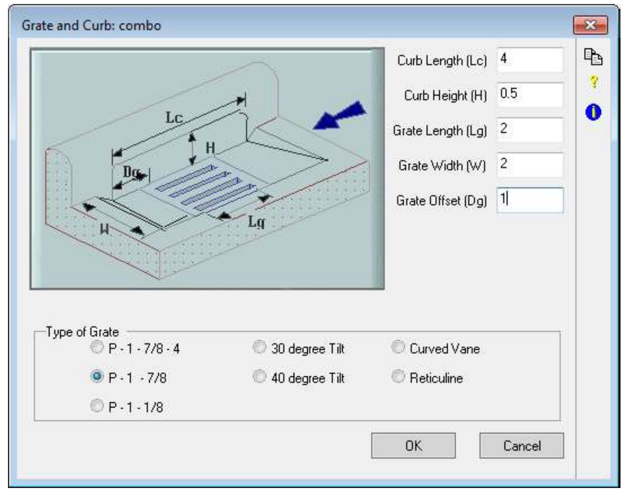


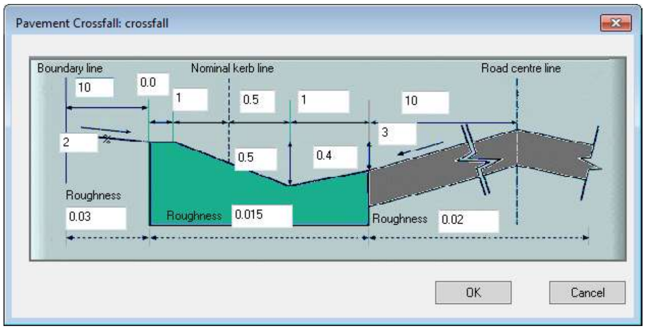
15. A new feature now allows the user to visualize the hydrograph of Inlet Capacity. Select the node **5/5**, **5/4** and **5/3** using the **CTRL** key and then **Review Results**. In the Review Results choose the **IC** icon on the toolbar.



16. Time permitting? Consider trying different types of inlet capacity including HEC-22 option for node 5/3 using the following choices with 1% longitudinal slopes.







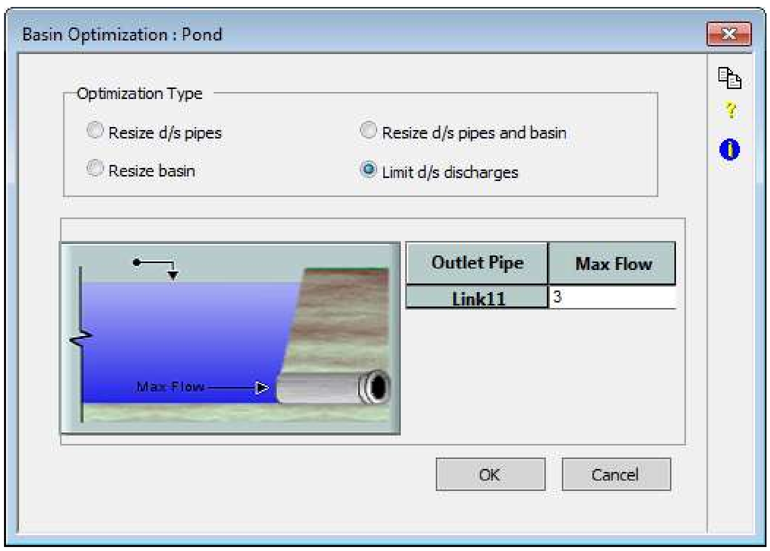
Pond Structures and Optimization

As in pipe sizing, determining the correct size for a pond is based either on the maximum elevation of a pond or the maximum discharge from a pond. The area available for the pond and inflow/outflow regulations will also impact the pond design. The developer wants to limit the size of the pond and regulators often require a predevelopment release rate. These goals usually require several iterations to achieve a successful design.

1. Open the file **Design4.xp** and save the file as **My_Design5.xp**.

2. The node Pond is a wet pond and should have an impounded dead pool of water. Double-click on the node **Pond**. Type **5** as the Initial Depth. Double-click on the **Storage** button. Click **OK** two times to return to the project window.

3. We will also need to change Link11 because Pond has an initial depth. Double-click on **Link11** and change the Upstream Pipe Invert to **1276**. *Hint*: **Right-click->Conduit Profile**. Don't forget to resolve for the conduit slope to avoid a warning message.



4. Double-click on the node **Pond**. Double-click on the **Storage** button. Click on the **Optimization** button to start the pond design process. The first step will be limiting the downstream discharge to a release rate of 3 cfs. Click on **Limit d/s discharges** and enter a Maximum Flow of **3** for Link11. Click **OK** three times to return to the project window.

5. **Solve** the model.

6. **Review Results** of **Pond** and see that the maximum elevation of Pond is about **1278.128**. The rim elevation is 1279.00, which leaves a freeboard of less than 1 ft.

7. **Review Results** of Link11 to verify the flow has been reduced below 3 cfs. This is accomplished by iteration and a reduction of the conduit size. Double-click on **Link11** to see the new conduit diameter to accomplish this flow throttling.

8. **Browse** the model output file to see the optimization results in Table E23:

```
#########################################################
# Table E23. New Basin Design Information               #
#            Maximum Hydraulic Grade Line,              #
#            Out Conduit Sizes and Maximum Flow         #
#########################################################

A) Resize d/s Pipes based on given HGL
B) Resize Basin based on given HGL
C) Resize d/s Pipes and Basin based on HGL and max discharge
D) Resize d/s pipes based on given max discharge

   Basin Name    Type    Max.HGL   Conduit         Depth   Width   Barrels   Max.Flow
                          (ft)                      (ft)    (ft)             (ft^3/s)
--------------- ------  --------- --------------- ------- ------- --------- ---------
           Pond    D)   1278.1283          Link11  0.4167  0.4167    1.0000    2.0426
```

9. The next step is to resize the downstream pipes and the basin to accomplish this maximum release rate of 3 cfs and ensure enough freeboard. Double-click on **Pond**, **Storage** and then **Optimization**. Change the selection to **Resize d/s pipes and basin** and enter a Maximum Water Level of **1278** in that field. Click **OK** three times to return to the project window. **Solve** the model.

10. Review results of **Pond** and **Link11** to see that both the maximum elevation of Pond and maximum discharge criteria are satisfied.

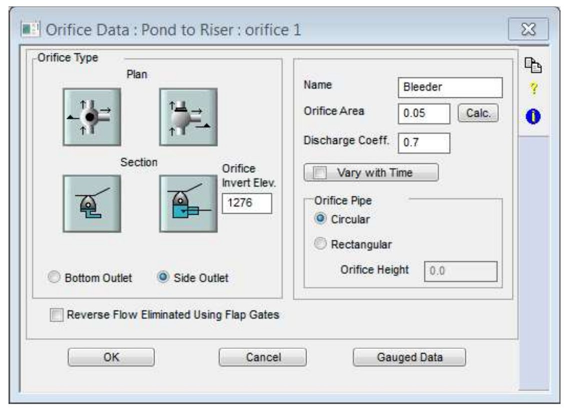
11. **Browse** the model output file and search for the new optimization for the Pond in Table E23.

```
#########################################################
# Table E23. New Basin Design Information               #
#            Maximum Hydraulic Grade Line,              #
#            Out Conduit Sizes and Maximum Flow         #
#########################################################

A) Resize d/s Pipes based on given HGL
B) Resize Basin based on given HGL
C) Resize d/s Pipes and Basin based on HGL and max discharge
D) Resize d/s pipes based on given max discharge

   Basin Name    Type    Max.HGL   Conduit         Depth   Width   Barrels   Max.Flow
                          (ft)                      (ft)    (ft)             (ft^3/s)
--------------- ------  --------- --------------- ------- ------- --------- ---------
           Pond    C)   1277.9639          Link11  0.4167  0.0000    1.0000    2.0152

Number of Restarts:    9

#########################################################
# Table E23A. New Basin Design Information              #
#             Basin Storage Parameters                  #
#########################################################

   Basin Name     Storage Type    Parameter   Depth      Area
                                               (ft)      (ft^2)
--------------- ---------------- ---------- ---------- ----------
           Pond  stepwise linear                0.0000     6.2432
                                                2.0000 43078.3353
                                                4.0000 85580.3718
                                                5.0000 85580.3718
                                                6.0000 96530.0490
                                                7.0000 107479.726
                                                8.0000 123568.068
```

12. Right-click **Link11** and choose **Properties**. Change the link name to **Structure**. Then click on the **Browse** button. Select **Structure.jpg**. Click on **Open**. Then click **OK** to return to the project window. Right-click to open the pop-up dialog and select **View Image** to see the image we just attached to this link.

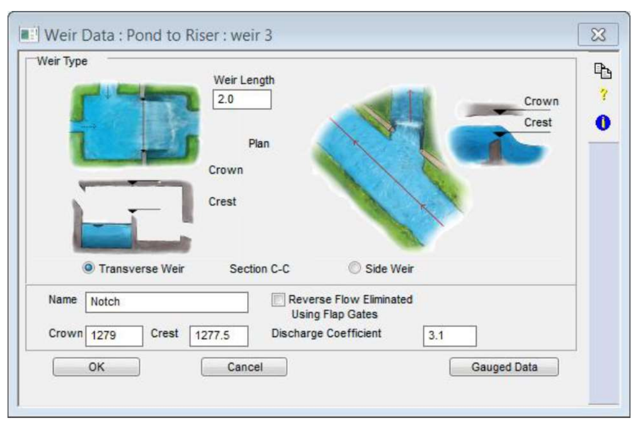
13. Right Click on **Structure** and change to a **Multi-Link**. Double-click on **Structure**. Turn off the **check box** for Conduit 1 and delete the conduit name.

14. The next step is to enter the data for two orifices. The orifice dialog box is shown to the right. Click on **Orifice 1**. Enter **Bleeder** as the Name, **0.05** as the Orifice Area, **0.7** as the Discharge Coeff. and **1276.0** as the Orifice Invert Elev. Make sure that the **Side Outlet** and the **Circular** Orifice Pipe are both selected. Then click **OK**.



15. The second orifice is entered in **Orifice 2**. Enter **Rectangle** as the Name, **0.5** as the Orifice Area, **0.7** as the Discharge Coeff. and **1276.5** as the Orifice Invert Elev. Make sure that the **Side Outlet** and the **Rectangular** Orifice Pipe are both selected. Enter **0.5** as the Orifice Height. Then click **OK**.

16. The next step is to enter the data for the weir. Click on **Weir 3**. Enter **Notch** as the Name, **2.0** as the Weir Length, **1279.0** as the Crown, **1277.5** as the Crest and **3.1** as the Discharge Coefficient. Make sure that the Transverse Weir is selected. Then click **OK**.



17. The second weir is entered in Weir 4. Enter **Rim** as the Name, **10.0** as the Weir Length, **1279.0** as the Crown, **1278.0** as the Crest and **3.1** as the Discharge Coefficient. Make sure that the Transverse Weir is selected. Then click **OK.**

18. Double-click on **Pond**, **Storage** and then single click on the **Optimization** Button to turn off the Optimization routine. Click **OK** twice times to return to the project window.

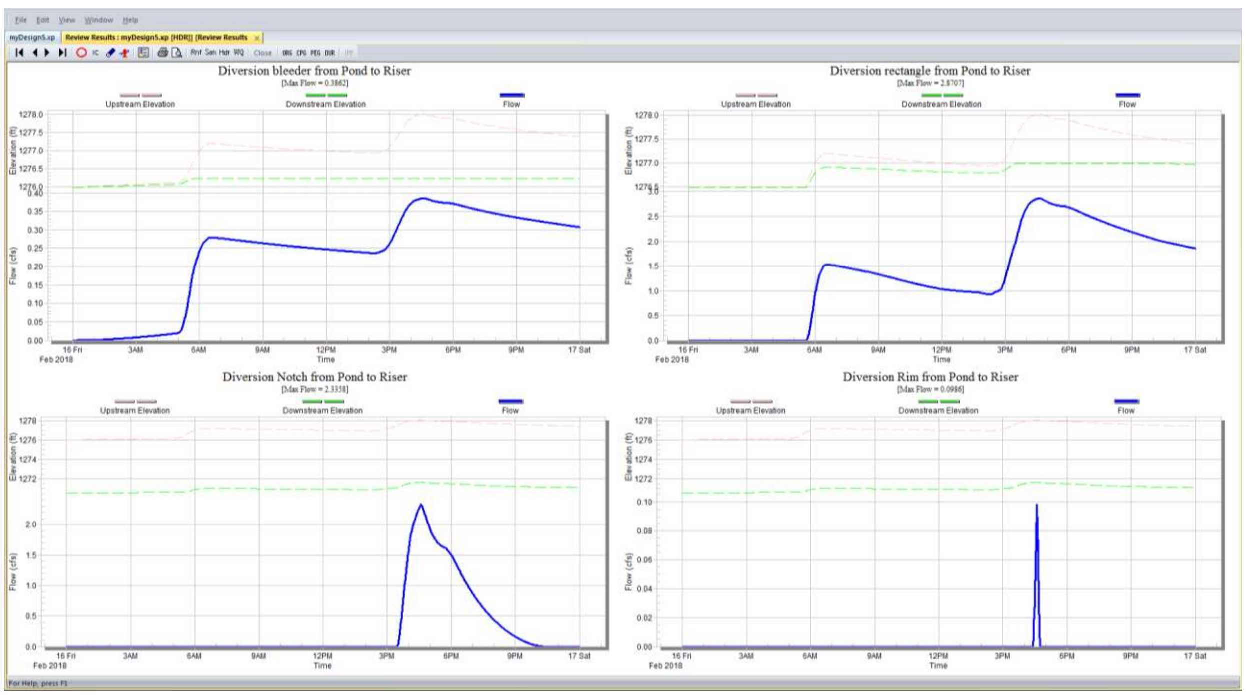
19. **Solve** the model and review results of **Pond** and see that the maximum elevation of Pond is now a bit more than 1278. The rim elevation is 1279.00, which leaves a freeboard of about 1 ft. Review results of **Culvert** and see that the maximum flow is

about 5.7 cfs. We would need to refine the orifice and weir parameters until we obtained the correct pond elevation and release rate.

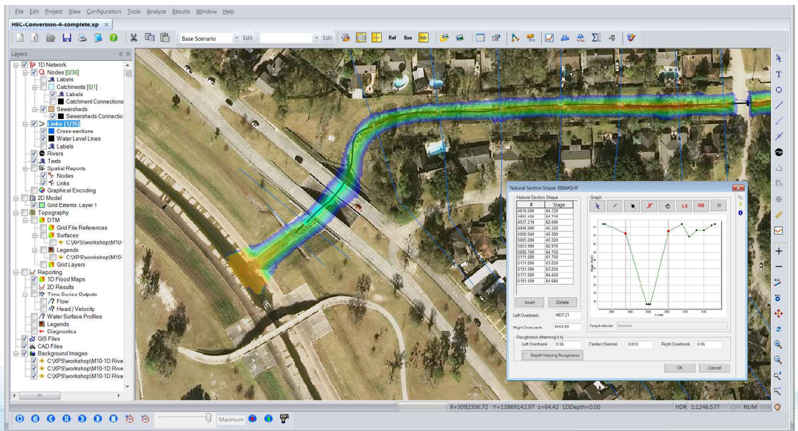
20. The final step is to use Conduit Factors to define the Inlet Type for the discharge through Culvert and apply minor losses. The Inlet Type can significantly impact the head water elevation of Pond, if the Pond becomes under Inlet Control. Double-click on **Culvert** and then click on **Conduit Factors**. Click on **Inlet Type** and choose **Square Edge with Headwall (Circ, Conc)**. Enter and Entrance loss of **0.5** and an Exit Loss of **1.0**. Click **OK** three times to return to the project window.

21. **Solve** the model and review results of **Pond** and see that the maximum elevation of Pond is now **1278.022**. Look at Table 13a and see that the Culvert did go into Inlet Control for 510 minutes, but didn't impact the Pond head water elevation.

22. **Review Results** on **Structure** to see the flow rates for the orifices and weirs.



1D River Modeling

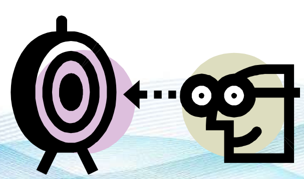


Page 1

1

Workshop No. 10 Objectives

- Review SWMM 1D River Hydraulics Theory
- Import Cross-sections from HEC RAS
- Build a 1D River model
- River and Bridge Links
- 1D Flood Maps

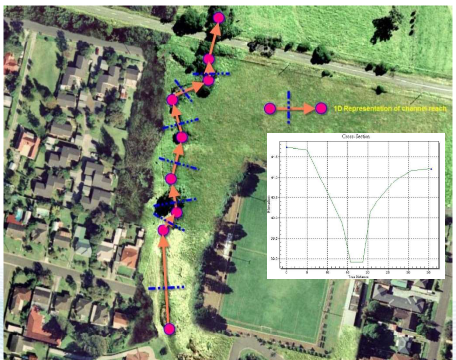


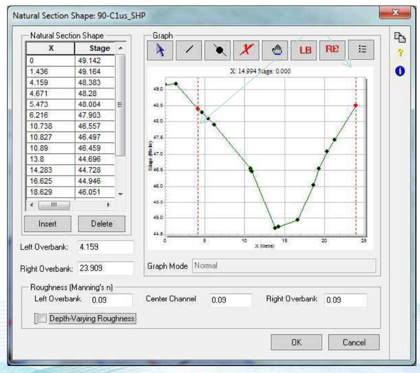
Page 2

2

1D Representation of Channels

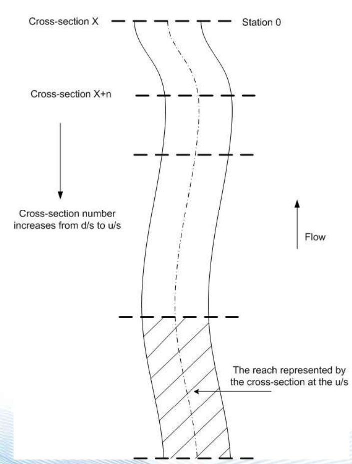
- Each river reach (link) represented by one cross-section
- Prismatic (same geometry u/s to d/s)

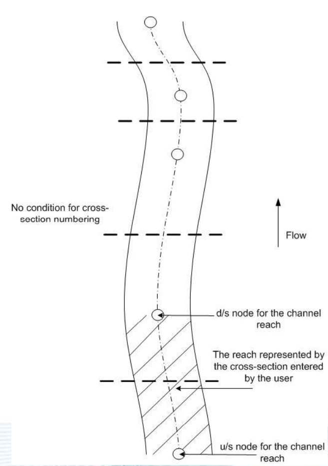




Innovyze®

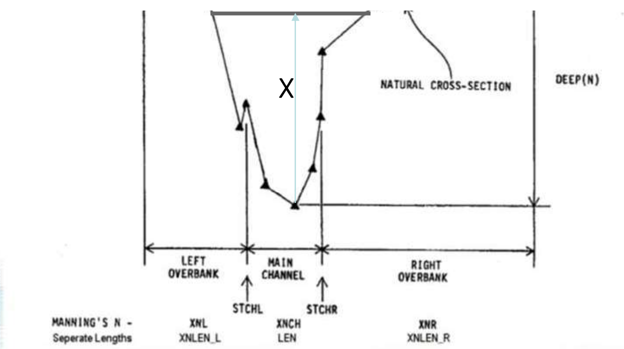
HEC RAS vs SWMM representation





Innovyze®

Natural Channel Definition



A vertical wall is created when max depth = 0 and:

1) when one side is lower
2) when top of channel is less than node ground elevation
3) No vertical wall when max depth = X and X is less than full channel depth

Note: Vertical walls are not created when max depth = X or configuration parameter VERT_WALLS=OFF

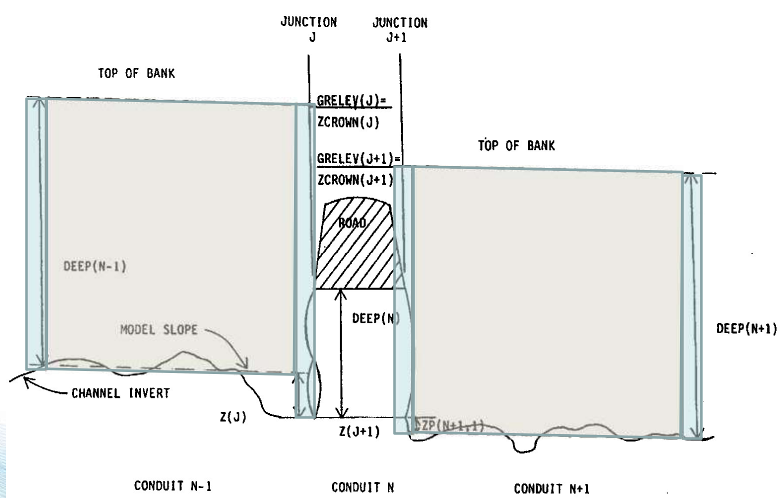
1) Sometimes used to model flow under a bridge

Innovyze®

Page 5

5

Open Channel Elevations



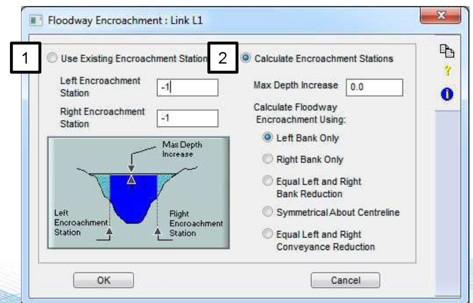
Innovyze®

Page 6

6

Floodway Encroachment

1. Option1: Water level increase can be calculated by entering the encroachment stations
2. Option2: Encroachment stations can be calculated by entering the maximum depth increase and method



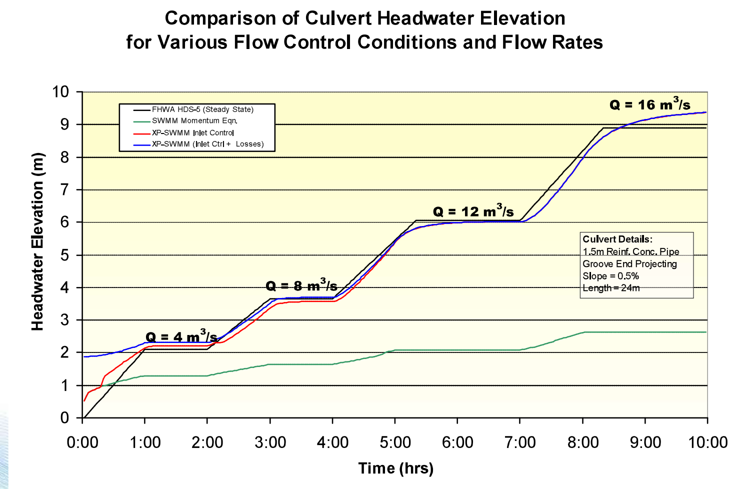
Innovyze®

FHWA Inlet Control Equations

- The head under inlet control may be significantly greater than that estimated assuming outlet control
- xp uses Inlet Control equations from the FHWA's “Hydraulic Design of Highway Culverts”

Innovyze®

Inlet Control



Innovyze®

Page 9

9

Most Important Hydraulic Results

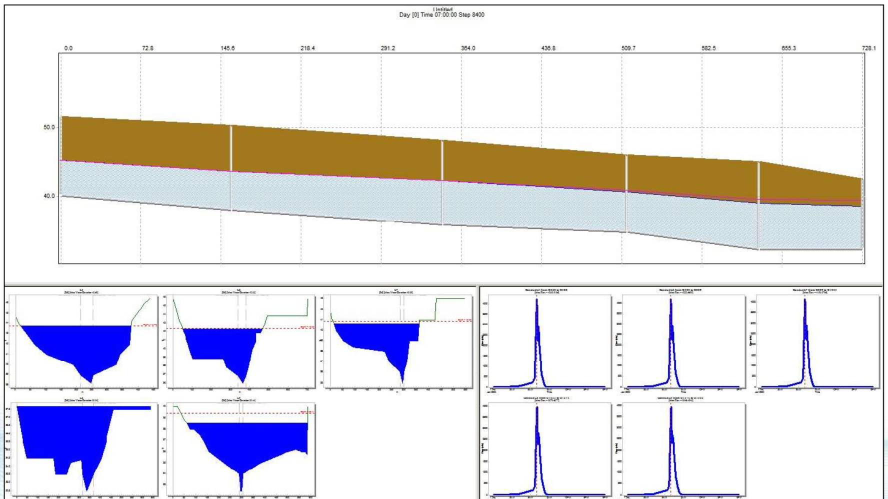
- HGL
- Velocity (weighted average for the link)
- Flow in links
- Losses (losses at maximum reported in Table E13)
- Volume through and maximum for links and nodes

Innovyze®

Page 10

10

Hydraulic Results - Gradeline



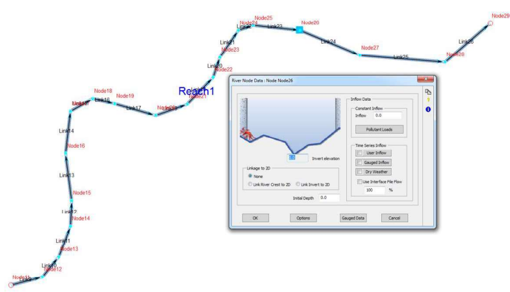
Page 11

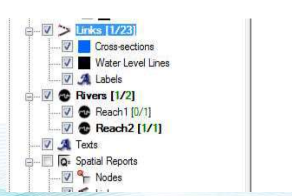
Innovyze®

11

River Links

- Designed for Modeling Natural Channels
- Nodes Automatically Generated at Vertices
- Dialogs Similar to HEC RAS
- Default for HEC RAS Imports
- Each River Reach is Displayed in the Layers Panel





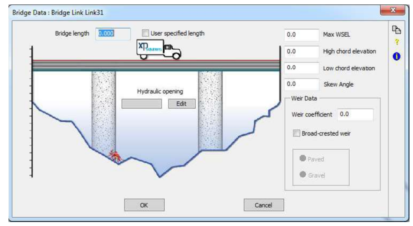
Page 12

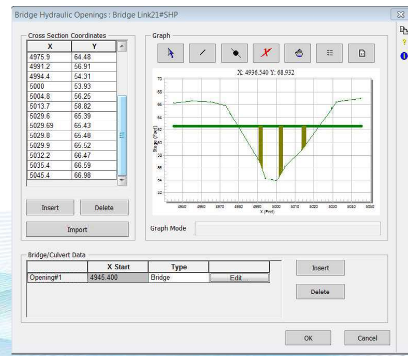
Innovyze®

12

Bridge Links

- Simplified Bridge Modeling
- Represent Numerous Hydraulic Openings – Bridges, Culverts
- Hydraulic Openings stored in Global Database
- Skew Angle
- Piers



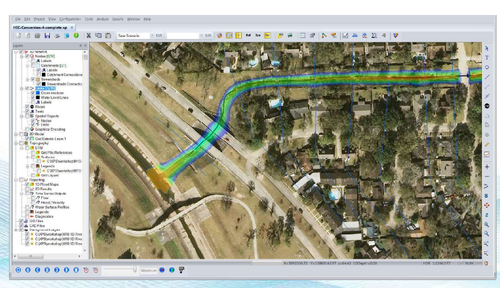


Page 13

13

Workshop Example Preview

- Load Background Images, and DTM
- Import River model from HEC RAS
- Complete the 1D River model including hydrographs
- Utilize River and Bridge Links
- Generate 1D Flood Maps



Page 14

Innovyze®

14

1D River Modeling

The purpose of this training module is to teach users how to use XP's tools to simulate the storage and transport or routing of water through a 1 Dimensional (1D) river system. Users will learn how to build the nodes and links in a river modeling collection network. XP tools will be used to extract data from GIS files as well as to create a digital terrain map (DTM). Other tools in XP will be shown to help users understand and model 1D river modeling.

Objectives

A river model network can be developed in the graphical interface using a variety of methods. In this example, users will learn how to utilize tools to build on the model started in Workshop Example 2 using various tools to add data to the model. You will learn how to:

1. Build a river model from HEC-RAS data import
2. Create a digital terrain model (DTM) from a .ASC grid file
3. Enter the basic configurations settings for solving in Hydraulics mode (HDR)
4. Obtain results from the output file and the Review Results tool
5. Generate Cross-Sections from DTM and add a lateral
6. Prepare 1D Flood maps and animate water levels

Data files to be used are:

1D River Starter.xp including: Image1.jpg and Image1.jpw, Image2.jpg and Image2.jpw, Image3.jpg and Image3.jpw
E116-05-00.g01
whiteoak1.xptin
HECRAS Flows.xls

Building a xpswmm/xpstorm model using HEC-RAS and a DTM

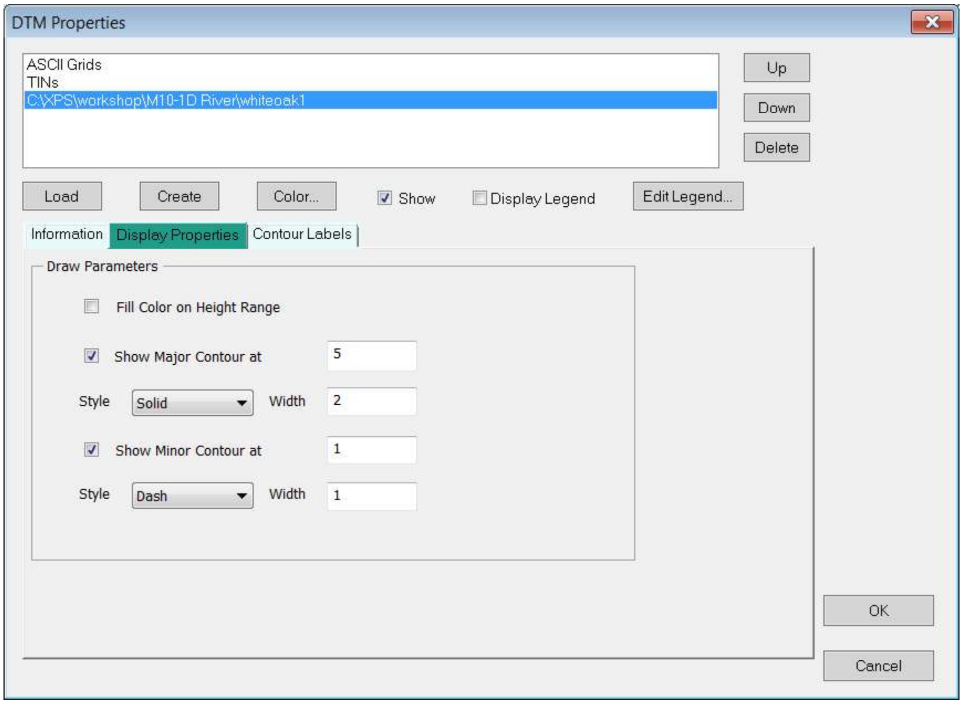
This example will show users how to convert an existing HEC-RAS model to an XPSWMM/XPSTORM model.

Importing HEC-RAS Data

1. Open xpswmm/xpstorm. At the opening dialog, select **Browse**.

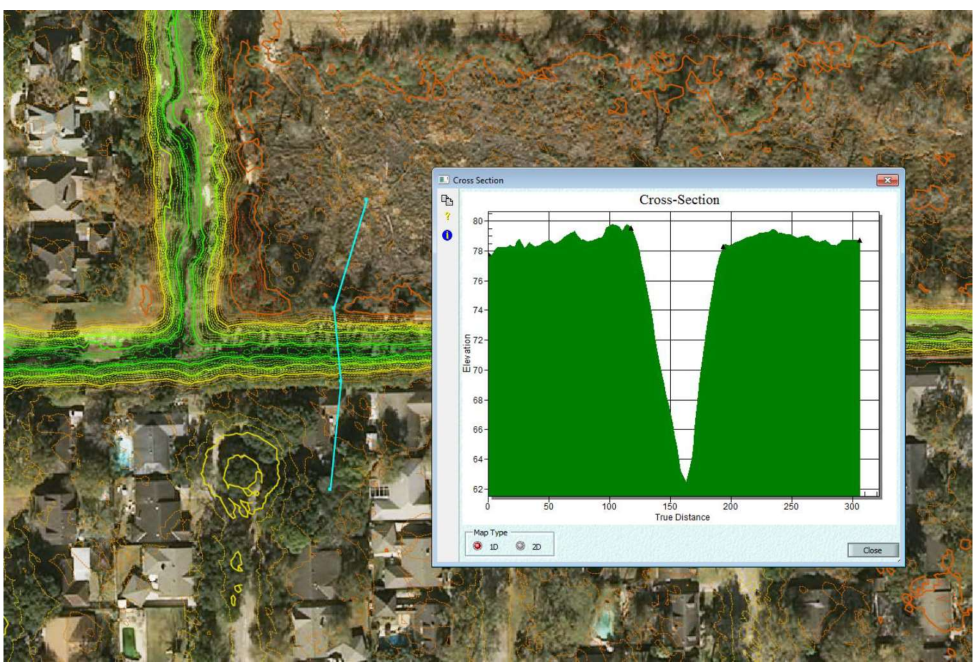
2. In the Windows Explorer, navigate to **XPS\Workshop\M10-1D River** folder and select the file **1D River Starter.xp**. Click on **Open**.

3. **Right-click** on the **DTM** layer, select **Properties**. Choose the **Display Properties** tab and turn off the Fill Color on Height Range and then set the **Major Contour** interval to **5ft** and **Minor Contour** intervals to **1ft**, as shown below.



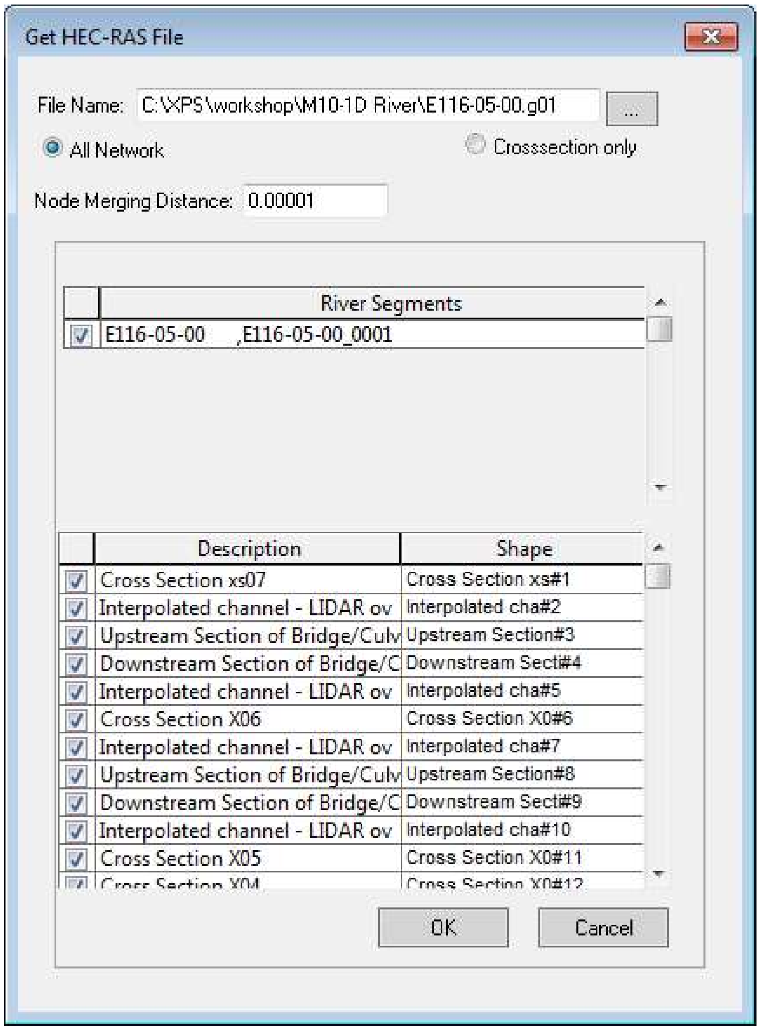
4. From the **File** menu choose **Save As… 1D River.xp**

5. Zoom to a location to clearly see the river and contours. Using the **Section Profile** Tool  compare how well the xptin surface and the aerial photos align at the river reach.



6. Browse to **File**->**Import/Export Data->Import Hec-Ras Data...**

7. The HEC–RAS file geometry file which we will be importing is located in the workshop folder and is named **E116–05–00.G01**. Using the ellipsis button browse to this file in the Module-10 directory and select **Open**.

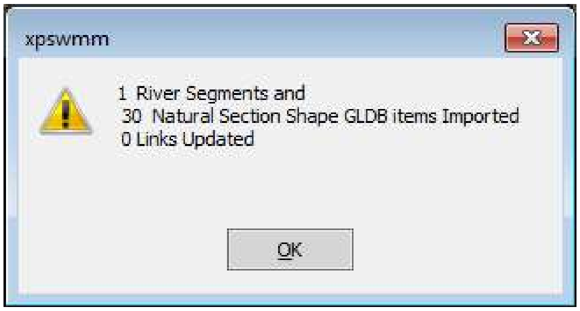


8. After selecting the HEC-RAS geometry file ensure that **All Network** is **selected.** This will import not only the HEC-RAS cross section data but also the reach centerline and apply the cross sections to the appropriate corresponding reach links. By default the **Node Merging Distance** is left at a **default** of **0.00001**.

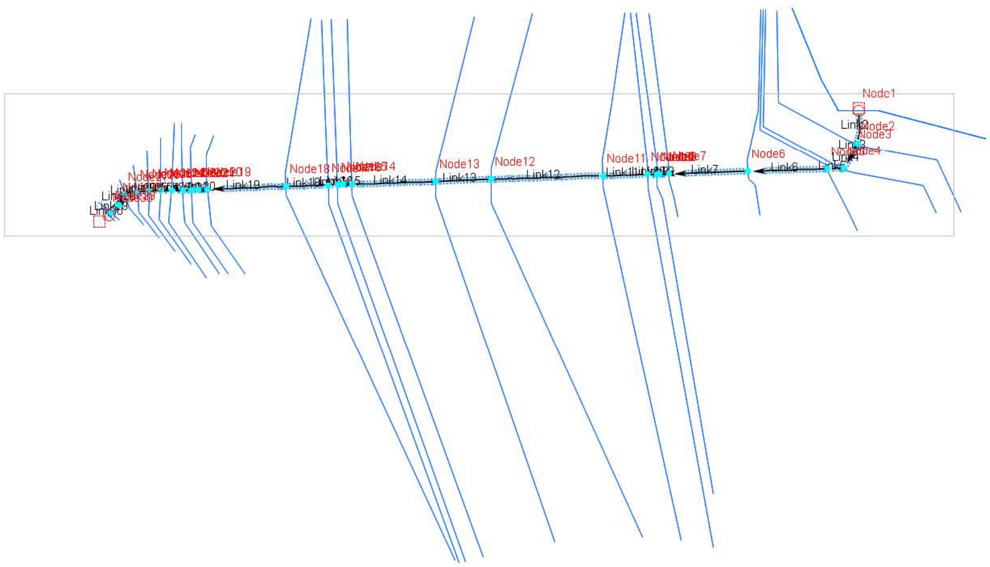
   If only select HEC-RAS cross section and reach data is desired to be imported, the check box for the shapes shown could be unchecked, leaving only the items desired to be imported checked.

   Note that the Natural Channel shape names within xp are limited to 20 characters, HEC-RAS shape names longer than 20 characters will be abbreviated when imported.

9. Select **OK**, the import log message will appear showing what objects are updated. Select **OK** at this dialog.

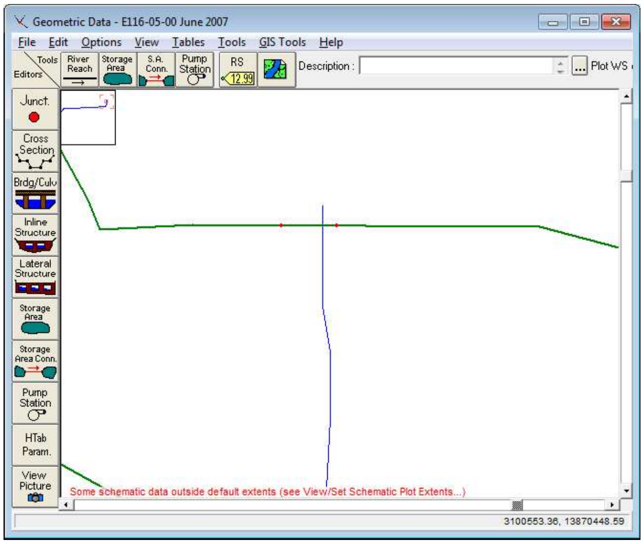


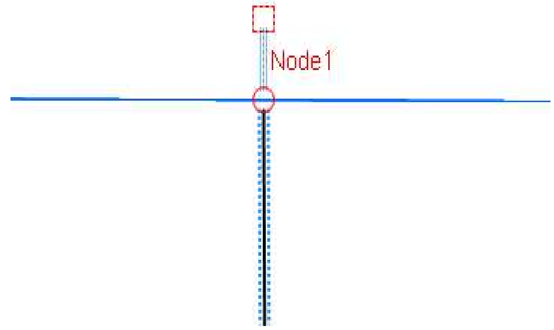
10. The xp model objects are now imported and the error.log file will be generated, listing any warnings or errors encountered during the import. Most of these will be regarding processing of bridge data due to the difference in model setup for bridges.



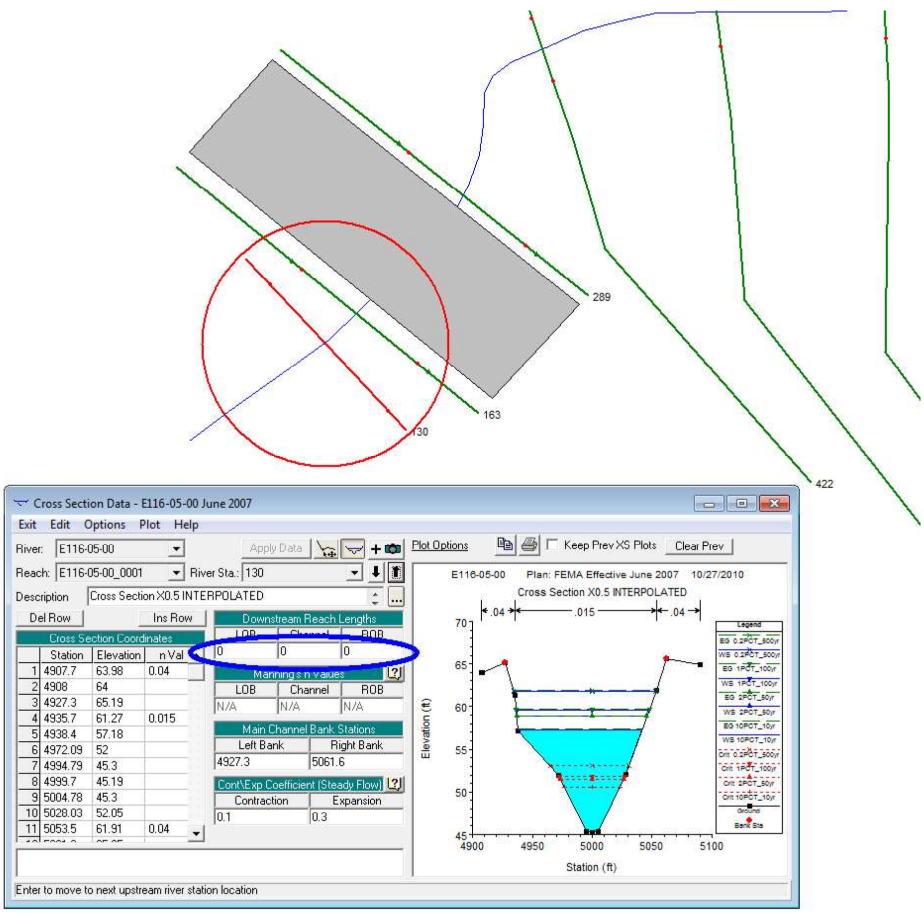
11. Check the imported model - review the most u/s and d/s nodes and links. The u/s most node and link are 'extra' objects which are brought into the model based on the segment of HEC-RAS river reach centerline which extends beyond the top most cross section.

    Since this u/s node and link are not hydraulically significant and have missing data, Node0 and Link1, have been automatically made inactive.

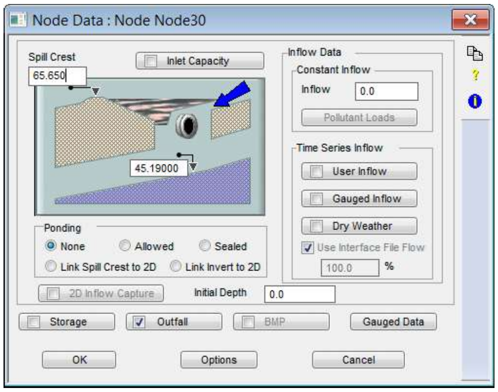


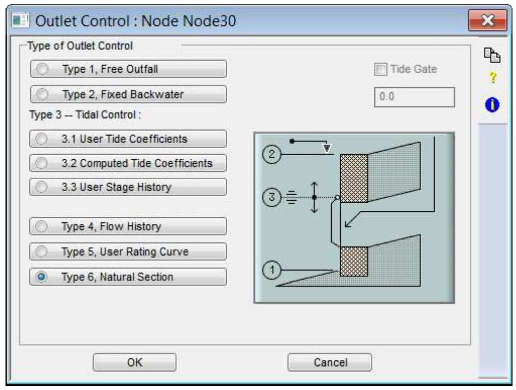


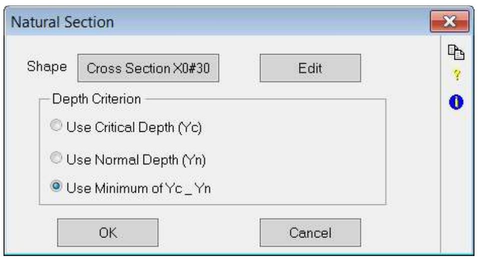
Likewise, the d/s most node and link are created from the HEC-RAS reach geometry though are not used in the hydraulic simulation. As shown below, the final cross section within the HEC-RAS model has a zero length and is only used as a boundary condition. Hence **Node31** and **Link31** have also been made **Inactive**.



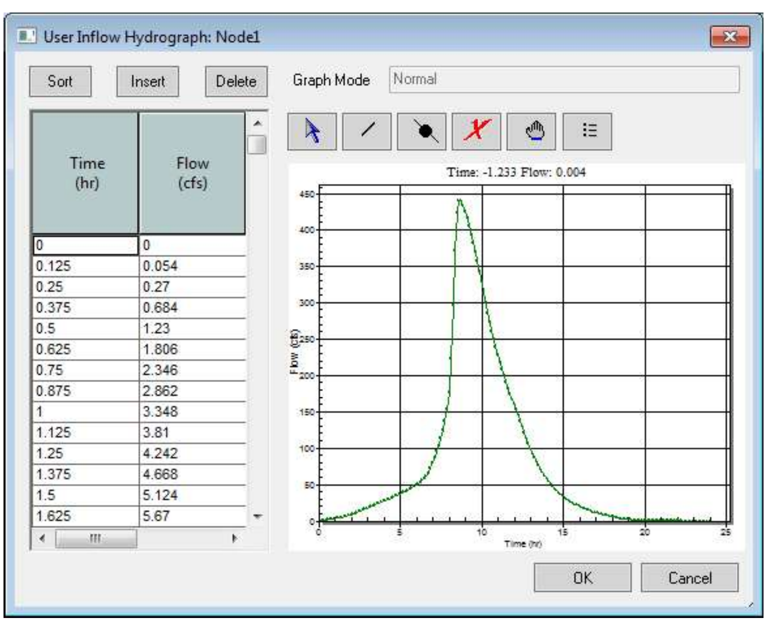
12. **Double-click** the new d/s most node, **Node30**, select **Outfall** and then **Type 6, Natural Section**.







13. HEC-RAS flow hydrographs can be added to the xp model by viewing the chosen dss hydrograph in HEC-RAS and copying the data to an excel spreadsheet. The date/time typically need to be adjusted to decimal hours. This has been done already and is in the file called **HECRAS flow.xls**. **Double-click** on **Node1** and click on **User Inflow** under Time Series Inflow. Insert one row and copy the time and flow hydrograph data from the Upstream tab into the dialog. Click **OK** twice to return to the project window.



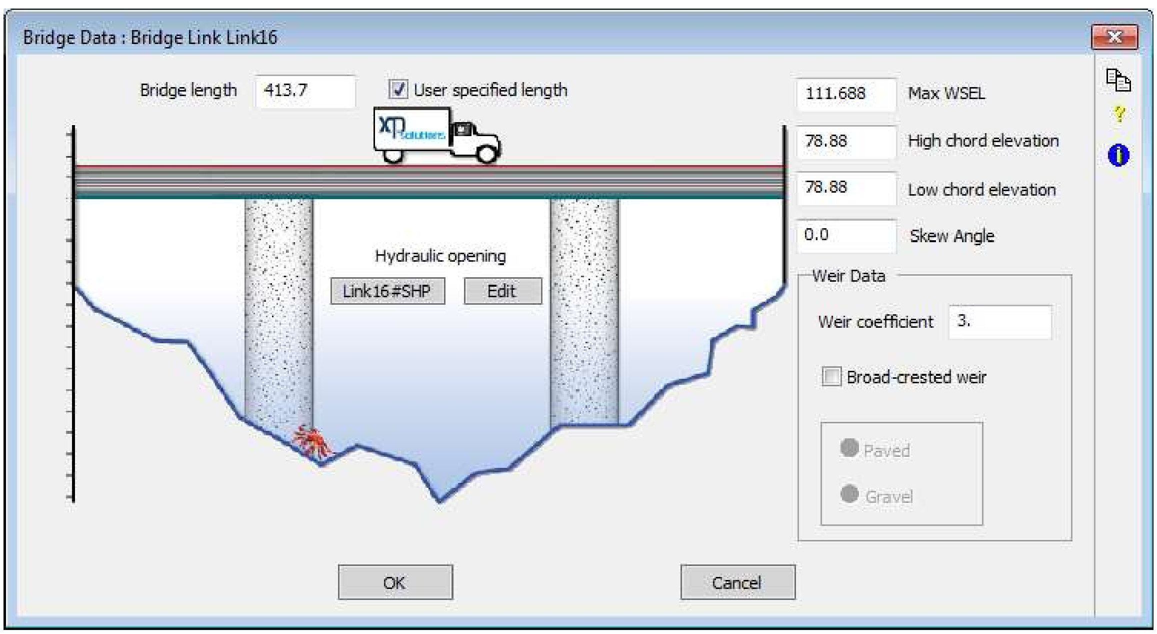
Repeat this process and copy flow from the **IN-LINE** tab to **User Inflow** at **Node12**.

14. Browse to the **Hydraulic Job Control**, update the **Time Control** if not already a **24 hour** simulation duration. Set the **Save Results Every** interval to be **5** minutes.

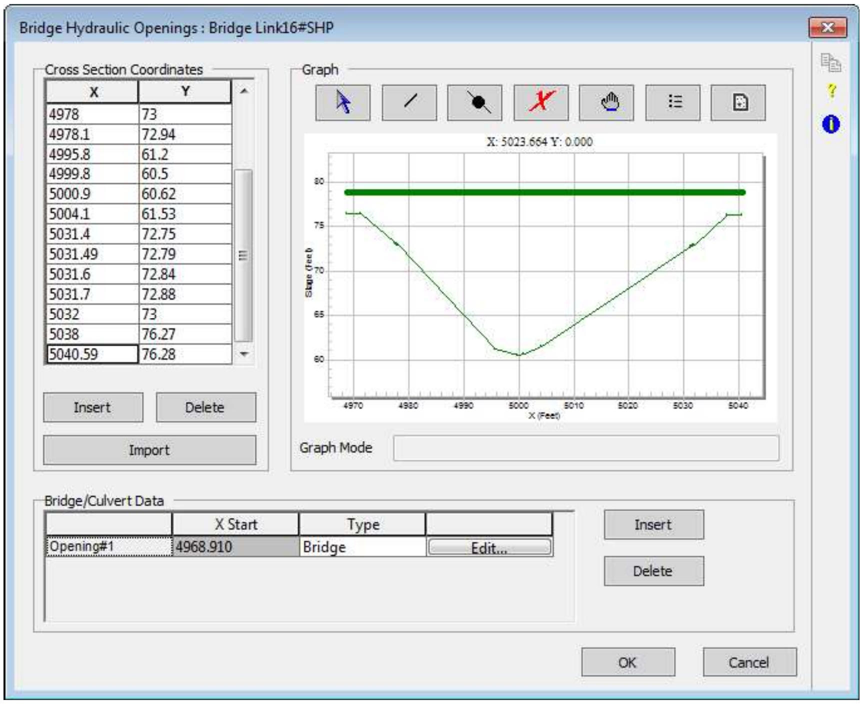
15. **Save** your model as **1D River2.xp**.

16. Most bridge data is imported from HECRAS models, however we will manually update some bridge crossings to ensure completeness.

17. Use **CRTL-F** to find **Link 16. Double-click** Link 16 to get the bridge dialog.



18. Select **Edit** for the Hydraulic Opening. In that dialog from the bottom and the top delete the rows representing the left and right overbanks to trim the section to just the part under the bridge deck. This would leave data from only stations 4968.91 and 5040.59 as shown below. Press OK twice to return to the plan view. **Save** the model.



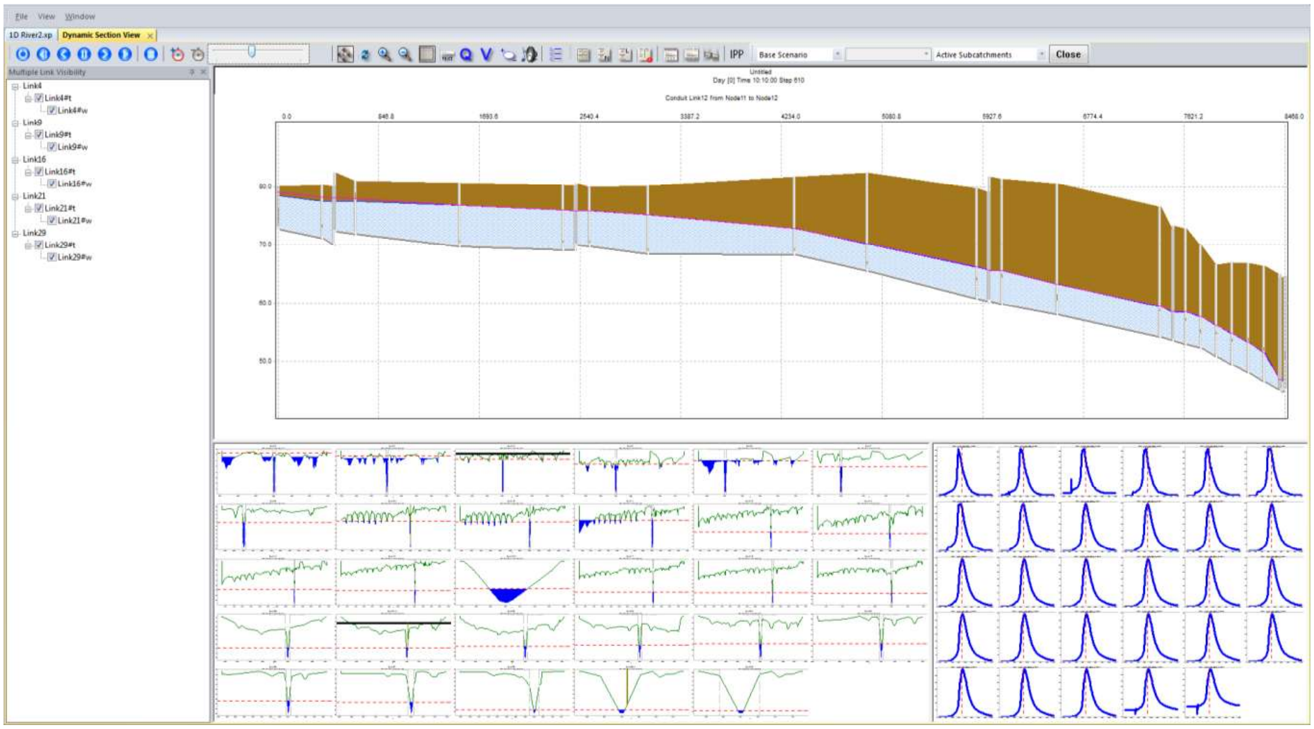
Model results

1. **Solve the model** by browsing to Analyze->Solve, by selecting the toolbar icon or by pressing F5.

2. A list of model errors and/or warnings will be shown in the Error.log file. Note that there are no errors in the model, so we can proceed with the simulation by pressing **Yes**, on the warning message check dialog if enabled. The warnings listed in the error file are due to the roughness values adopted from the HEC-RAS model being outside the typical range of open channel roughness values.

```
Data Generation Diagnostics

WARNING: HDR: Links 'Link2': The mannings roughness, 0.99, for the left overbank, is out of reasonable range.
WARNING: HDR: Links 'Link2': The mannings roughness, 0.99, for the right overbank, is out of reasonable range.
WARNING: HDR: Links 'Link7': The mannings roughness, 0.99, for the left overbank, is out of reasonable range.
WARNING: HDR: Links 'Link7': The mannings roughness, 0.655, for the right overbank, is out of reasonable range.
WARNING: HDR: Links 'Link12': The mannings roughness, 0.903, for the left overbank, is out of reasonable range.
WARNING: HDR: Links 'Link13': The mannings roughness, 0.68, for the left overbank, is out of reasonable range.
WARNING: HDR: Links 'Link14': The mannings roughness, 0.933, for the left overbank, is out of reasonable range.

0 Error(s) and 7 Warning(s) were encountered
Data Export Completed Successfully
```

3. After solving, hold **Shift** while clicking the **Select All node** and **Link** keys , to select only the active network objects, then select **Dynamic Section Views**



4. Animate the HGL or slide the time control midway to see the water fill the channel cross sections and profile.

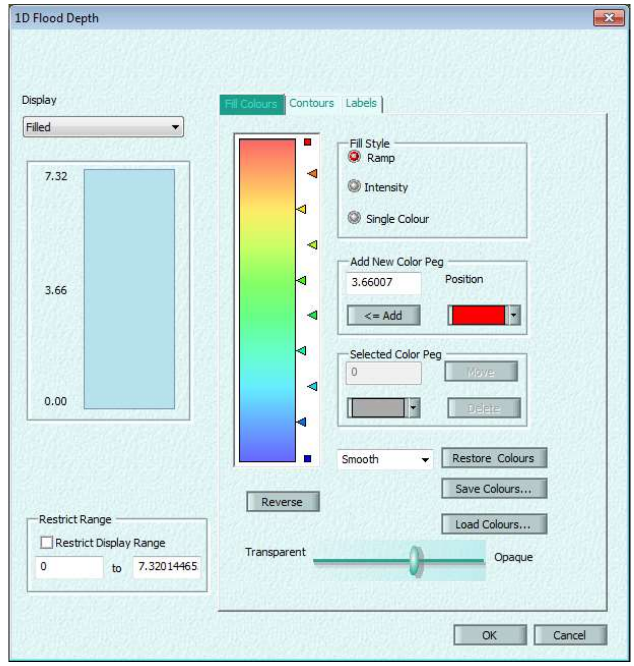
5. Save the model using **Save As… 1D River3.xp**

1D Flood Maps

1. The upstream most cross section is fully inundated outside the channel banks. We can use the 1D Flood Maps tool in order to create a flood map of the simulation based on the 1D channel WSL results and the xptin surface.

   In the Layer Control Panel turn on the **1D Flood Maps**. The 1D hydraulic results will be interpolated against the xptin surface at each time step – providing not only a maximum extents but a simulation animation of the results. This process may take several seconds to complete.

2. Right-click on the **1D Flood Maps** and choose **Properties**. **Reverse** the color gradient and move the **Transparency** slider to about 50%. Click **OK**.

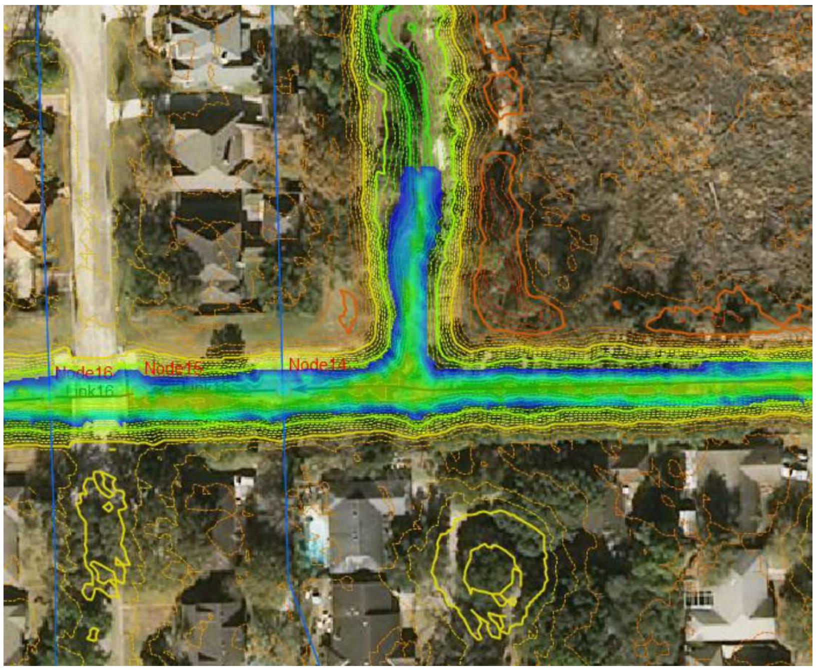


3. Use the **Green Arrow** (Maximum) from the Animation Controls in the Status Bar to show the maximum flooded extent.

Adding in Lateral Catchment

1. Use **Save As…** using the name **1D River4.xp**.

2. Using the 1D Flood Maps tool we can see that immediately u/s of Node14 there is a lateral drainage area which is not being represented in the original HEC-RAS model. In our model we can see the 1D Flood Map back filling a lateral channel which should be incorporated into our study area.



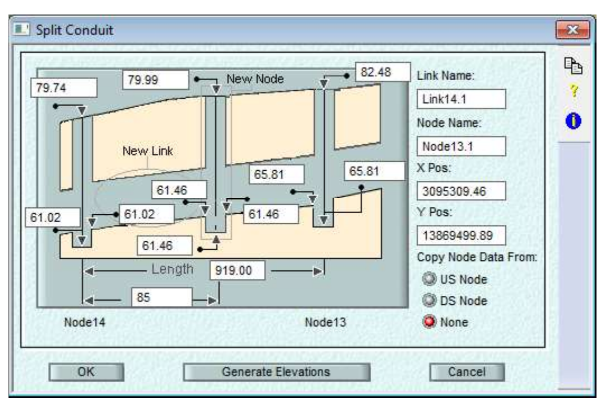
We will add the Lateral Channel segments, including the bridge crossing, into the model as well as model the inflow into this reach by defining the hydrologic catchment in the Runoff mode.

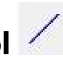
3. Turn off **1D Flood Maps**.

4. The Confluence of the lateral and the main reach is **85 ft** upstream of **Node 14**. We need to Add in a node at this point to create the junction. First step is to **right-click** on **Link14** and select **Non-river Link** since a River Link cannot be split.

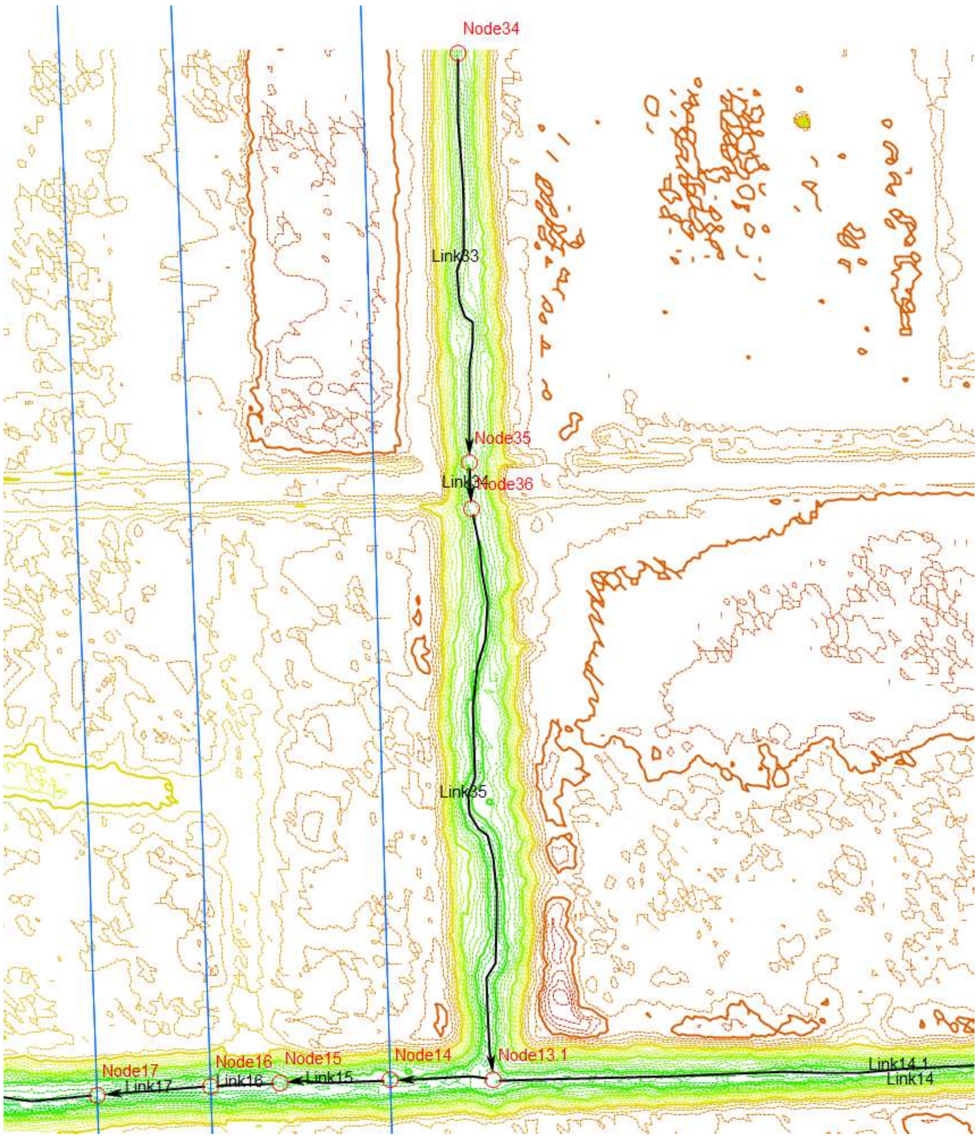
5. Next **right-click** on **Link14** and select **Split Conduit**.

6. By default, the conduit will be split in half, update the d/s split length to **85 ft** and select **Generate Elevations**. Click **OK**. This will create a new node at the desired location and duplicate the link geometry upstream and downstream of the new node.



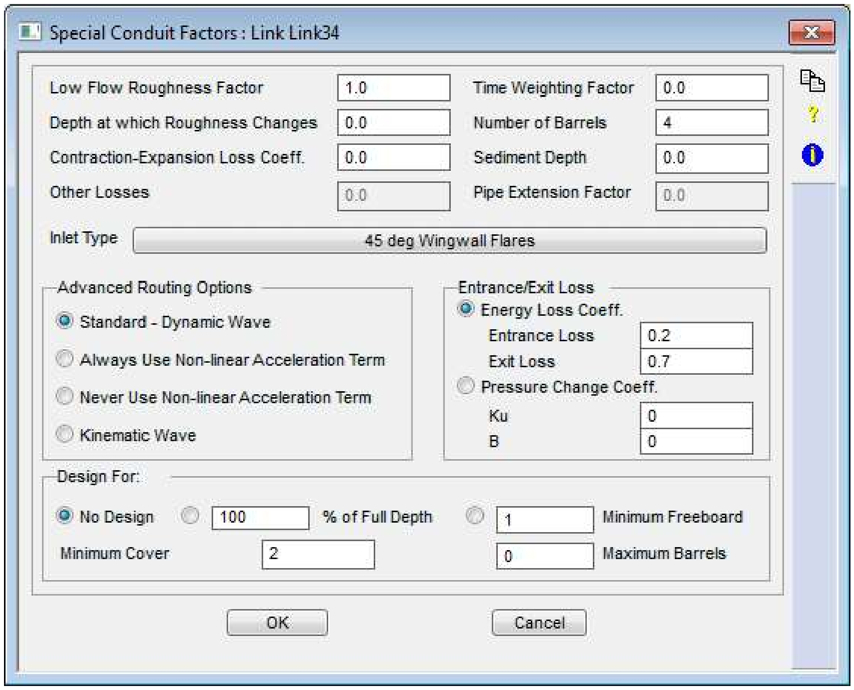
7. Begin drawing the lateral reach at the u/s end. **Select** the **Link drawing tool**  and **click once** at the edge of the xptin surface to draw the first node. While **holding CTRL** click along the centerline of the lateral reach until you reach the bridge. **Release CTRL**, click once on each side of the bridge to draw the bridge objects, then hold CTRL and click along the lower lateral reach length centerline until Node13.1, then double-click on Node13.1 to finish.

   Note: To save time simply import the Lateral.XPX file which will create the 3 nodes and 3 links.

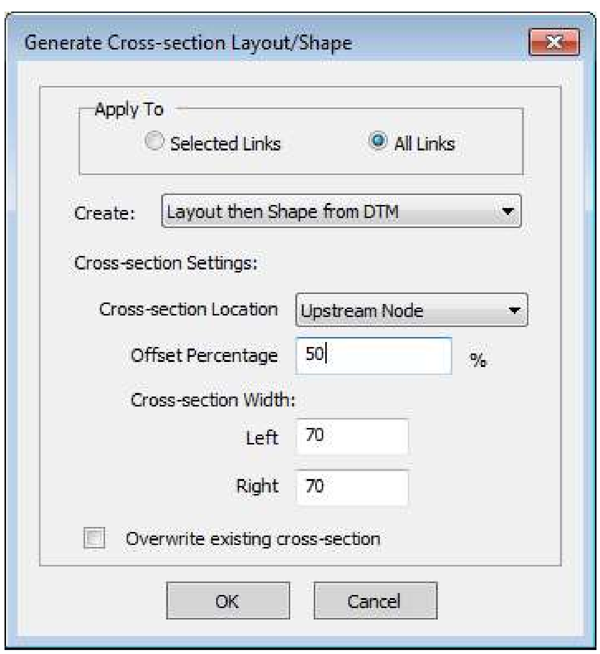


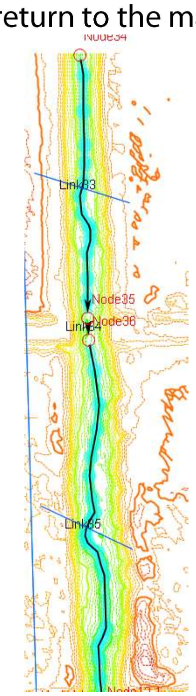
8. Edit **Link33** and **Link 35** and select **Natural** as the link type. Now, the link types are Natural for the reach segments u/s and d/s of the bridge. Double-click and open **Link34** and change the conduit type to **Rectangular**. Note that this link represents the 4 Box Culverts under the bridge span. Enter the height and width as **5 ft**.

Tick on the **Conduit Factors** box and enter the **number of barrels** as **4**. There is also an inlet wall, from **Inlet Type** select **45 deg Wingwall Flares**. Add **0.2 Entrance** and **0.7 Exit** loss values. Click on **OK** twice and return to the network window.



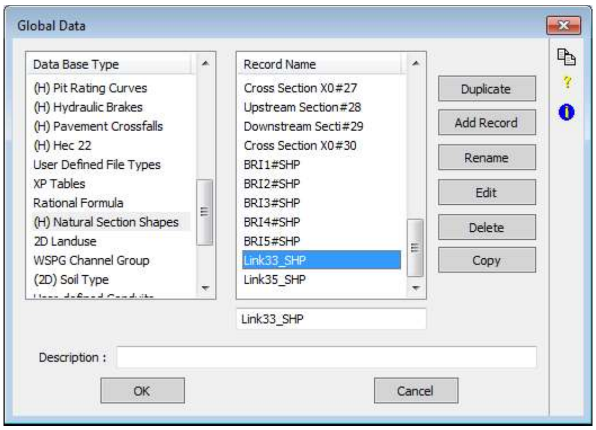
9. Get cross-sections for the links. Select **Link33** and **Link35** and go to the **Tools->Calculate Conduit/Cross-sections** menu. Complete the data entry as shown in the dialog. Click on **Yes** when prompted about overwriting any existing cross sections and you will be able to see the list of new cross-sections obtained. Click on **OK** and return to the main window.

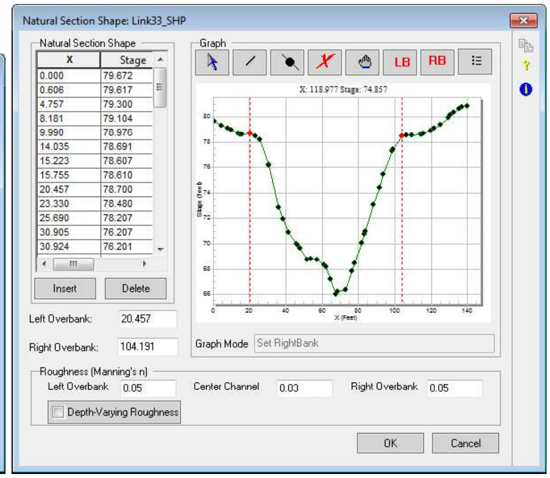




10. Select **Configuration->Global Data**. Select **Natural Section Shape** as the Database Type. Highlight and **Edit Link33_SHP**. Click on the **LB** button and click on the cross-section to add the left overbank station. Similary add right overbank also. Enter the **Mannings n** values for the overbanks as **0.05** and main channel as **0.03**.

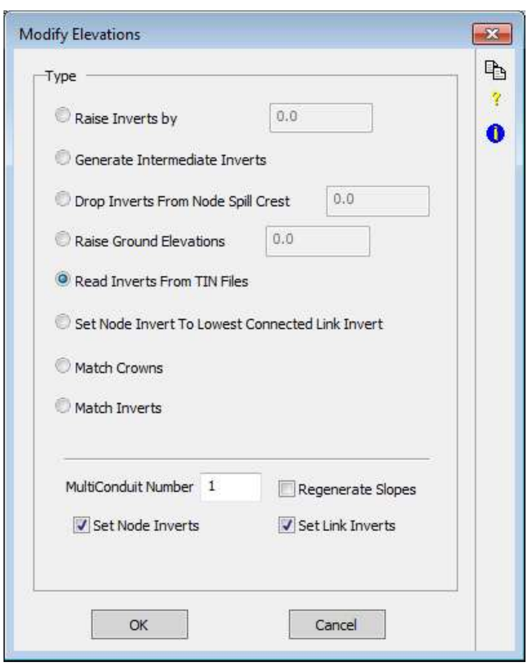
    Note: An XP-Table could be created for the Natural Channel cross sections in order to allow a fast copy and paste for all of the roughness values.

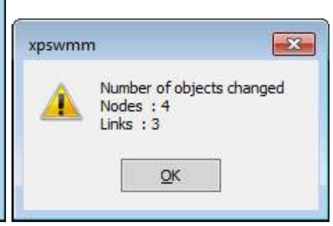




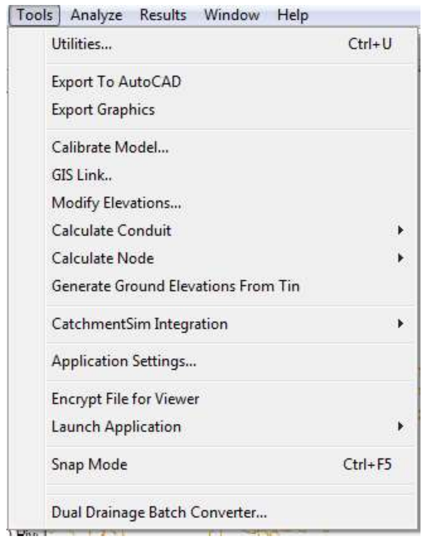
Repeat this procedure using the same roughness values for **Link35_SHP**.

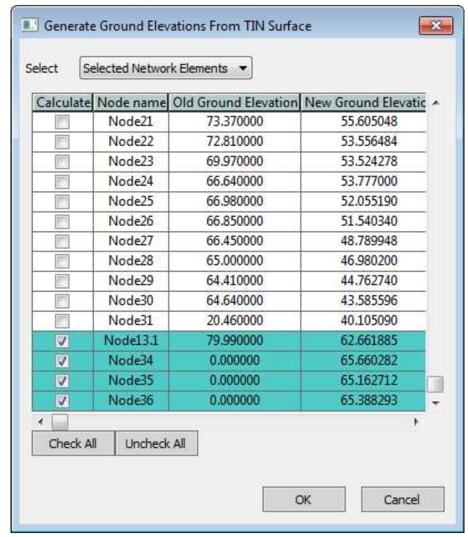
11. Select the lateral reach objects, from **Node34 to Node13.1**, go to the **Tools Menu** and access the **Modify Elevations** dialogue. Select the **Read Inverts from TIN Files** and switch on the **Set Node Inverts** and **Set Link Inverts** tick boxes. Click on **OK** and you will see the message 4 nodes and 3 links have been modified. Click on **OK**.



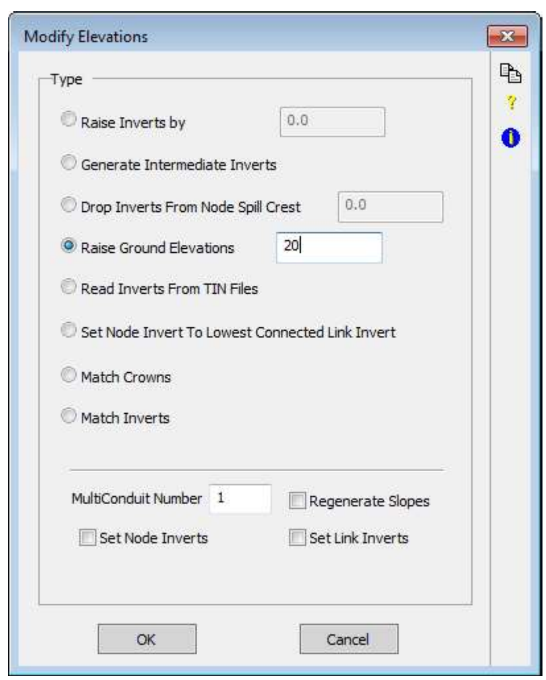


12. Now we will create the ground elevations for the nodes. With the same objects selected, from the **Tools Menu** select the option **Generate Ground Elevation from Tin.** Ensure that **Selected Network Elements** is chosen, scroll down to see the lateral reach 4 node objects. Click on **OK.**





Now, select **Node34**, **Node35** and **Node36,** from **Tools Menu/Modify Elevations**, **Raise Ground Elevations** by **20 ft**.



13. Select **Link33**, **Link34** and **Link35**, and go to the **Tools Menu/Calculate Conduit/Lengths**. Choose **Selected** and Calculate. Repeat the process for **Calculate Conduit/Slopes**.

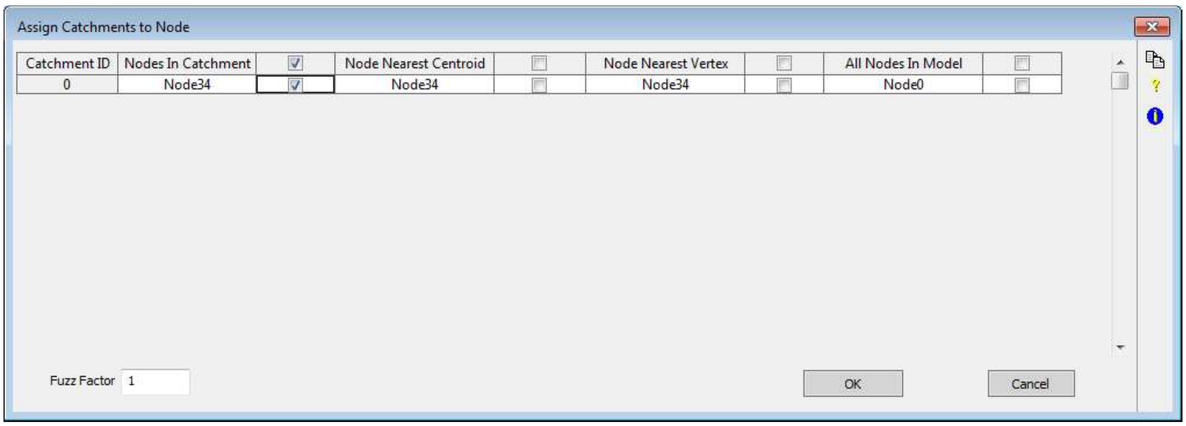
14. **Save** the model.

Add Lateral Catchment Inflow Hydrology

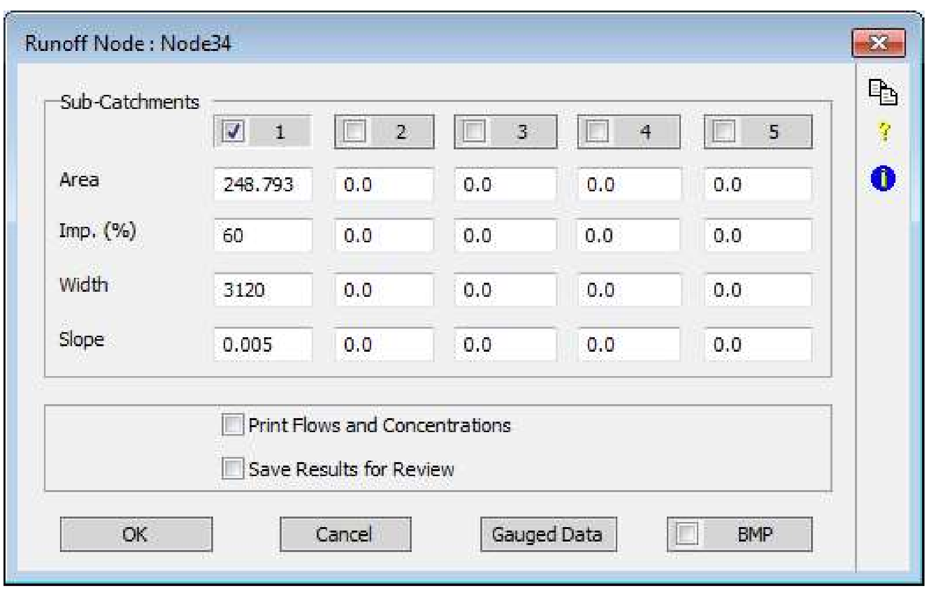
15. **Right click** on the **Catchments** layer and select **Import from GIS**. Browse to the M10 folder and select **Catchment.shp**. Uncheck the import attributes flag and click **OK**.

16. Switch to the **Runoff mode**, and make **Node34 Active**. Under **Tools->Calculate Node->Catchment Drainage**, click Nodes in Catchment to automatically link the polygon to Node34. Note that other options also would have connected to polygon to Node34.

    Alternatively, the catchment can be manually connected to the node by selecting the catchment centroid and dragging it to the node.

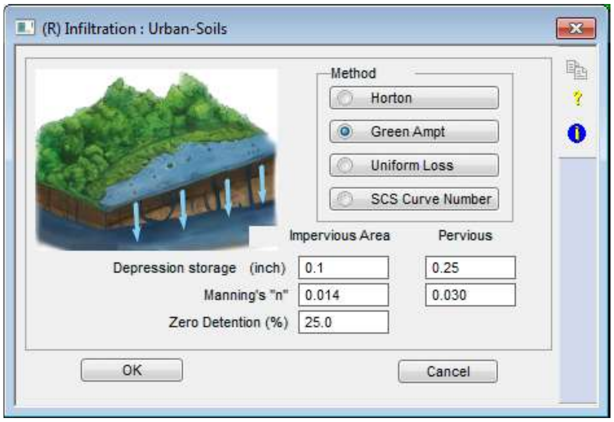


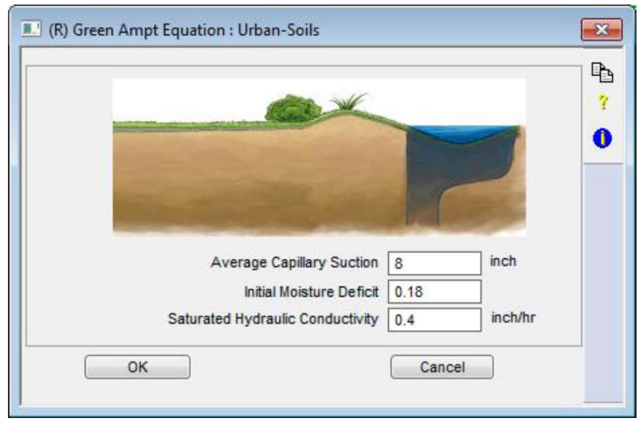
17. Edit **Node34** and enter the Sup-Catchment data as shown below:



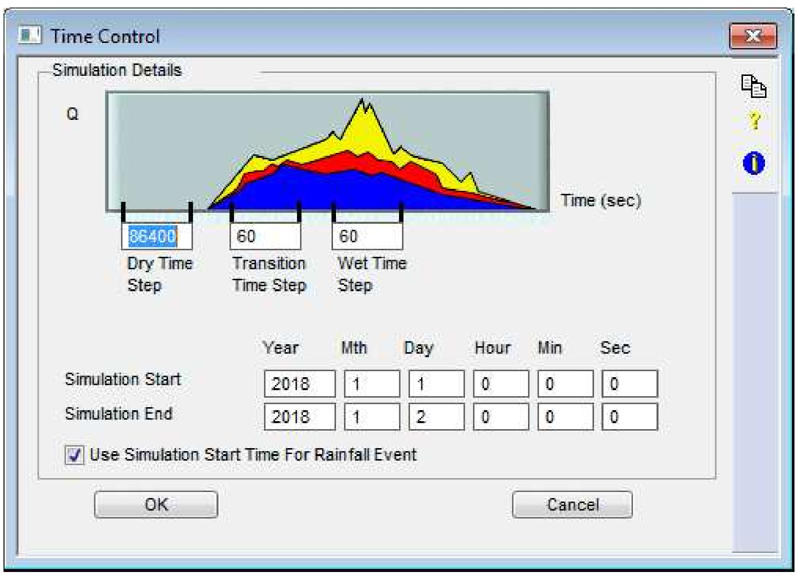
18. Click on Sub-Catchment 1 flag, as we are using the SWMM Runoff routing method the rainfall and infiltration must both be specified. Click the **Rainfall button**, highlight the **SCS Type II rainfall record** and select **Edit**. Under **Constant Time Interval**, update the **multiplier** to **6.1** inches – representing the total depth of the 100-year rainfall ARI event. Click **OK** twice and then **Select**.

19. Click on **Infiltration** and type in name **Urban-Soils**, then select **Add**, then **Edit**. Update the Infiltration record based on the images below. Select this record then click **OK** twice to accept the sub-catchment data.

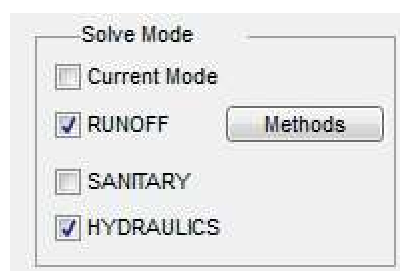




20. Edit the **Runoff Job Control->Time Control** to set a 24-hour simulation period and check the **Use Simulation Start Time For Rainfall Event** option, as shown below.



21. Browse to **Configuration->Mode Properties**, under **Solve Mode** uncheck **Current Mode** and select both **Runoff** and **Hydraulics** modes. Within the **Hydraulic Job** Control enable the **Run Hydrology/Hydraulics Simultaneously** option.



22. **Save** the model.

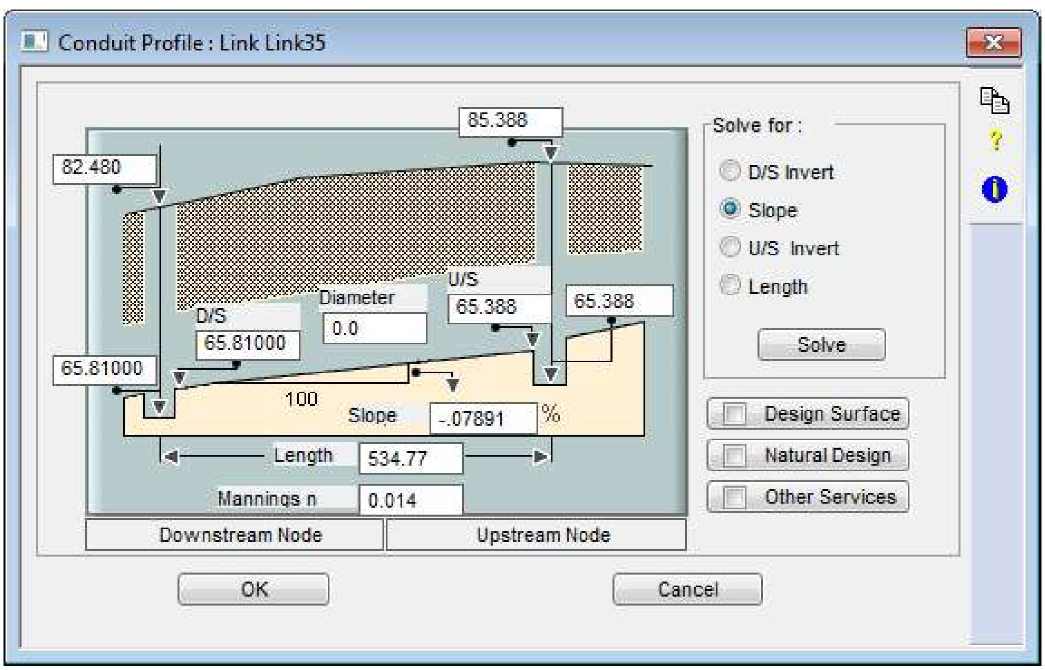
23. Now click on the **Solve** icon button to analyze the model. You may get a few error messages as shown:

```
Data Generation Diagnostics

ERROR: HDR: Links 'Link14': Downstream Conduit invert below Node invert
ERROR: HDR: Links 'Link14.1': Upstream Conduit invert below Node invert
WARNING: HDR: Links 'Link34': Conduit Length (30.75) less then minimum length for Analysis (32.81): Set Configuration Parameter MINLEN=
WARNING: HDR: Links 'Link2': The mannings roughness, 0.99, for the left overbank, is out of reasonable range.
WARNING: HDR: Links 'Link2': The mannings roughness, 0.99, for the right overbank, is out of reasonable range.
WARNING: HDR: Links 'Link7': The mannings roughness, 0.99, for the left overbank, is out of reasonable range.
WARNING: HDR: Links 'Link7': The mannings roughness, 0.655, for the right overbank, is out of reasonable range.
WARNING: HDR: Links 'Link12': The mannings roughness, 0.903, for the left overbank, is out of reasonable range.
WARNING: HDR: Links 'Link13': The mannings roughness, 0.68, for the left overbank, is out of reasonable range.
WARNING: HDR: Links 'Link14': The mannings roughness, 0.933, for the left overbank, is out of reasonable range.
ERROR: HDR: Links 'Link14': Upstream Crown or Top of Bank (82.48) above Node Ground Elevation (78.88)
WARNING: HDR: Links 'Link14.1': The mannings roughness, 0.933, for the left overbank, is out of reasonable range.

3 Error(s) and 9 Warning(s) were encountered
Data Export Was NOT Completed Successfully
```

These error messages refer to Node13.1 in the Hydraulic mode which was adjusted vertically when the lateral reach was configured. Select **Node13.1** and browse to **Tools->Modify Elevations**, and select **Set Node Invert to Lowest Connected Link Invert**.

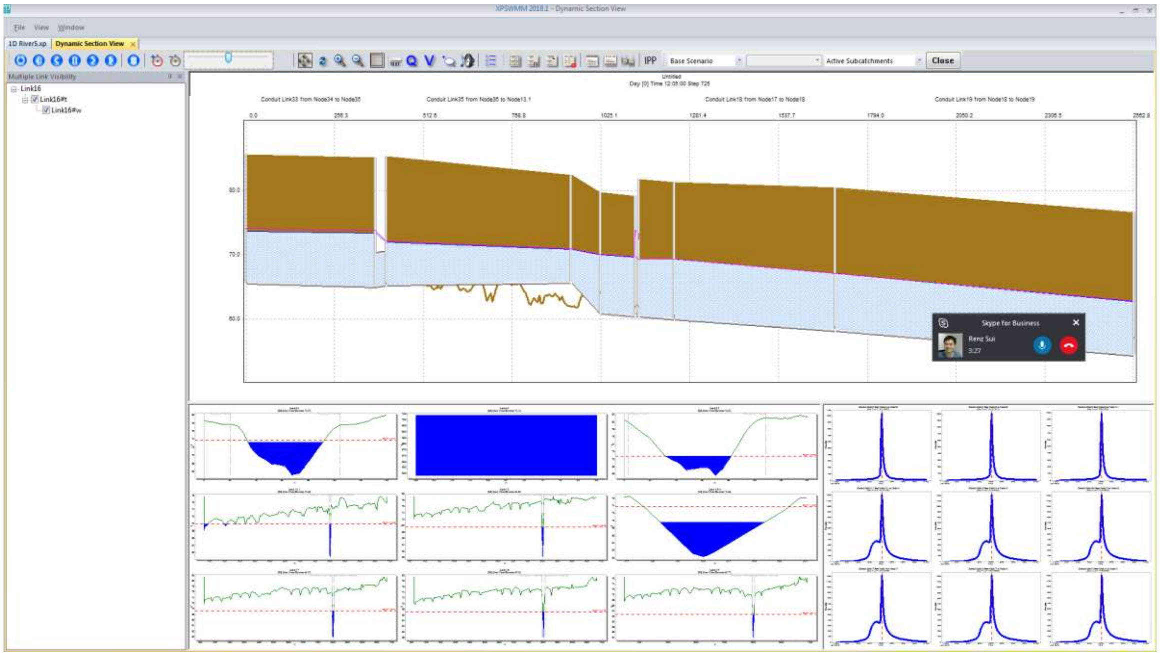
To fix the error related to Link14 **right-click** on the **Link14** and **Link14.1** and choose **River Link**. Edit the **Node13** and increase the **Ground Elevation** to **82.48**.

Then to complete the edits to resolve the errors select **Link35** then **F3** to get the conduit profile. Make the downstream invert match the node invert and resolve for the slope.

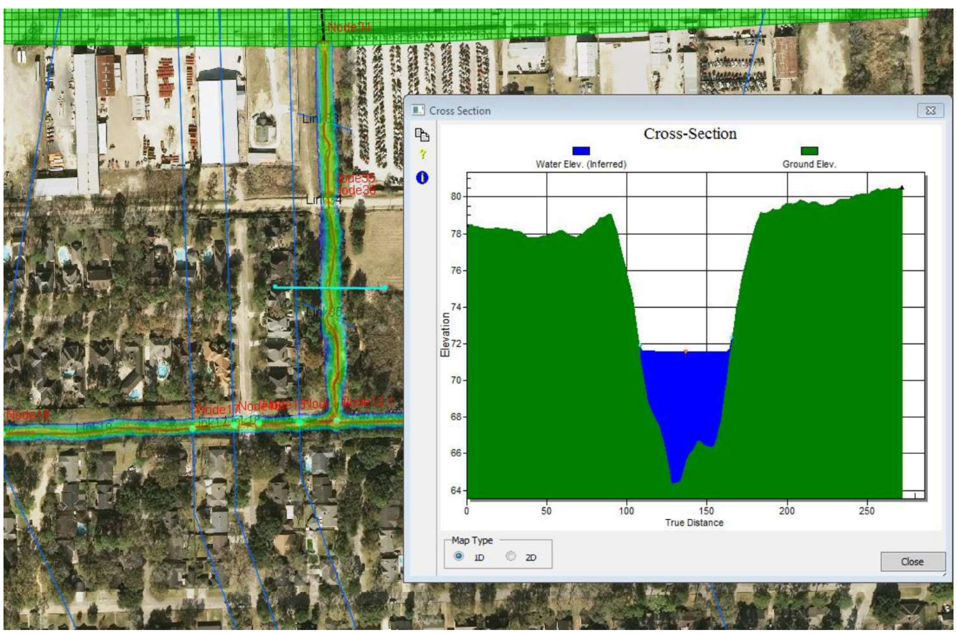


24. **Save** the model and then **Solve**. Select **Yes** to proceed with the simulation despite any remaining roughness value warnings.

25. Select from the top of the lateral reach to the end of the network, **Node34 to Node19**, and review the **Dynamic Section Views** for this reach segment.



26. As before, **select** the **1D Flood Maps** layer and allow the 1D hydraulic results to be triangulated into simulation extents maps for the current model configuration.



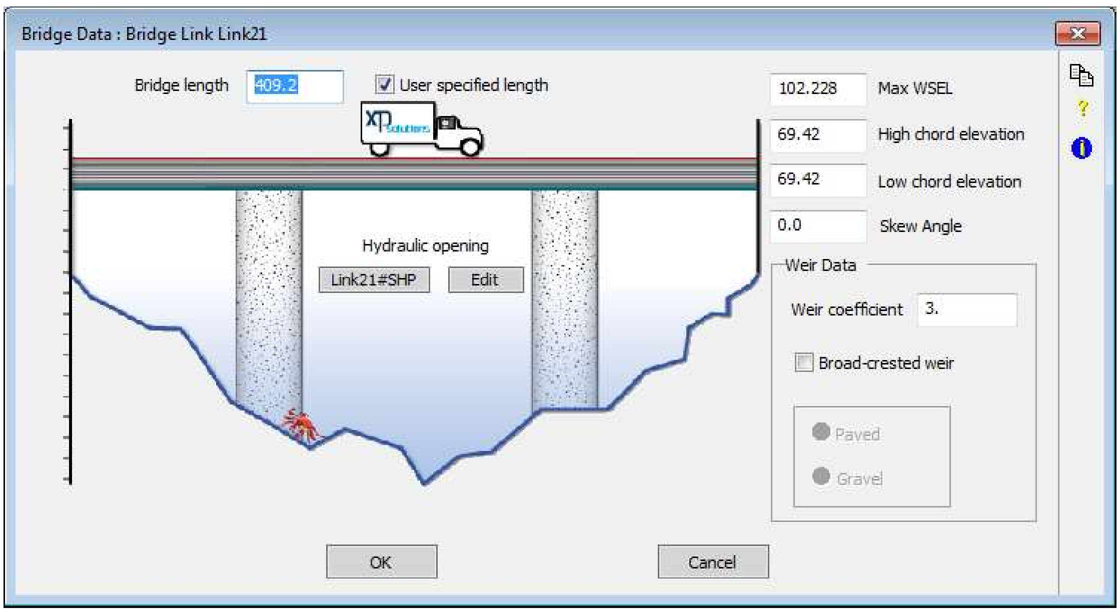
27. **Save As…** and use the name **my1D River6.xp.**

Modeling bridges

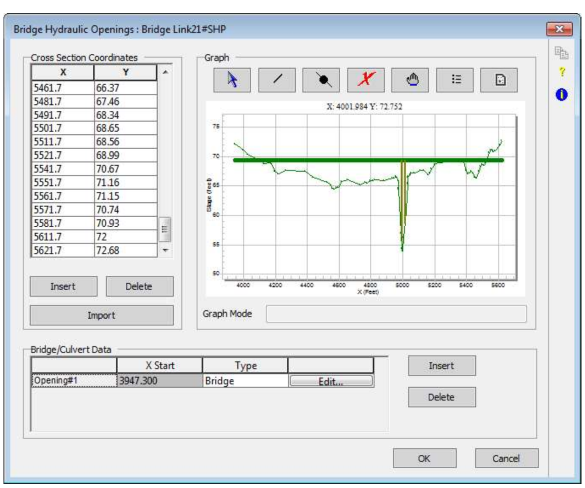
1. Open **1D River6.xp** if not directly continuing from the previous section.

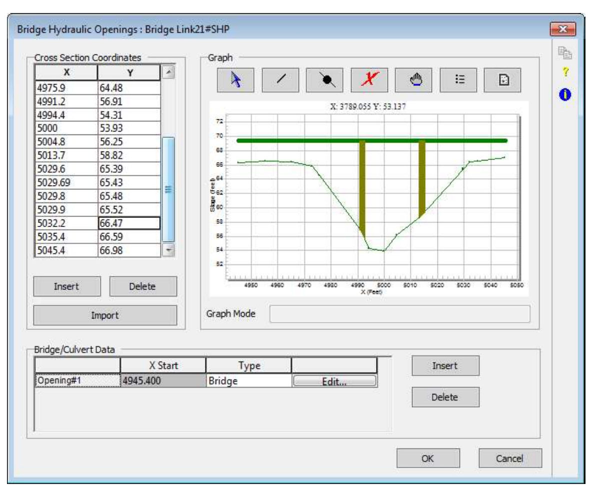
2. There is another bridge crossings, at **Link21** which we can inspect and improve by clipping the Hydraulic Opening. **Right-click** on **Link21**, then **Edit Data**.

3. **Bridge Link Data** …dialog shows the **Max WSEL**, **High/Low Chord** and **Weir Data** as shown below – note the bridge deck overtopping occurs at the High Chord Elevation and the Max WSEL is the level at which the bridge deck overflow would become pressurized. In our case this was set arbitrarily high at import.

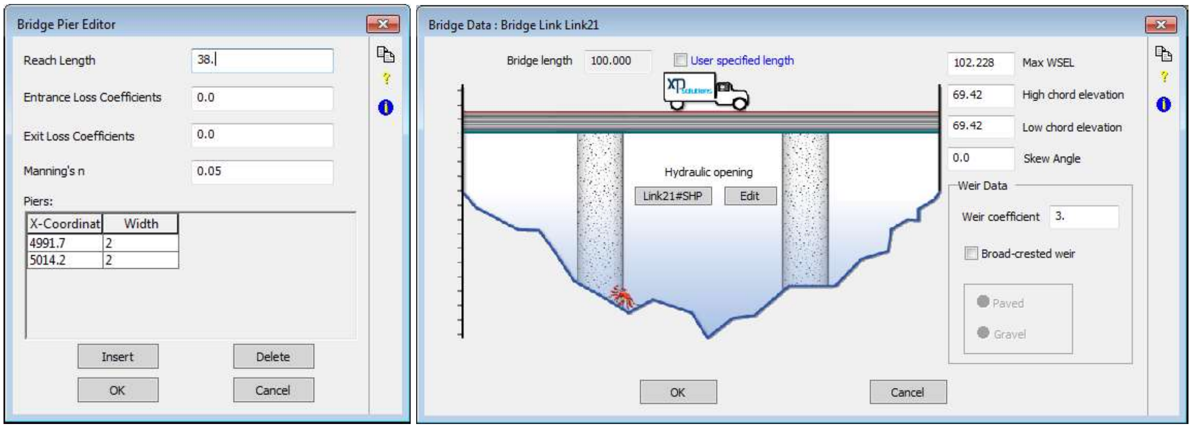


4. **Edit** the **Hydraulic Opening**. Notice the default Hydraulic Opening uses the entire cross section imported from HEC-RAS. This should be trimmed down to represent only the bridge opening. **Delete** the **cross section points** in the X,Y table so that the table range is from **X=4945.4** to **X=5045.4**.

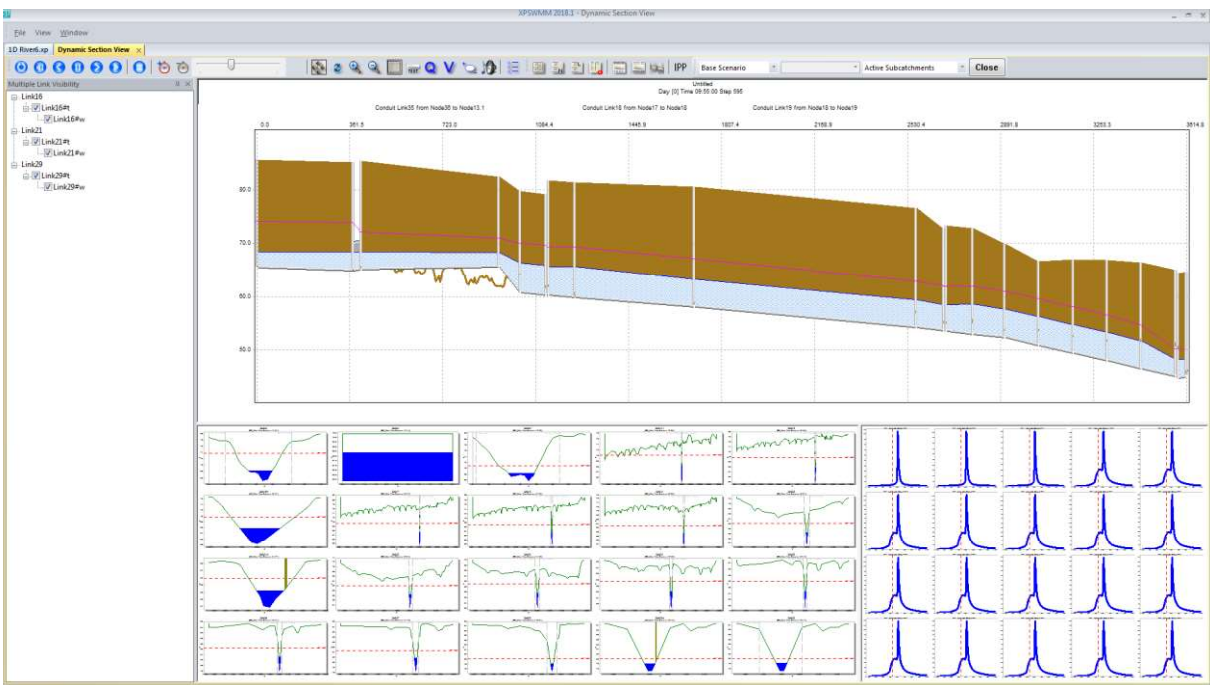




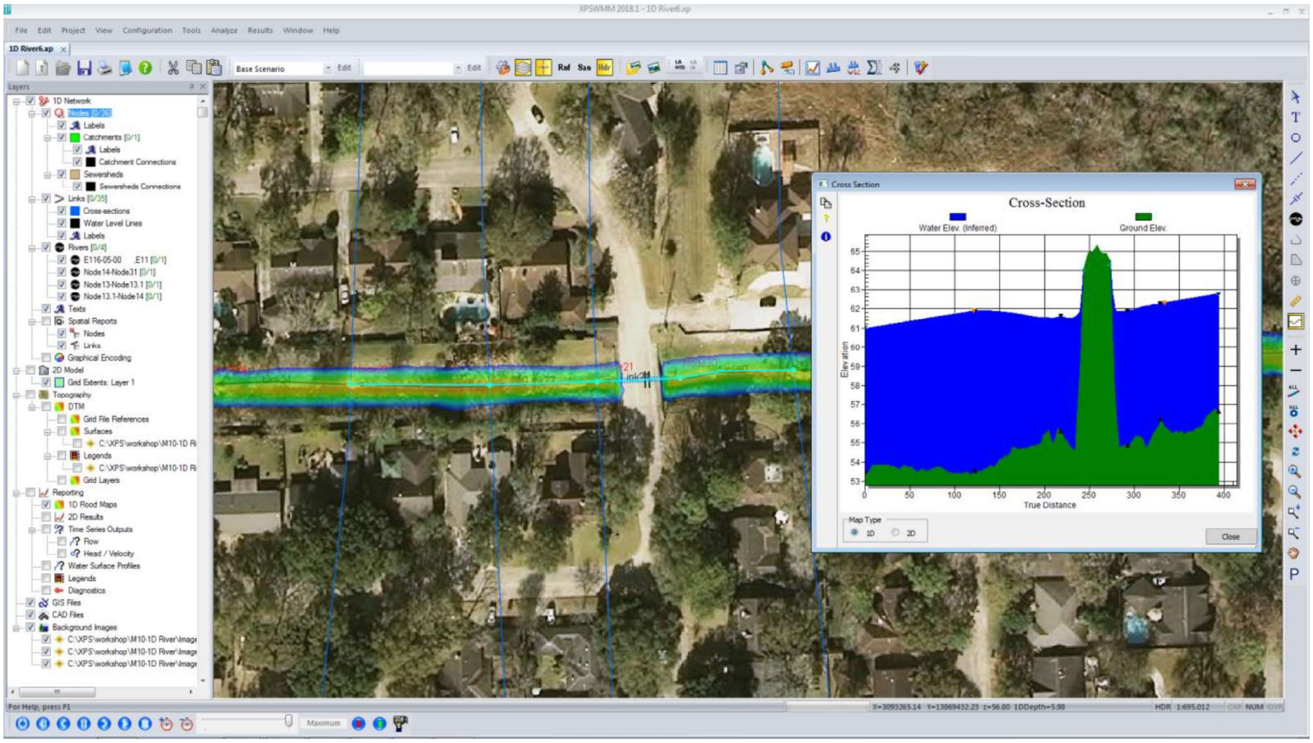
5. **Edit** the **Bridge/Culvert Data**, view the **reach length**, **loss** and **roughness** values as shown below imported from the HECRAS model. We also see and could edit piers in this dialog. Click **OK** two times. Uncheck **User Specified Length**. Note that the new length of bridge is 100 ft.



6. **Save** the model, then select **Solve**. After the model solves select from **Node34 to Node30** and select **Dynamic Section Views**.



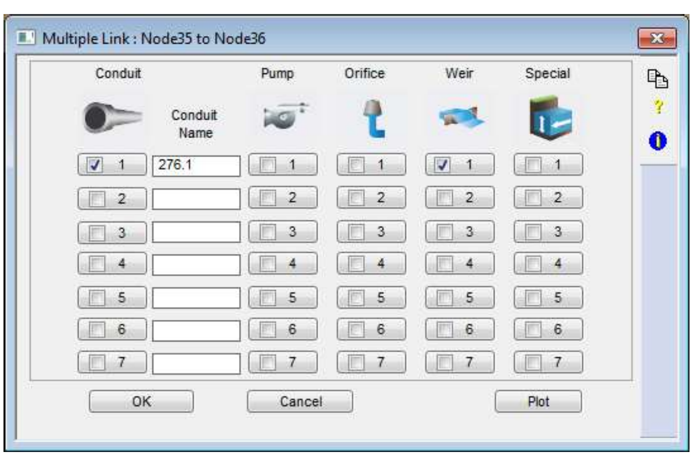
7. **Check** the **1D Flood Maps** layer and allow the map generator to interpolate the 1D results from the xptin surface. Create a Section Profile Line from a few links upstream to downstream through the bridge.



Lateral Reach Crossing – Bridge Deck

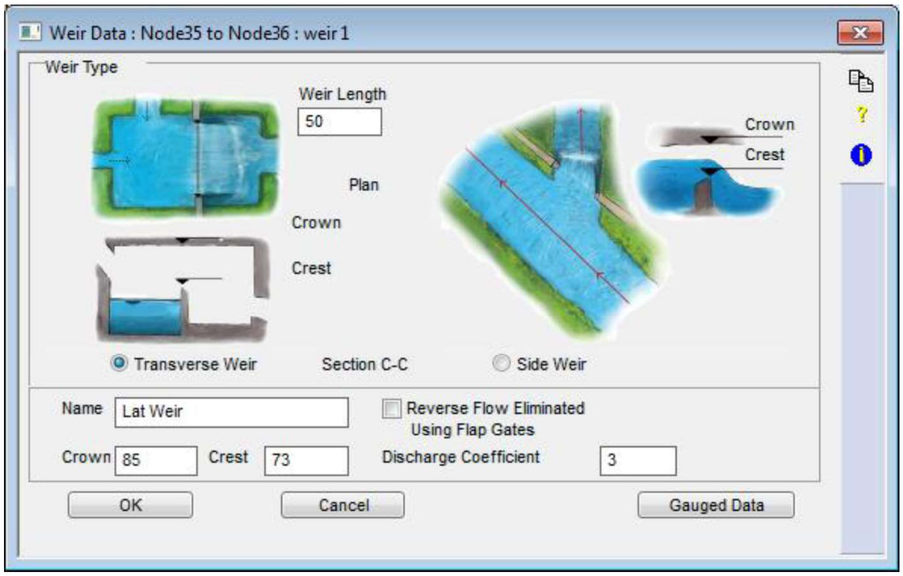
1. Within the Lateral Reach, the bridge crossing is currently only represented as 4 box culverts, with the flow over the roadway deck ignored. This may be artificially raising the Lateral Reach WSL and should be corrected so large flows can be accommodated that would overtop.

2. **Right-click** on **Link34** and select **Multilink**, then a **Weir** check box.

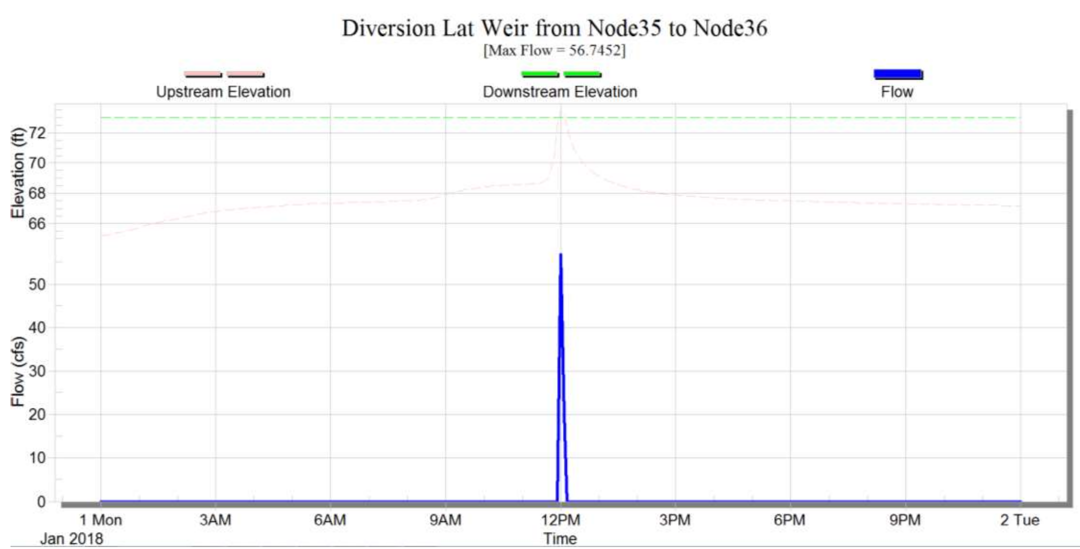


3. **Name** the Weir **Latweir**, input **50 ft** for the **Weir Length** of the **Transverse Weir** and **3** for the **Discharge Coefficient**.

4. **Set Weir Crest** level to **73.0** and the **Crown** to **85**. This will represent the flow over the top of the bridge deck (at elevation 73.0) and the Crown is set arbitrarily high at 85 as flow over the roadway deck will never become surcharged.

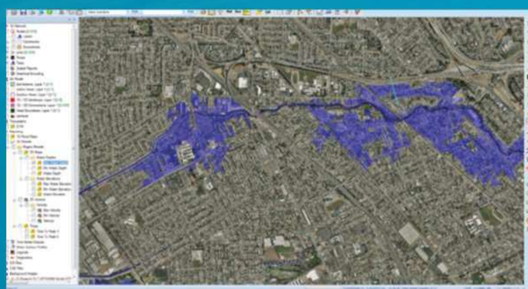


5. Click **OK** twice and **Save** the model. After **Solving** the model review the Lateral Reach crossing at **Link34** using **Review Results**, use the diversion icon  to assess the overtopping flow through the weir.



Integrated 1D/2D Modeling

Modules 12-14



Innovyze®
Empowering water experts

1

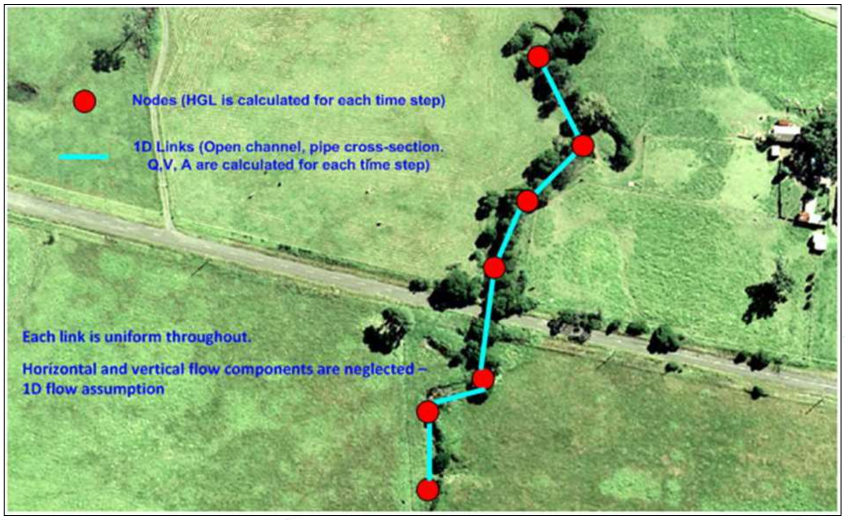
Workshop 12-14 Objectives

- Learn theory of 1D/2D integrated modeling
- Model river and culverts as integrated 1D/2D
- Model urban flooding - nodes connected to 2D
- Simulate sudden levee/dam break flooding
- Prepare flood inundation maps
- Interpret 2D flooding results

Innovyze®

2

1D Modeling Tools



Innovyze®

3

1D Flood Modeling Tools

1-Dimensional Models – Advantages:

- Easy and fast to build using cross-sections
- Small amount of input data and a low hardware requirement
- Very large areas can be modeled
- Less complicated theory and easy to troubleshoot
- Plenty of supporting literature and tools available
- Many experienced modelers, it is comparatively easy to find these skills

Innovyze®

4

1D Flood Modeling Tools

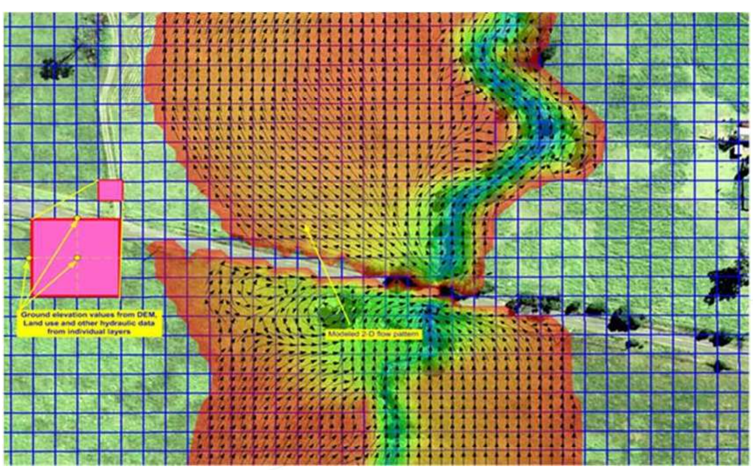
1-Dimensional Models – Disadvantages:

- Cannot model shallow overland flows correctly - pattern is essentially **2D**
- Tends to overestimate the water depth and velocity due to a 1D assumption
- Usually excludes the storage effects in channels such as floodplain storage during high flows events
- Cutting cross-sections can be a tedious trial and error procedure as we can't predict the exact direction and extent of flow before modeling
- Fine scale modeling difficult due to instability
- Requires a lot of engineering judgment and is therefore subject to the skill of the modeler
- Needs a thorough calibration to make the results reliable

Innovyze®

5

2D Flood Modeling Tools



Grid or Mesh covers the terrain

Innovyze®

6

2D Flood Modeling Tools

2-Dimensional Models – Advantages

- Flow paths do not need to be pre-determined by the modeler
- Easy to build overland 2-dimensional flood flow models from surface data (Urban flood flow patterns are 2 dimensional)
- Flow paths can change based on water level and the complex topography
- Suitable for fine scale modeling
- Automatically accounts flood flow storage effects (flood fringe, floodway, etc.)
- Losses are automatically accounted for bends, constrictions & expansions
- Suitable for 2D structures such as bridges, culverts and obstructions
- Directly create impressive flood inundation maps
- Less engineering judgment required

Innovyze®

7

2D Flood Modeling Tools

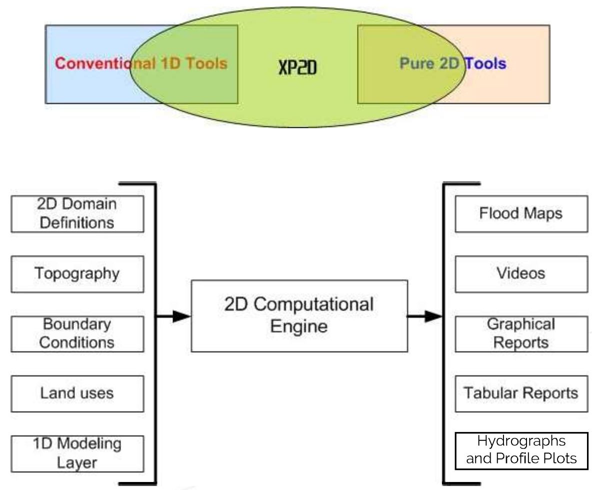
2-Dimensional Models – Disadvantages

- Precise and dense survey data is desired
- More computational power and time required (GPU helping)
- High storage requirement as the data sets are large for long durations and large cell counts
- Difficult to model very large areas with small cells
- Familiarity means it may be more difficult to troubleshoot than a 1D model
- Fewer experienced 2D models – small user community
- Not all regulators will accept 2D models (changing)

Innovyze®

8

XP2D: Integrated 1D/2D Modeling



Innovyze®

9

2D Computational Engine

- XP2D is powered by Tuflow
- Shallow Water Equations in 2D Plane
  - Continuity and momentum equations solved in X,Y Plane
  - Hydrostatic pressure distribution assumed
  - Equations are depth averaged
  - Depth of flow is negligible compared to the wavelength

Innovyze®

10

2D Shallow Water Equations

$$\frac{\partial \zeta}{\partial t}+\frac{\partial (Hu)}{\partial x}+\frac{\partial (Hv)}{\partial y}=0 \qquad \text{(2D Continuity)}$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}-c_f v+g\frac{\partial \zeta}{\partial x}+g\,u\left(\frac{n^2}{H^{4/3}}+\frac{f_l}{2g\partial x}\right)\sqrt{u^2+v^2}-\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)+\frac{1}{\rho}\frac{\partial p}{\partial x}=F_x$$

(X Momentum)

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+c_f u+g\frac{\partial \zeta}{\partial y}+g\,v\left(\frac{n^2}{H^{4/3}}+\frac{f_l}{2g\partial y}\right)\sqrt{u^2+v^2}-\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)+\frac{1}{\rho}\frac{\partial p}{\partial y}=F_y$$

(Y Momentum)

where

ζ = *Water surface elevation*

u *and* v = *Depth averaged velocity components in X and Y directions*

H = *Depth of water*

t = *Time*

x *and* y = *Distance in X and Y directions*

c_f = *Coriolis force coefficient*

n = *Manning's n*

f_l = *Form (Energy) Loss coefficient*

μ = *Horizontal diffusion of momentum coefficient*

p = *Atmospheric pressure*

ρ = *Density of water*

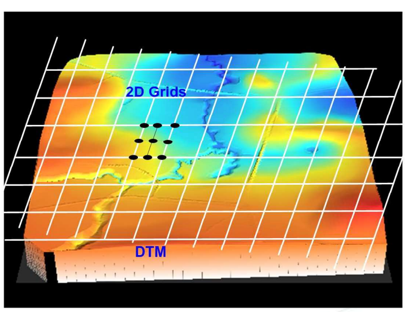
F_x *and* F_y = *Sum of components of external forces (eg. wind) in X and Y directions*

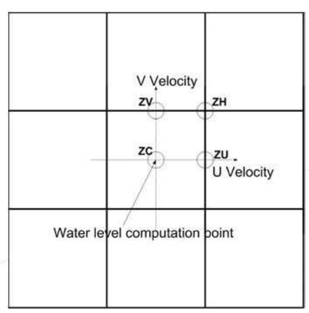
Innovyze®

11

2D Grid Topography

- Topography defined by Z point interrogation of the DTM (TIN)
- 2D model is described as a grid
- C, U, V, and H elevations are interrogated (ZC,ZU,ZV, and ZH)

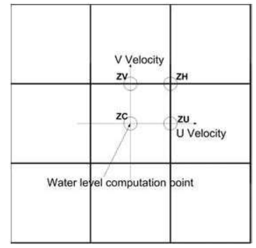




Innovyze®

12

2D Elevation "Z"-Points



ZC Point:

- Defines volume of active water
- Volume = cell area * cell water depth
- Controls when a cell becomes wet and dry

ZU and ZV points:

- Control how water is conveyed from one cell to another
- Are where the momentum equation terms are centered
- Are deactivated if the cell has dried (based on the ZC point) and cannot flow

ZH points:

- Play no role hydraulically
- Are, by default, the only elevations to be written to the SMS .2dm mesh file

Innovyze®

13

2D Domain Definition

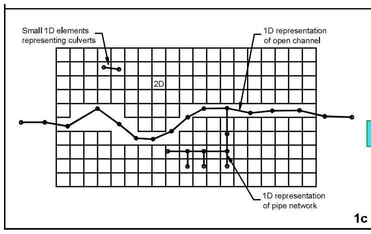
2D Domain can contain:

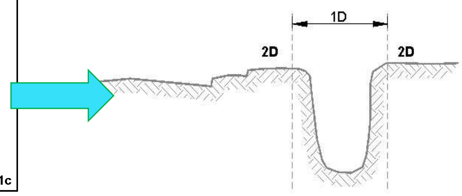
- Active and Inactive area (buildings, areas of 1D computation)
- Head and Flow Boundaries as polylines
- Landuse Polygons to prescribe roughness and infiltration
- Flow and Rainfall Polygons
- Ridge and Gulley Breaklines
- Fill Area polygons (cut or fill)
- Static and Dynamic Elevation Shapes
- Connections to 1D

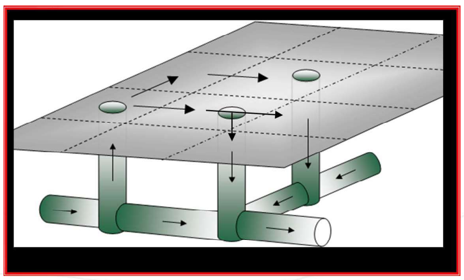
Innovyze®

14

Typical 1D/2D Linkages





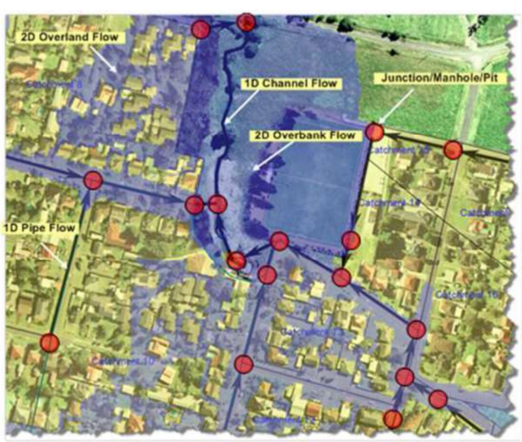


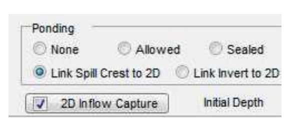
- Flooding at storm manholes and inlets
- Flooding due to lack of network capacity
- 1D closed system and 2D overland flow system
- 1D channel flow and 2D floodplain flow

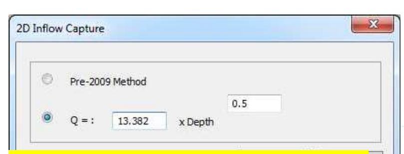
Innovyze®

15

Node: 1D/2D Linkage







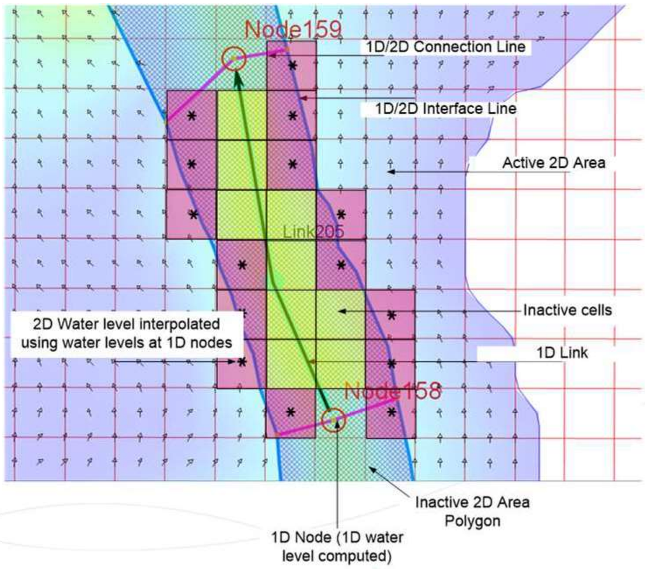
Default rating curve for overland flow capture to the node. This can be modified if the user has another one.

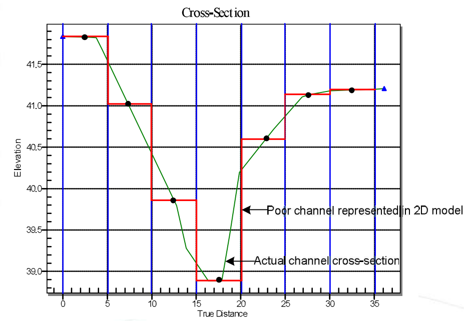
The outflow from node to 2D domain is computed by the mass balance at the node.

Innovyze®

16

Channel: 1D/2D Linkage

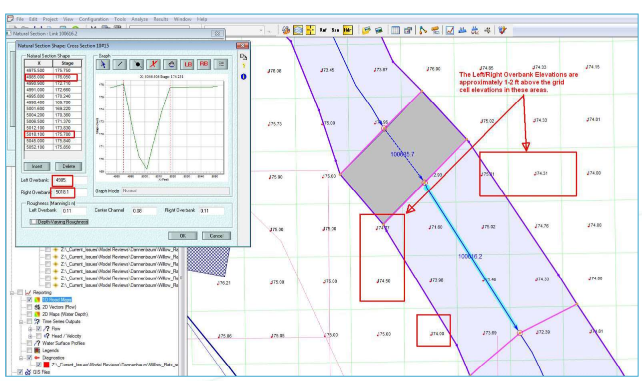




17

Typical Model Setup Errors

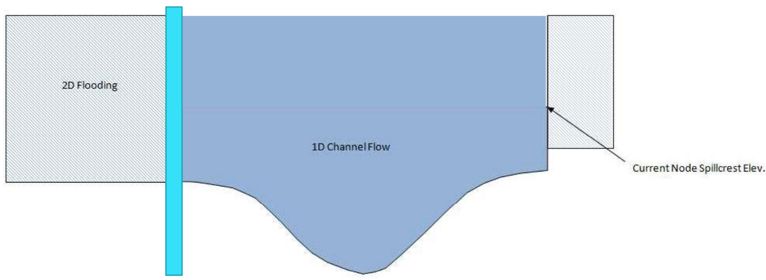
- Locate the 1D/2D Interface along the top of bank
- Width of 1D and width of inactive cells should be equal
- Elevation in 1D does not match well with the grid



18

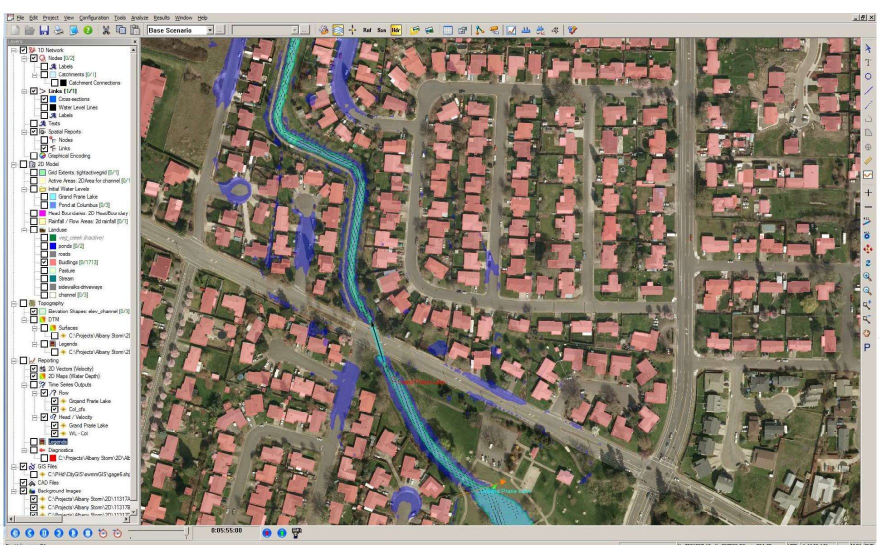
Typical Model Errors

- 1D Nodes are not high enough to allow 1D flow to rise to a flood level (creates pressure flow in the river)
- Need VERT_WALLS=ON and nodes higher than expected flood level
- Raise nodes to at least the expected flood elevation



19

Suitable for Urban Areas



20

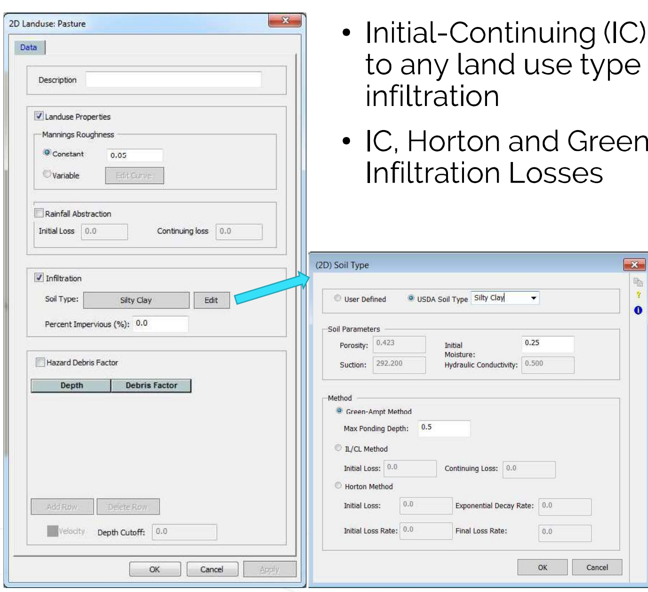
2D Direct Rainfall on Grid

- Rainfall applied equally to grid cells within a polygon
- Initial & continuing losses, Green Amp and Horton
- Rainfall applied to polygonal areas (1 or multiple polygons)

Innovyze®

21

2D Infiltration



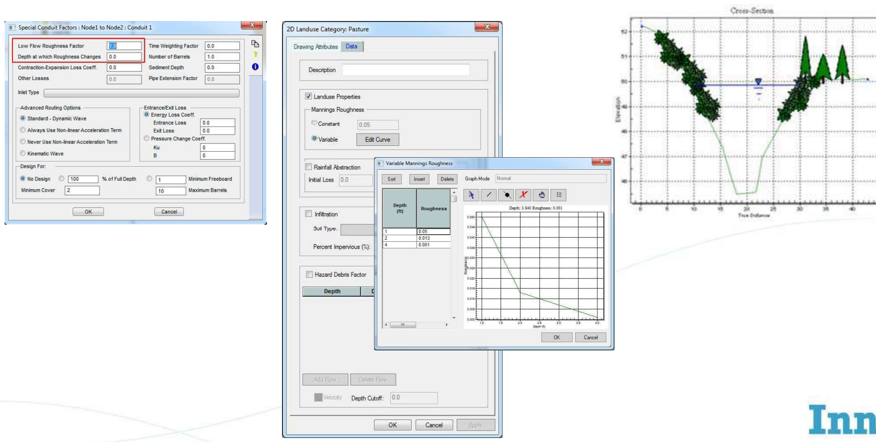
- Initial-Continuing (IC) loss model can be applied to any land use type as rainfall abstraction or infiltration
- IC, Horton and Green-Ampt can be applied for Infiltration Losses

Innovyze®

22

Depth Dependent Roughness

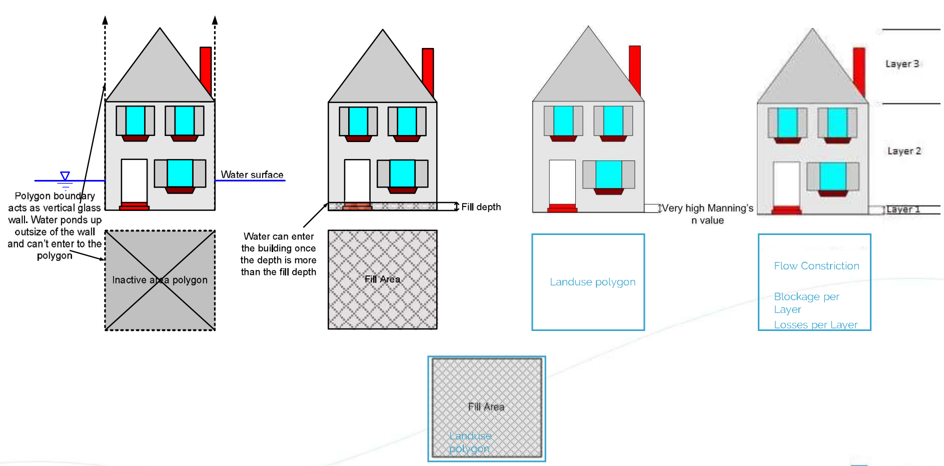
- Roughness is classic calibration parameter
  - 1D vertical changing roughness in Conduit Factors
  - 2D vertical changing roughness based on Landuse polygons



Innovyze®

23

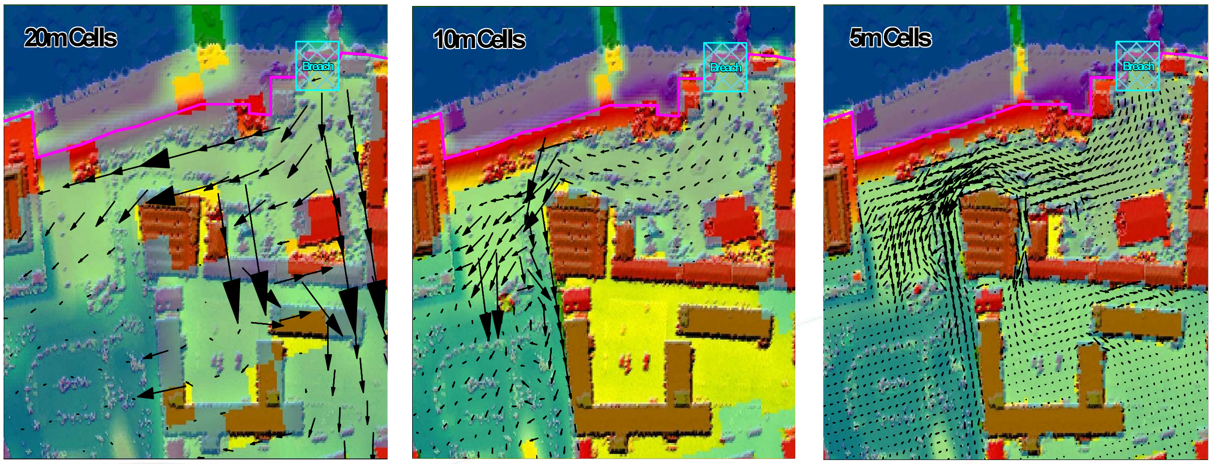
Simulation of Buildings in 2D



Innovyze®

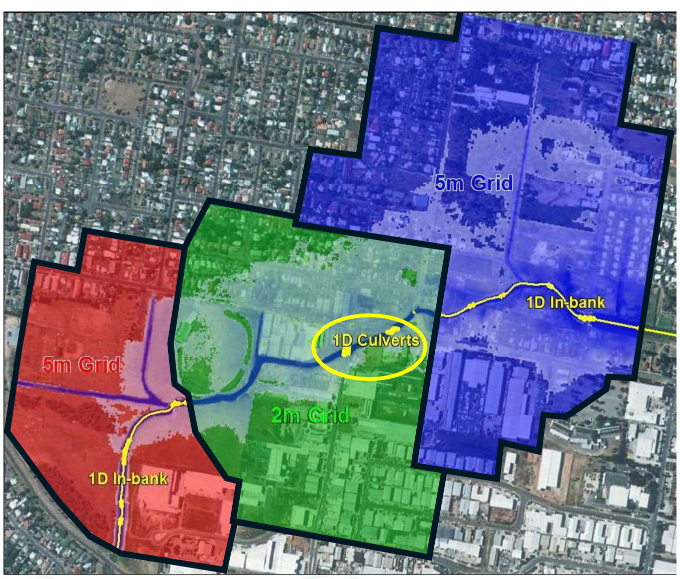
24

2D Grid Size: Implications



Innovyze®

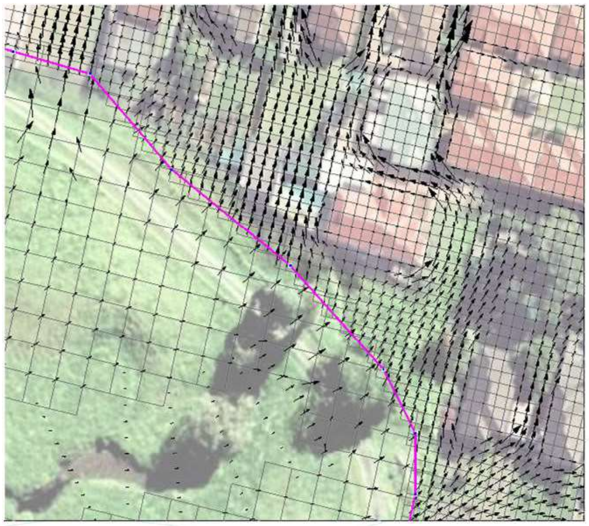
Multiple 2D Domains



- Fine grid over area of interest
- Coarse grid(s) elsewhere
- Use 1D where grid resolution too coarse

Innovyze®

2D/2D Interface – How It Works?



- Multiple Domains of different size
- 2D/2D Interface
- Vertices on 2D/2D become 1D nodes for flow exchange

Innovyze®

27

Workshop

Innovyze®

28

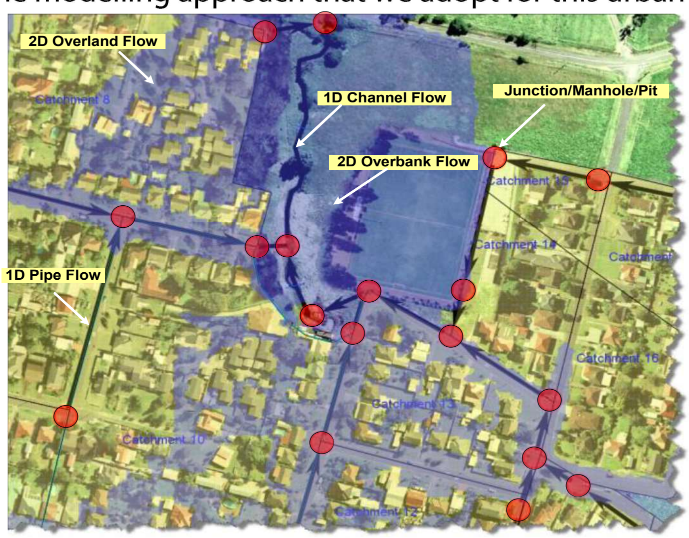
Innovyze®

Empowering water experts

29

1D/2D Urban Flood Modeling

This tutorial describes how to set up an integrated 1D/2D urban flood model for a post-development scenario. The study area is the same as that of the previous pre-development river flood model except the area is urbanized. Now the area has an urban development and associated pits and pipes for the stormwater drainage. The flood pattern for this scenario will be different from the river flooding. The flooding will be a combination of flood due to the overflow of drainage pits and river flood due to the overbank flow. Generally this happens in an urban area during higher return period storms. The user will set up a flood model using the xp tools and perform an analysis for the same 100 year return period storm event that was used in the river flood module. Note that the flow from the urban catchments will be simulated using the Runoff Mode and these hydrographs will be transferred to the Hydraulics Mode for Dynamic Wave Routing. The below diagram shows the modelling approach that we adopt for this urban flood model:



Objectives

Users will learn how to utilize xp's tools to create 1D/2D urban flood model. You will learn how to:

1. Integrate the 1D hydrology, 1D hydraulic, and 2D hydraulic models
2. Model buildings in the floodplain
3. Prepare flood hazard maps

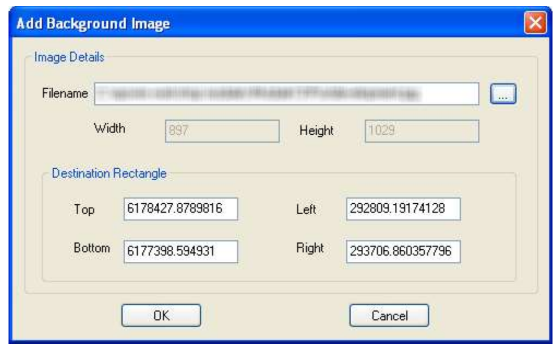
Data files: 1D2D_Urban_starter_model.xp 1D-2D_Urban_Flood_m.xp
1D2D_Urban_completed_buildings.xp 1D-2D_Urban_completed.xp
Model Data.xls
Postdevelopment.xptin
Postdevelopment.jpg, jpw
Ponds_Postdevelopment.shp Roads_Postdevelopment.shp
Creek_Postdevelopment.shp Buildings_Postdevelopment.shp
1D_2D_Interface.shp
2D_Active_Area.shp
2D_Inactive_Area.shp
Creek & Culverts.XPX

Adding background image and DTM

First, we will setup the background image and DTM using the xp interface as we did for the previous module. **Note:** for advanced and experienced users proceed to section **Using the Starter Model** on page 13-10 and load the **1D2D_Urban_starter_model.xp**.

1. From the **File Menu**, create a **New Blank Job** and name the file as **1D-2D_Urban_Flood1_m.xp** and save the file. Select the **Unit** as **Metric**.

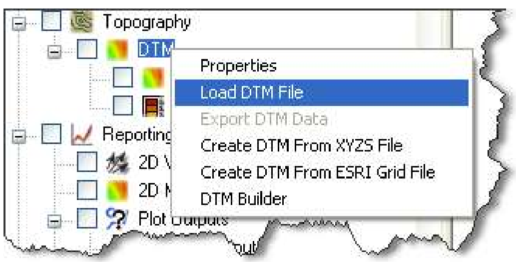
2. Now tick on the **Background Image Layer**, highlight with the left click. Now right click on the layer and select the **Add Background Image** from the menu. Click on the ellipsis, browse for the file **Postdevelopment.jpg**, select and **Open** it.

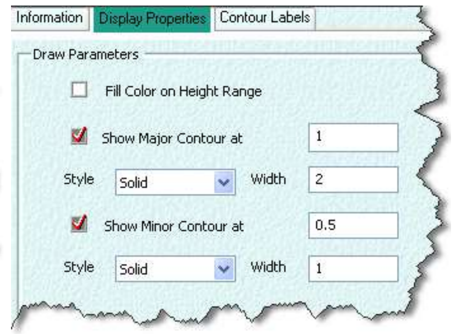


Click on **OK** to see the background image using the georeferenced coordinates.

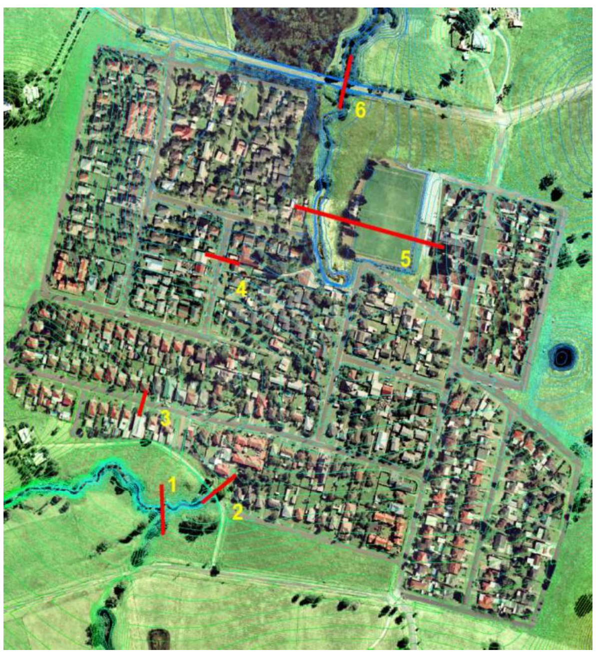
You will see the area is developed with many buildings and a sports field. The creek is partially covered and the water is conveyed through underground pipes, which are connected to the urban drainage network. For this post development site the hydrology will be essentially urban. The primary objectives of the flood modeling of such a site are: Mapping flood inundation due to the urban flooding and checking the safety of the houses and roads during flood events by creating hazard maps.

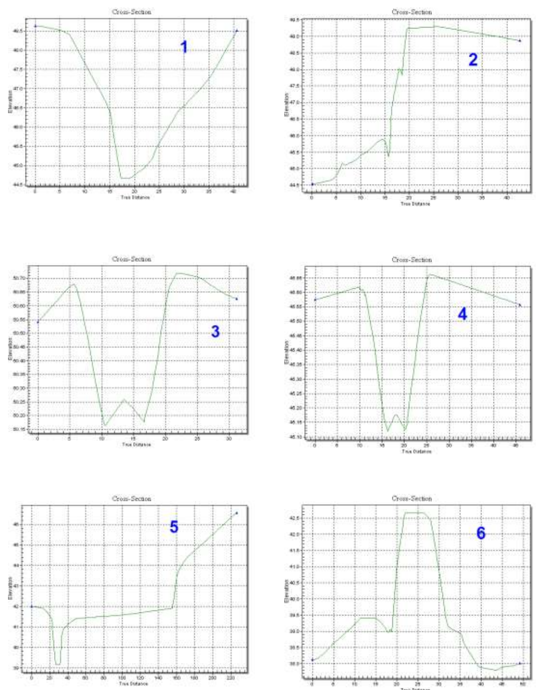
3. Next we will load DTM to the model. Load the DTM file **Postdevelopment.xptin**. Now display the **Major Contour at 1m** interval and **Minor Countours** at **0.5 m** interval. Adjust the **Transparency** to see the aerial image and DTM together.



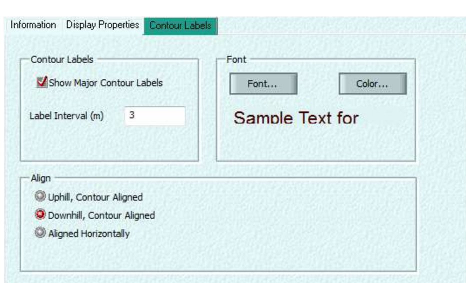


Get a few cross-sections and examine the terrain.





You can access the **Contour Labels Tab** and switch on the **Show Contours Labels** tab.

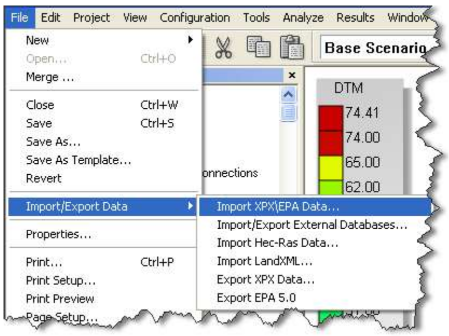


Now **Save** the file.

1D Domain Definition

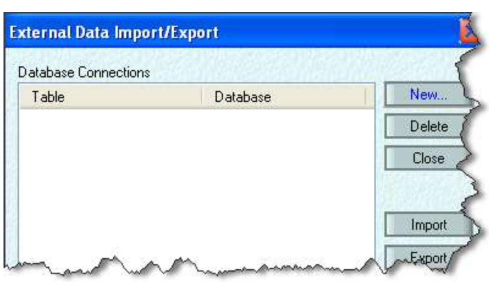
The next step is to define the 1D domain. We have pits, pipes, open channels, and culverts in the 1D network. We will get the 1D open channels and culverts through xpx import and the rest of the network through an external database linkage.

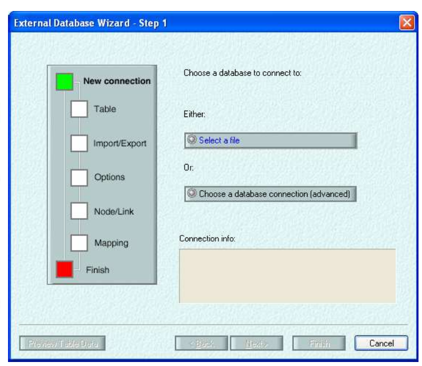
1. From the **File Menu**, select **Import/Export Data**, then **Import XPX/EPA Data**. Browse for the file **Creek & Culverts.XPX**, select, **Open** it and **Import**. Click on **OK** and ignore the generic warning message that reminds that this process overwrites existing data.



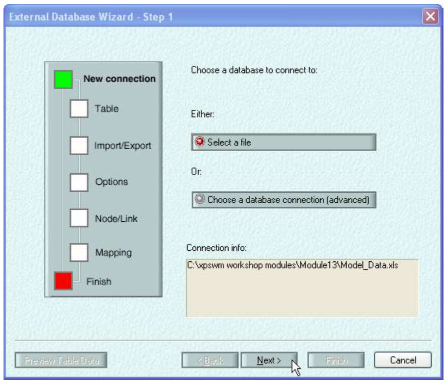
You can now see the natural creek and the culverts. The rainfall and infiltration data used in the hydrologic calculations has also been imported in this file as well as 2D Landuse Categories. **Open** the **Global Database** from the Configuration menu to see these items.

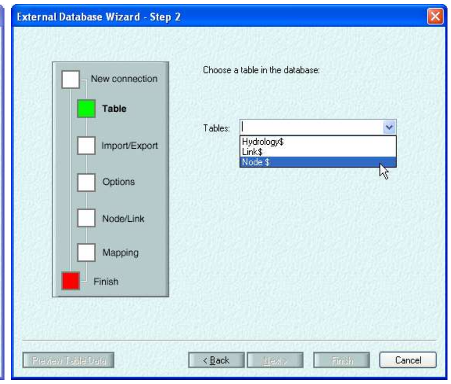
2. Now we will import the pits and pipe network from the external database. From the **File Menu** select **Import/Export Data** and **Import/Export External Databases** option. Click on **New**, and **Select a File** button. Browse for the file **Model_Data.xls**, select and **Open**.



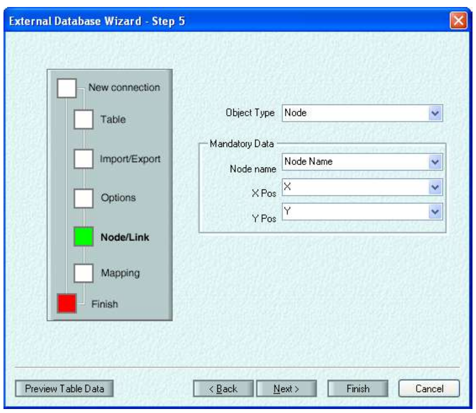


Now click on the **Next** button, and select the **Node** table from the **Tables** dropdown menu.

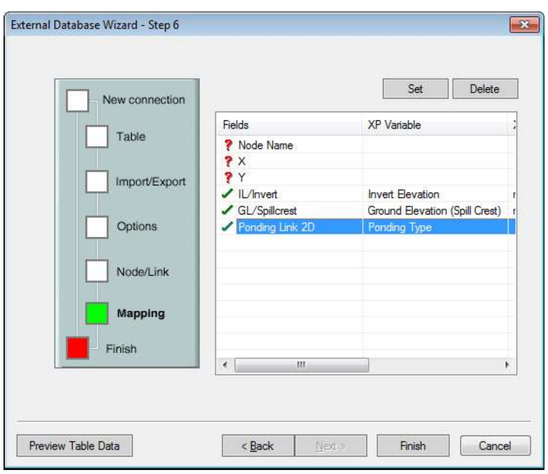


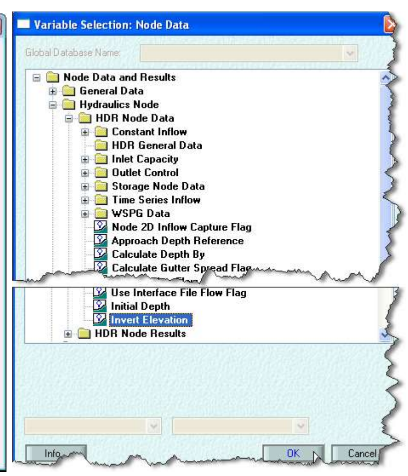


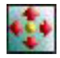
Again click on the **Next** button and select **Import data only** radio button. Again click on **Next**, and select the **Create New or Update Existing** option from the **On Import Object Creation** dropdown menu. Click on **Next** button again. In the step 5, select the **Object Type** as **Node** from the dropdown menu. Map the **Mandatory Data** as shown below:

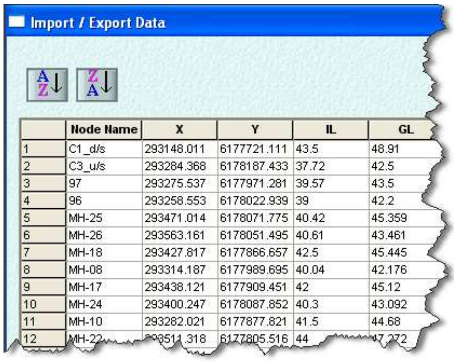


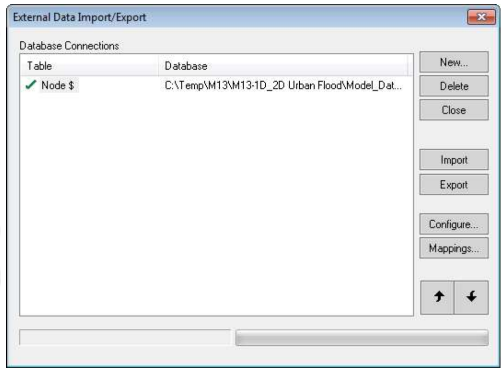
Click on **Next** button again, select the Field **IL** and click on the **Set** button. You will see the **Node Variable Selection** dialog now. Expand the **Node Data and Results** and go to **Hydraulic Node**, select **Invert Elevation** and click on **OK**.



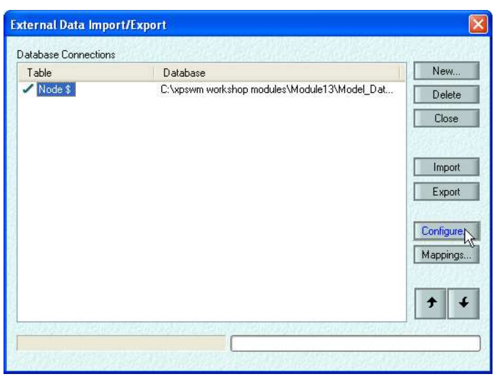


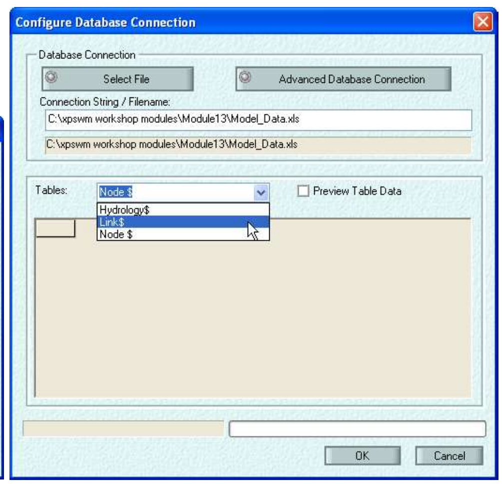
Similarly Map the field **GL/Spillcrest** to **Ground Elevation (Spill Crest)** and the **Ponding Link 2D** to the **Ponding Type**. Now click on the **Finish** button and **Import** in the next dialog. You will see all the data ready to be imported, click on **OK** two times and **Close**. When you click on the  button, you will see all 26 nodes imported with 78 variables read.



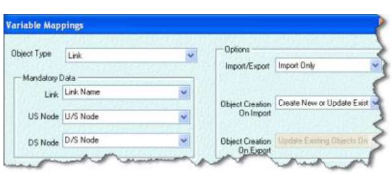


3. We will repeat the procedure for link import by modifying our existing connection. Select **File Menu/ Import or Export Data/ Import or External Database**. Highlight the **Node $** and click on the **Configure** button. Select the **Link $** and click on **OK**.





Click on the **Mapping** button and **Map** the **Mandatory Data** as follows:



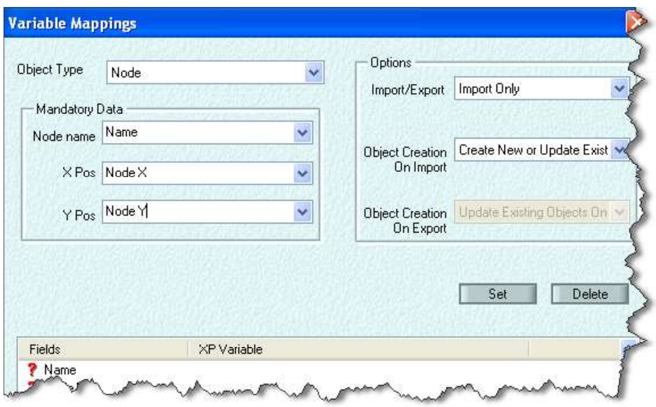
Excel Source Field	XPSWMM Target Field
Length	Conduit Data->Length
Shape	Shape
U/S Invert	Upstream invert elevation
D/S Invert	Downstream invert elevation
Diameter	Diameter (Height)
Conduit Factors Flag	Conduit Factors Flag
Number of Barrels	Conduit Factors->Number of Barrels

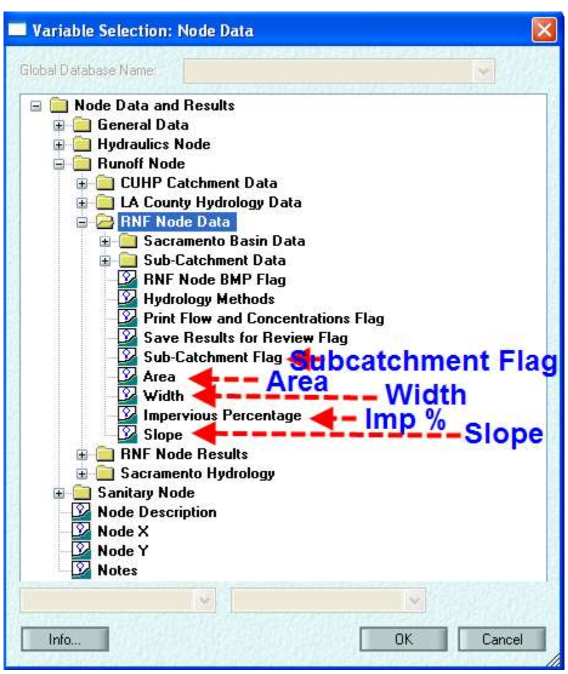
Finally click on **OK** and **Import** as we did for the nodes. You should see a message of 27 links created and 189 variables read.

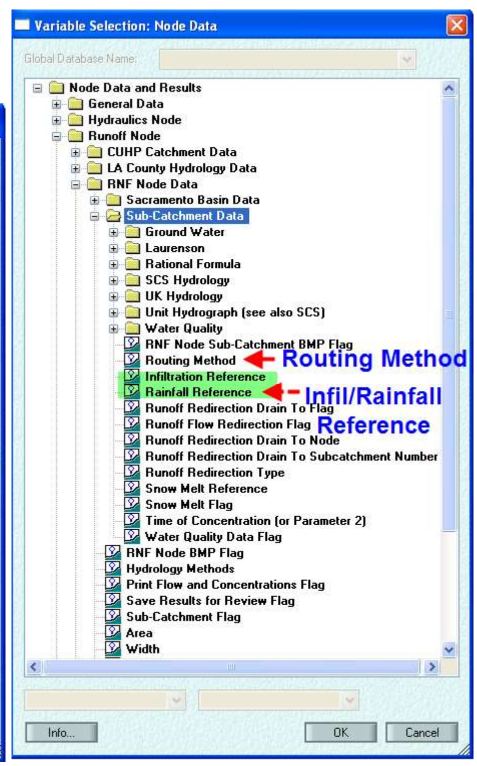
Now select the **Runoff Mode** by clicking on the 

button. Select all the nodes except the culvert and open channel nodes and make them active n the **Rnf Mode** by pressing the **+** button.

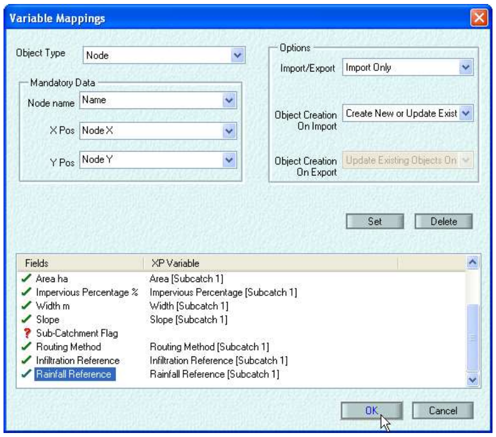
Repeat the **External Database Import** procedure for **Hydrology$** as well. Map the fields as shown below:

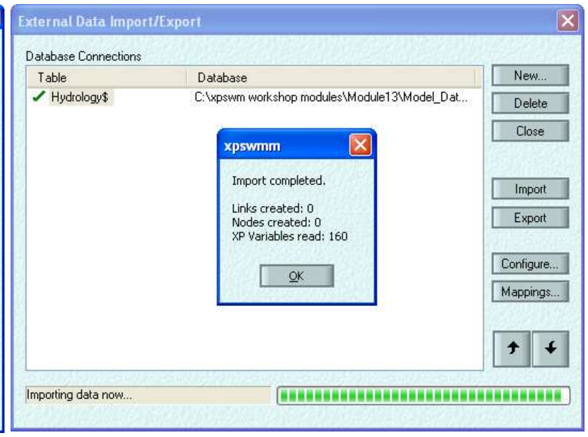




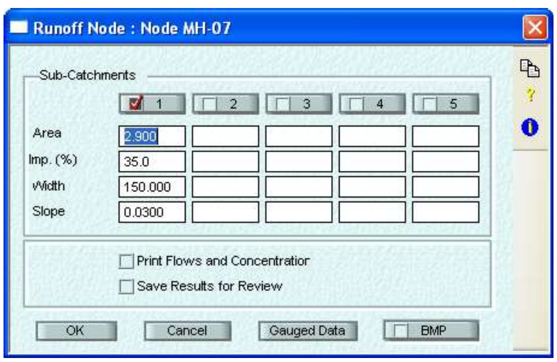


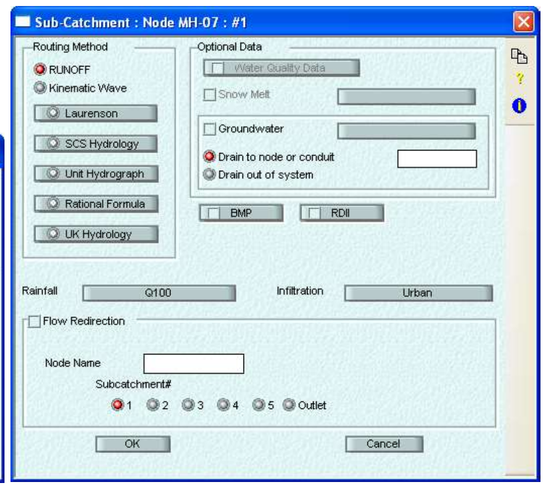
Click on **OK** and **Import**.





You can open the nodes and check the hydrology data imported including the rainfall and infiltration records.



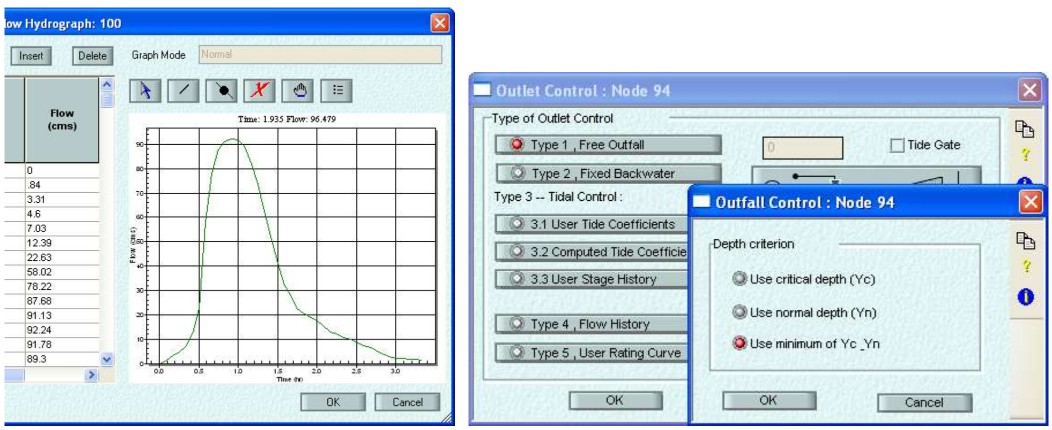


Using the Starter Model

All steps to this point have been completed in the starter model called **1D2D_Urban_starter_model.xp**

4. Now **Add New Layers** for **Inactive Areas**, **1D/2D Interfaces**, and **1D/2D Connections**. Import the **2D_Inactive_Area.shp** to **Inactive Areas Layer** and **1D_2D_Interface.shp** to **1D/2D Interface Layer**. Highlight the **1D/2D Connections Layer**, **Snap** and digitize **1D/2D Connections Lines** or import from the corresponding shape file from all the nodes to 1D/2D Interface Lines. Repeat this for all the nodes of channels and culverts.

5. Double-click and open node **100** and **94**. You will see the inflow hydrograph from the upstream area and the 1D outfall definition.



6. Right click and **Generate Water Level Lines** for all the open channel links. Trim the water level lines to the 1D/2D Interface Lines. Please refer to the previous module for details on the steps to generate and trim WLLs.

7. Now **Save** the file as **1D-2D_Urban_Flood2_m.xp**.

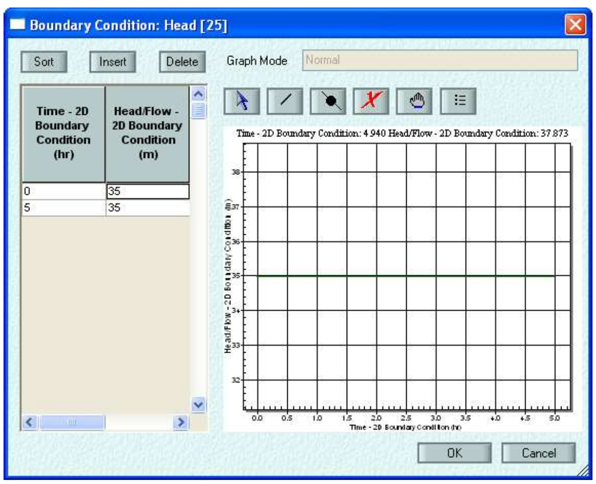
2D Domain Definition

We will define the 2D modeling domain in the same manner as the previous module.

1. Digitize the **2D Grid Extent** to the edges of the aerial image. Make the grid size **3m**. Typical sizes would range from 2m-5m. Click on **OK**. You may switch off the grid extents layer for better visibility of the model.

2. **Add New Layer** for **2D Active Areas** and create the 2D Active Area by importing the **2D_Active_Area.shp**.

   Now add a new layer for the **Head Boundaries** and digitize and snap a polyline to the 2D active area. Right click and **Edit** the **Head Boundary Polyline** and enter the data as follows to ensure a free outflow from the 2D domain.

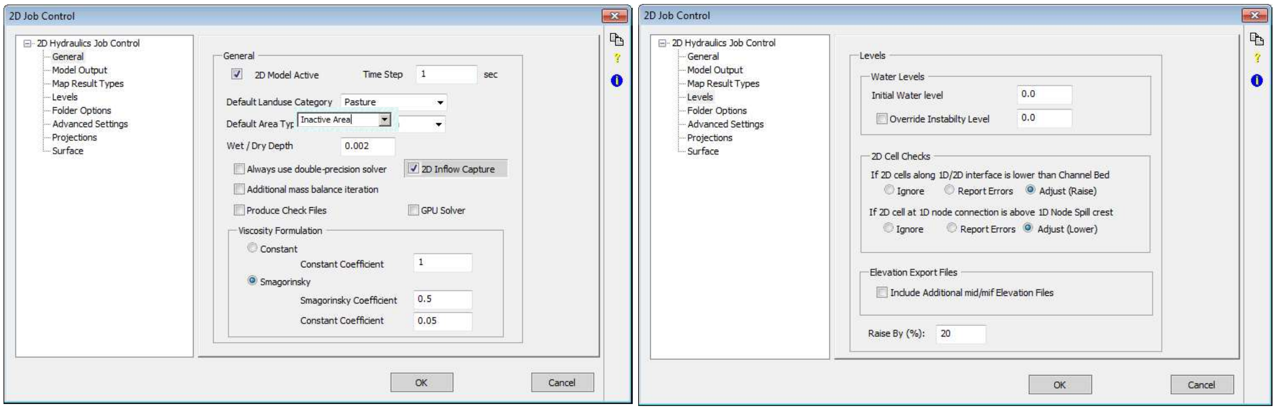


3. Next step is to define 2D land uses. Right click on each layer and import the land use polygons as we did in the previous model. Make sure that the priorities are as shown:

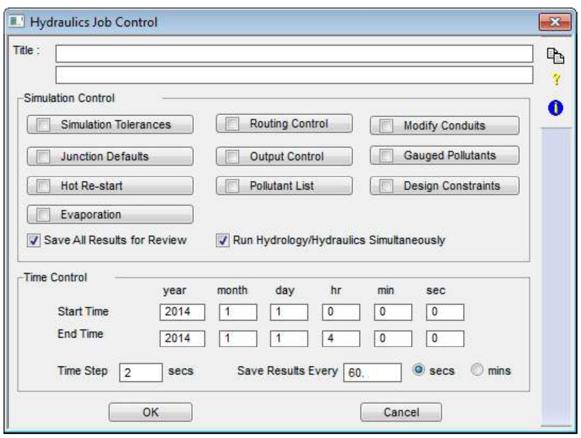
2D Landuse Layer	GIS File
Roads	Roads_Postdevelopment.shp
Vegetated Creek	Creek_Postdevelopment.shp
Ponds	Ponds_Postdevelopment.shp
Buildings	Buildings_Postdevelopment.sh

Landuse	n Value	Priority
Pasture	0.06	1
Ponds	0.025	2
Vegetated Creek	0.09	3
Roads	0.025	4
Buildings	0.3	5

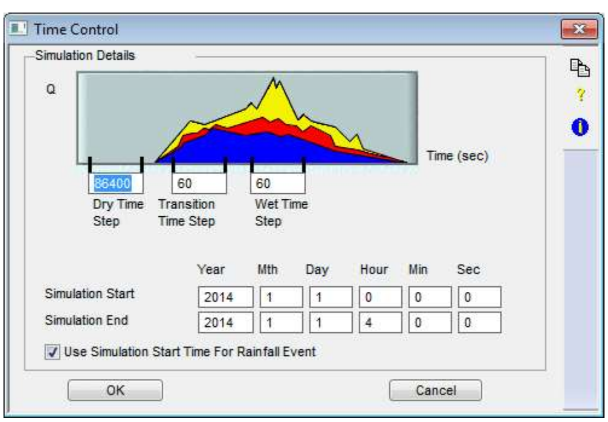
4. Go to **Configuration Menu/ Job Control/ 2D Model Settings** and complete the settings as shown:



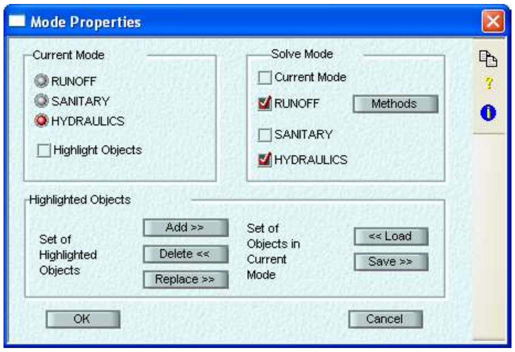
Under **Configuration/Job Control/Hydraulics**, complete the settings as shown:



Under **Configuration/Job Control/Runoff/Time Control** complete the data entry as follows:



Now, go to **Configuration/ Mode Properties** and tick on both the **Runoff** and **Hydraulics** tick boxes.

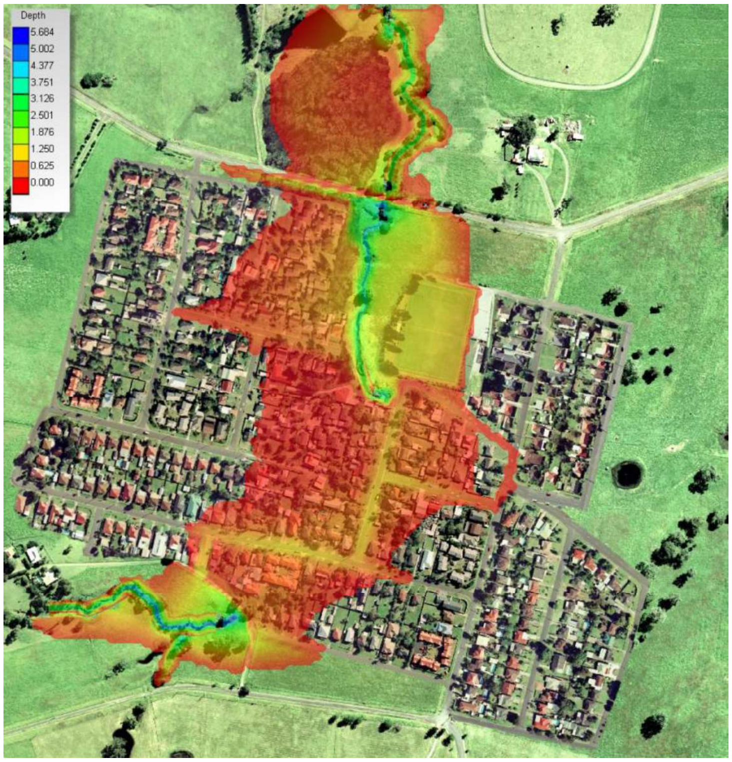


Finally, add few **Plot Output Points** as shown in the map near the **Culvert1**:

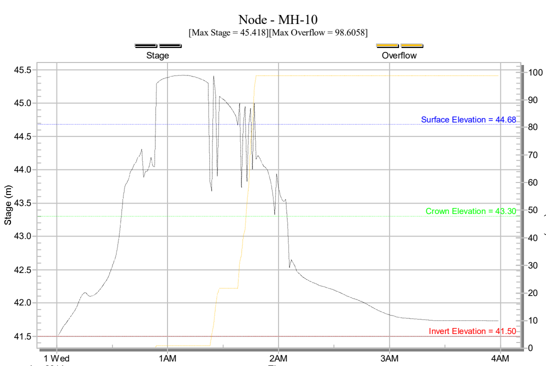
Note: The model **1D-2D_Urban_completed.xp** could be loaded as a completed model with all of the previous steps.

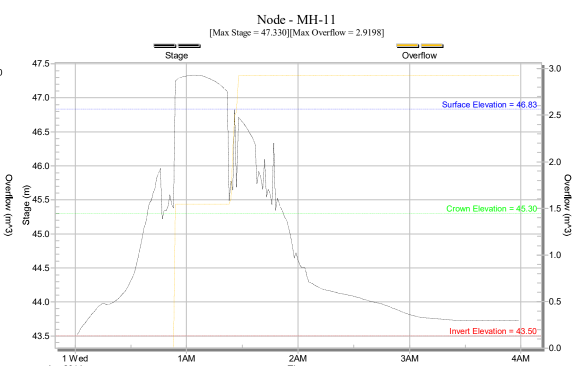
5. Now **Solve** the model. Tick on the **2D Maps (Water Depth) Layer** and you will be able to see the flood inundations.

   Note: If the model was not built from the starter model then a warning message will appear for all active Runoff nodes without subcatchment data. Ignore the warning or make all of the nodes without catchment data inactive in the Runoff mode.



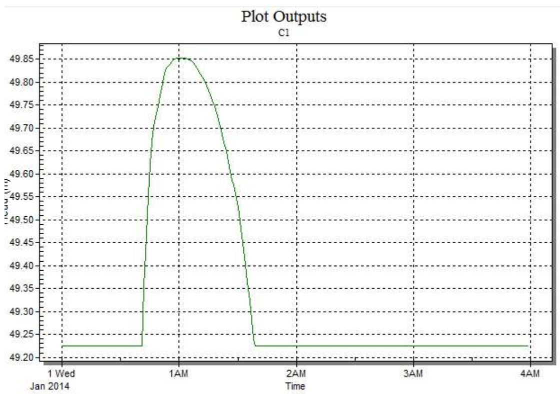
You will see that the maximum inundation depth is **5.7 m** which happens in the deep channel portion. You will see that many roads are acting like flood flow conveyance channels. Review results for **MH-10** and **11**, you will see that these pits are overflowed and the water is conveyed through the roadway. This is the same as that of the dual drainage analysis in 1D hydraulics.

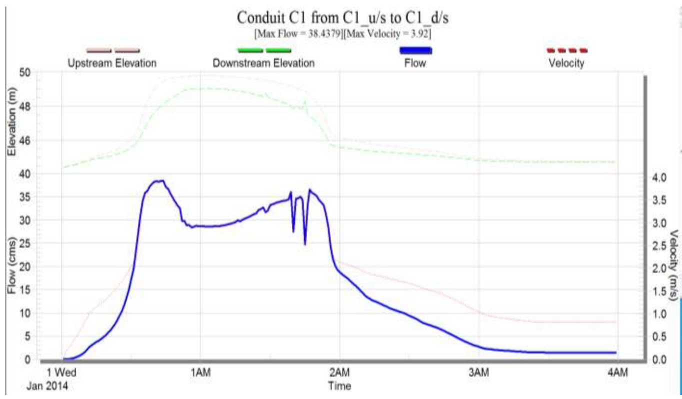




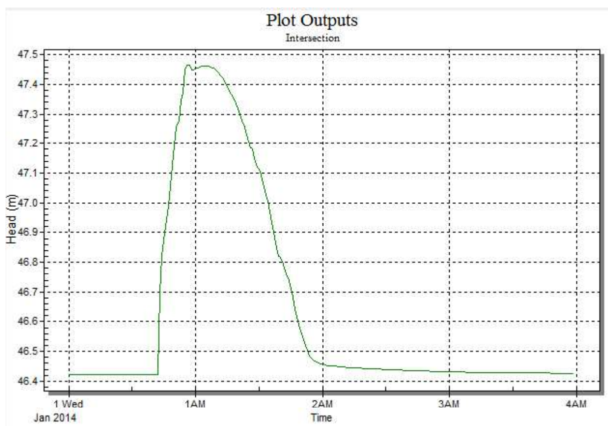
You may notice that the sports field acts like a flood storage area as we predicted early.

Zoom-in to the Culvert 1 area and check the water level output point C1. You will see that the Culvert 1 road is overtopped from about **40 min** to almost **1 hr 40 min**. Click on the Culvert 1 and review result and you will see a dip in the flow through the culvert. This is due to the road overtopping flow.





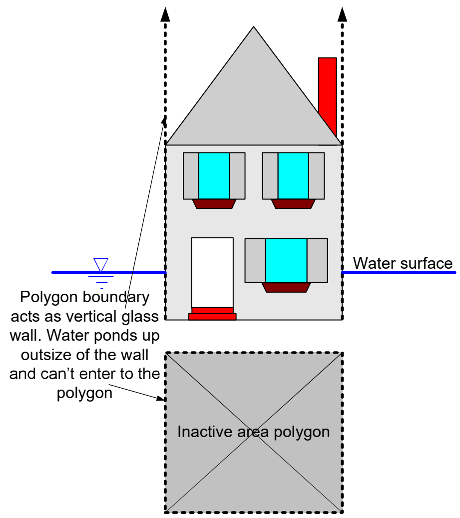
If you review the water level point **Intersection**, you will see the flow reaches at **38 min**. This is due to the road overflow from the further upstream side.



Switch on the maximum velocity vectors; you will notice that water depth is shown at the building locations also. Check the flow vectors and you will see flow is going through the buildings but higher velocity around the buildings. But the buildings can block the flow up to certain depths. Zoom-in and get a cross-section for any building location and you will see that the buildings are not present in the DTM.

We assigned a very high **Manning's n** value (**3.0**) to overcome this. There are other techniques also to define buildings in the flood plain:

Making the buildings' polygon inactive:

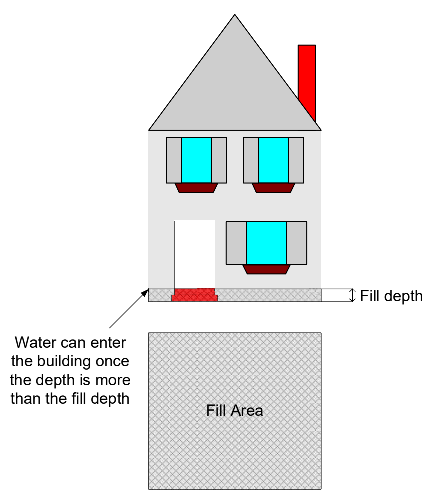


When you make the buildings' areas inactive, the polygon boundary acts as a vertical glass wall so that flood water can't enter the buildings. This might not always be a good modeling approach. The assumption of vertical water proof walls is not valid if the flow exceeds some threshold depth. When the water depth exceeds the sill-level of doors and windows, water can enter the buildings through these openings. Hence, this method may overestimate the water depth around the buildings.

Use model: **1D2D_Urban_completed_buildings.xp**

Adding fill areas to represent the buildings:

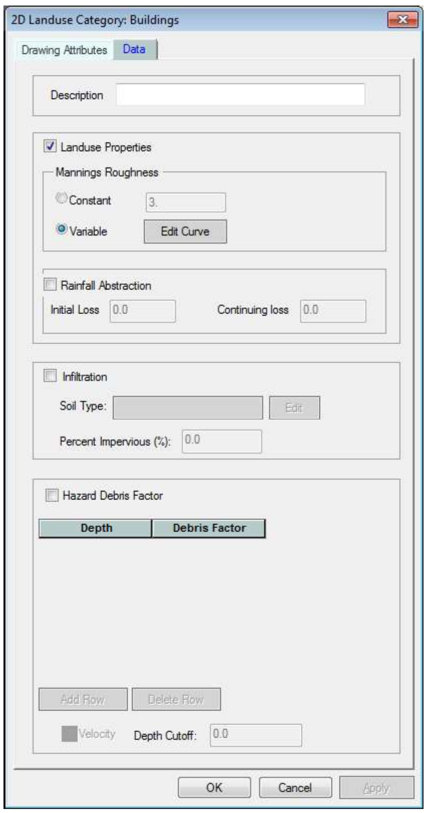
Another technique is adding fill areas on the buildings' locations. User can import polygons to the **Fill Areas Layer** and specify the fill elevation. The fill elevation can be up to the sill of the openings and once the water level reaches above this level, it can enter the buildings. How? Import the buildings.shp file to the fill area.

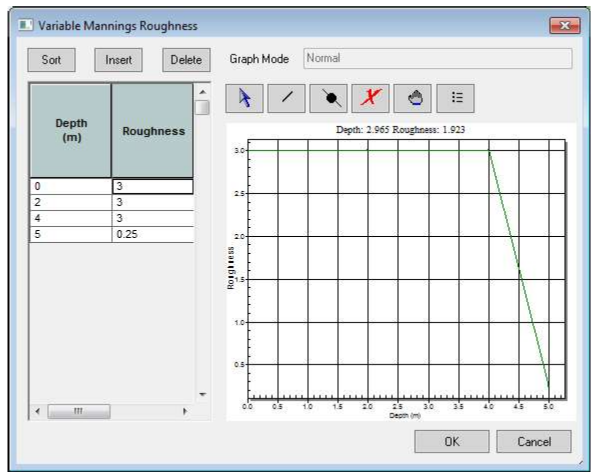


This would be an appropriate approach as it represents the flow condition properly. However estimation of the fill depths for buildings might not be easy as this varies for each building.

Assigning varying Manning's roughness depends on depth:

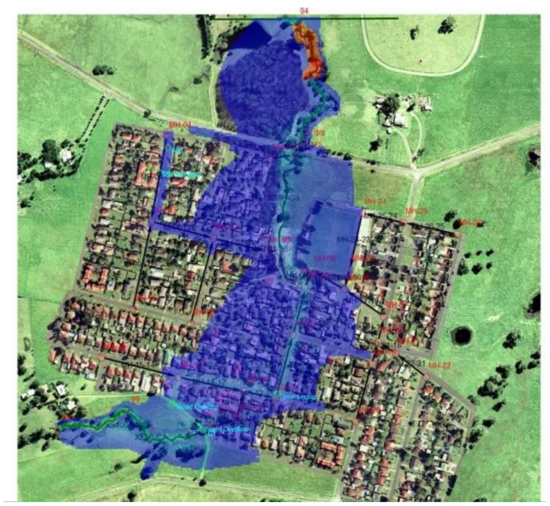
Another technique could be varying Manning's roughness depends upon the depth of flow. The roughness value can be very high up to the sill level of the opening and can be reduced after this depth. User can right click on each **2D Landuse** layer and edit the depth dependant roughness value. As per the entry in the following dialog box, the values change vs. depth.





Hazard Classification

On the **Reporting Layer**, right click and select **Hazard**. Note that this hazard map is obtained by multiplying **maximum depth** and **velocity** for each cell.



You can see from the map that the areas along the open channels and some of the main roads are showing a high hazard value. These areas can probably be classified as high hazardous areas.

Mitigation Option

The manhole/pit overflows are due to the insufficient conveyance of the drainage pipes. The easiest way to tackle this issue is to increase the conveyance of the drainage pipes. However, this increased conveyance will create more floods in the d/s channel overbanks. Due to a large inflow coming from the u/s node during a 100 yr design event the designed drainage system is unable to carry the flow downstream due to insufficient conveyance. Therefore an upstream detention is the most obvious solution in this case.

A model with a detention pond at the upstream area has been completed. Open the model **1D-2D_Urban_completed_buildings_Pond_mitigation.xp**, the detention pond is made using the **Fill Areas** option under the **Topography Layer**. The table below shows the storage characteristics of the pond:

Elevation (m)	Waterspread Area (Ha)
43	1.94
45	2.74
47	3.68
48	4.84
49	6.06

4 2 m diameter concrete pipes are connected from the channel to the pond to divert the flow from the channel to the pond. **Solve** the model and review results. You will see that the flood depth at the urban areas is greatly reduced due to the u/s pond.

If you check the hydrographs at the u/s and d/s of the node where the diversion pipe is connected and the flow from the node to the pond, you will see that significant flow is diverted to the pond. After 1.5 hrs the pond discharges back to the channel and retains 3m depth of water in the pond. Thus pond acts as an offline detention cum retention pond.