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 Hdr 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.

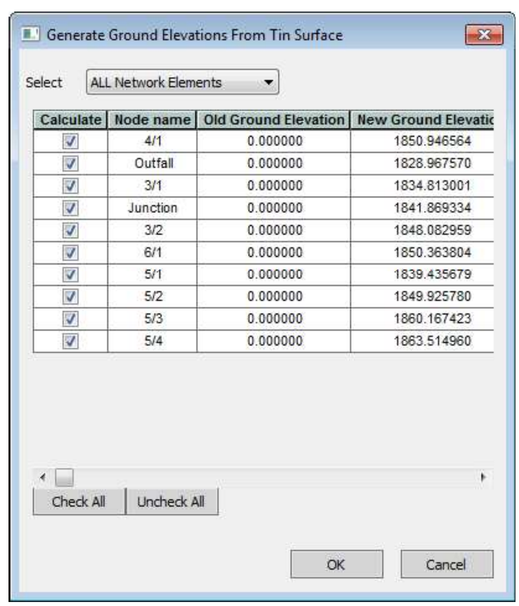
9. Use **File->Save as...** and use the new name **stormwater7.xp**.

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.

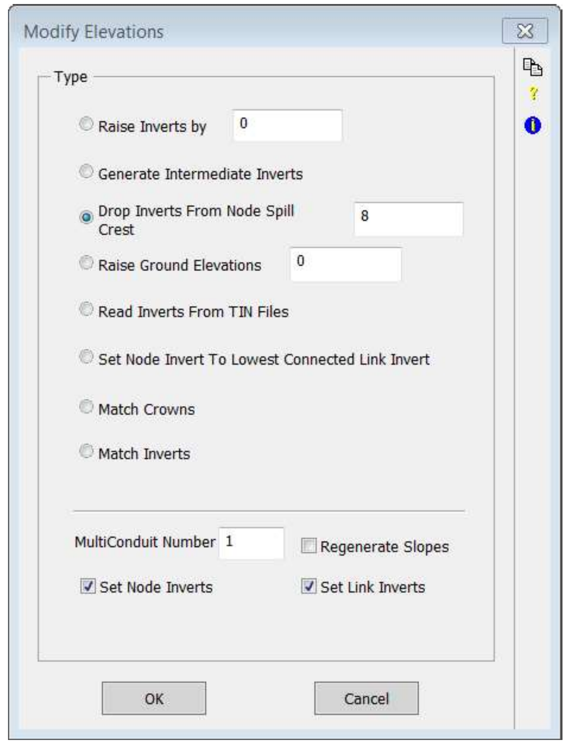
4. Click on the **Tools ->Modify Elevations**.

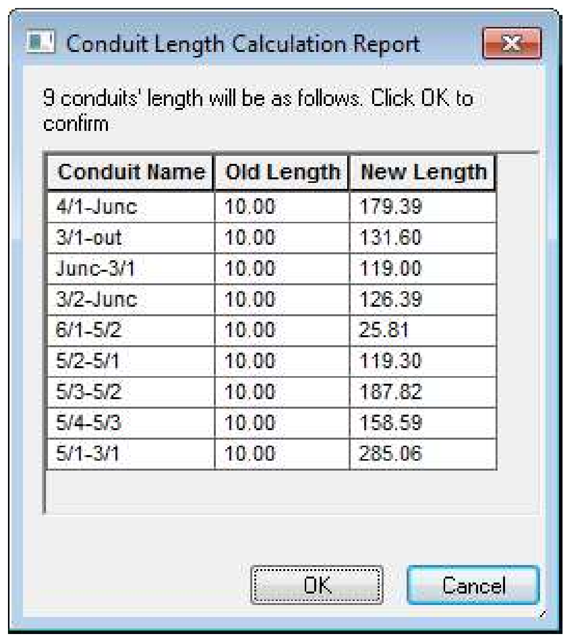
5. In the dialog box, check **Drop Inverts From Node Spill Crest** and enter **8** feet. Make sure that the check boxes for **Set Node Inverts** and **Set Link Inverts** are both active.

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.

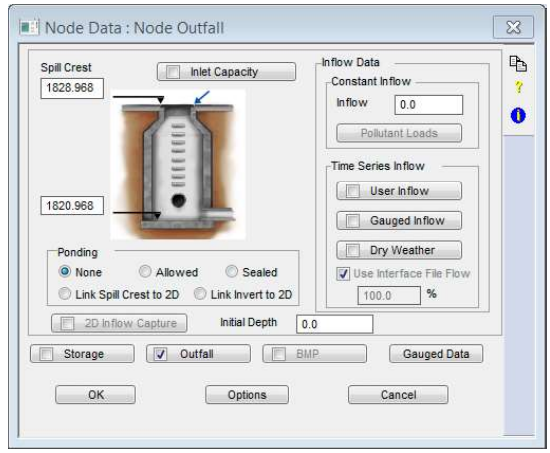
8. With inverts and link set we can now calculate conduit slopes. Conduit slopes can be calculated individually by selecting a conduit and Conduit Profile (**F3**) or for all Conduits. **Tools->Calculate Conduit->Slopes** then choose **All** and **Calculate**. Calculated slopes are shown in a similar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





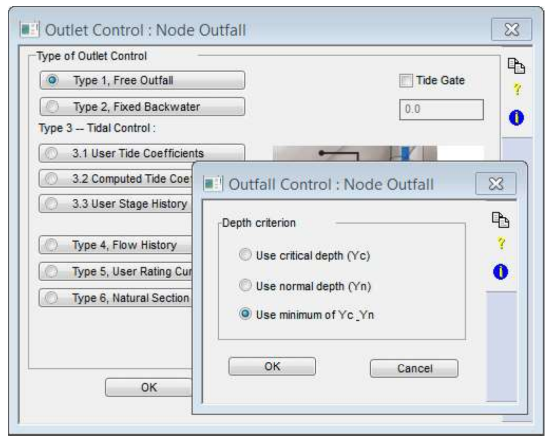
Conduit Name	Old Length	New Length
4/1-Junc	10.00	179.39
3/1-out	10.00	131.60
Junc-3/1	10.00	119.00
3/2-Junc	10.00	126.39
6/1-5/2	10.00	25.81
5/2-5/1	10.00	119.30
5/3-5/2	10.00	187.82
5/4-5/3	10.00	158.59
5/1-3/1	10.00	285.06

Entering node and link data in Dialog Boxes

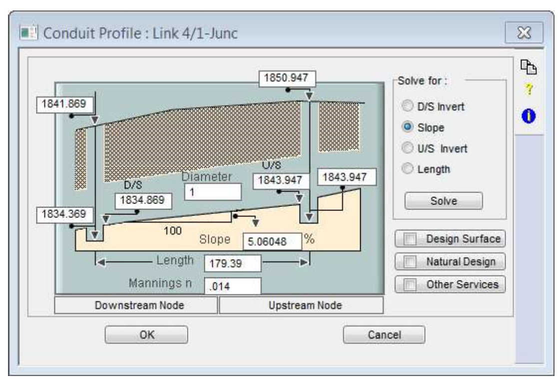


1. **Double-click** on node **Outfall** to open the node data dialog. Click on the **Outfall** button to convert the node to an outfall.

   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 **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.

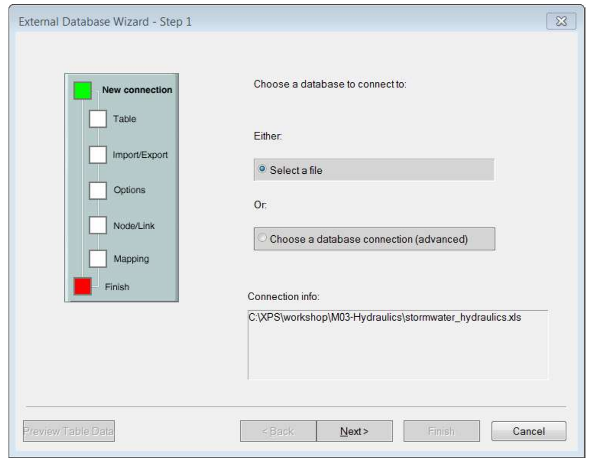


3. Double click on link **4/1-Junc**. Click on **Conduit Profile** or **F3** with the link selected. The upstream and downstream inverts and the length should have been imported in previous steps.

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.

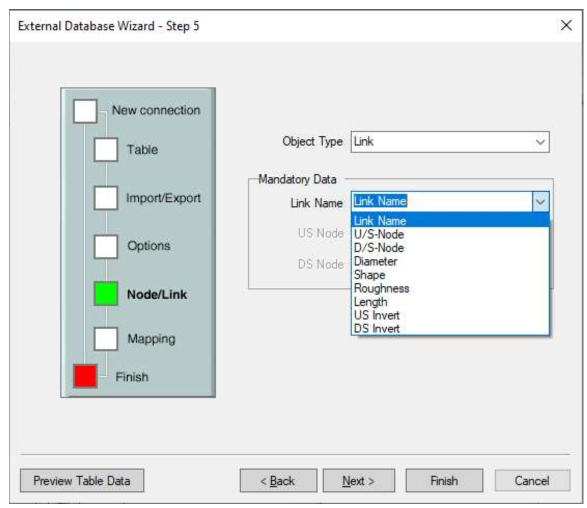


7. Click on **Select a File**. In the Open dialog, highlight **conduits.dbf** and click on **Open**. Click on **Next**. In the next screen of the wizard (step 2), select **conduits** in the Tables dropdown list. Select **Next**.

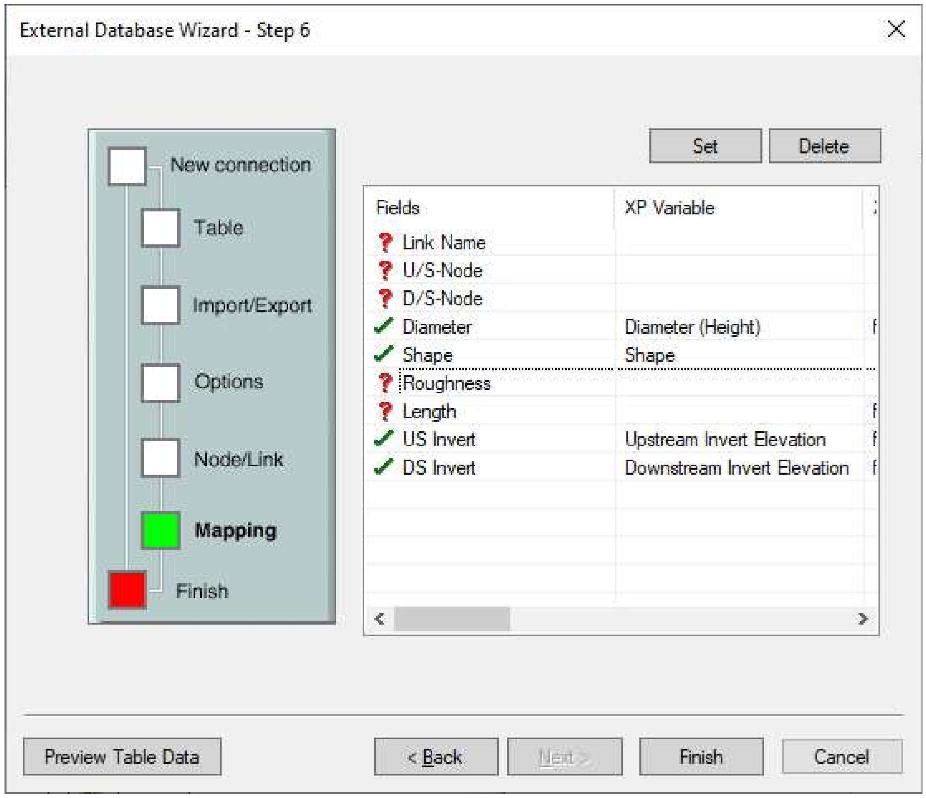
8. In step 3 of the wizard, select **Import Data Only** and select **Next**.

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.



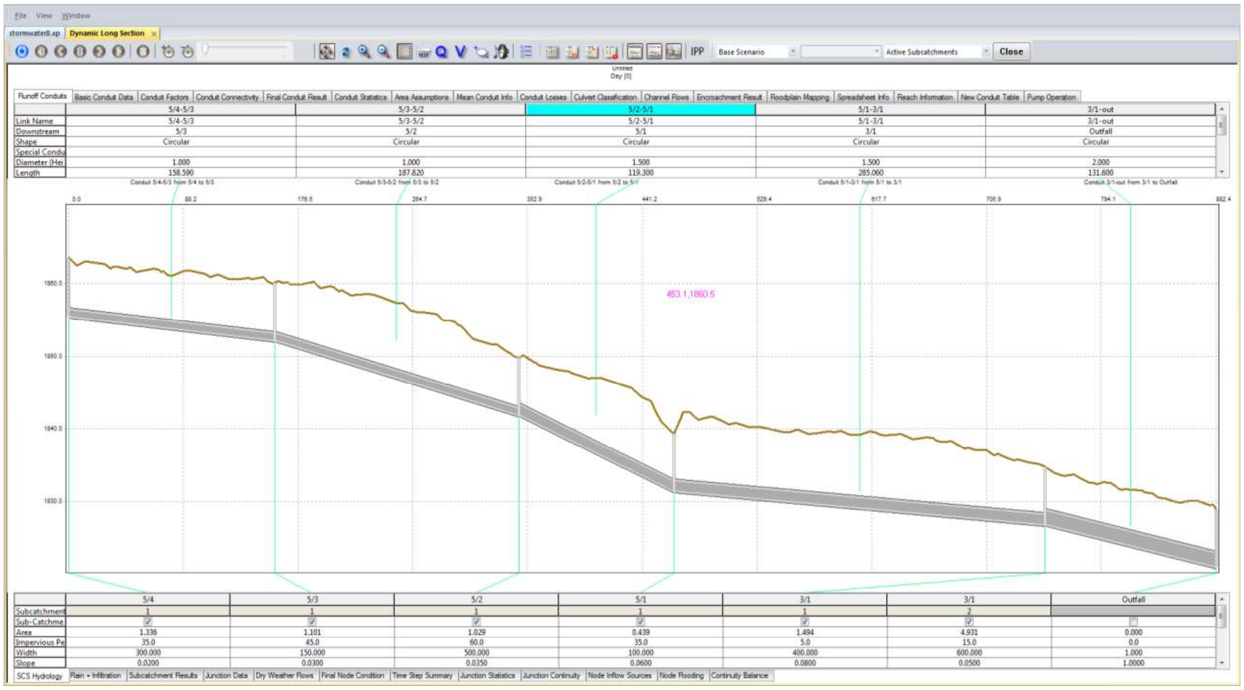
12. With **conduit** highlighted, select **Import**. A preview table will be displayed. Data may be viewed but not edited here. After reviewing the data, click on **OK.**

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.



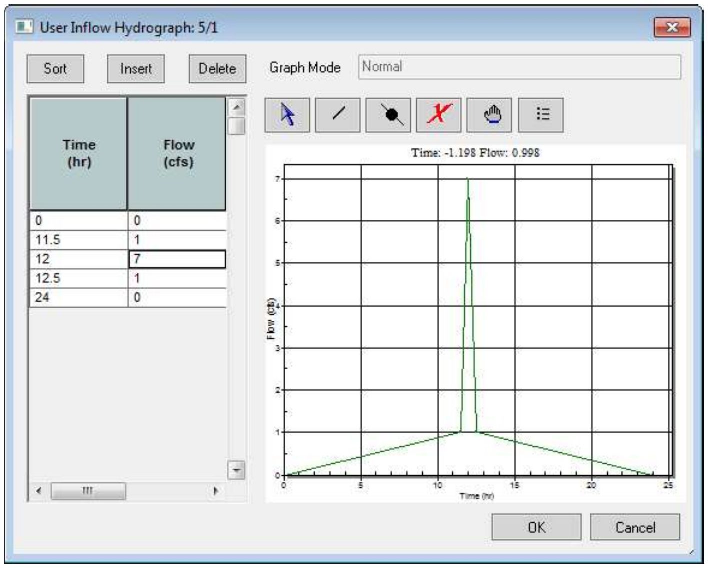
17. From the view above we can see that the inverts are matched. Using **Tools -> Modify Elevations…** we could automatically **Match Crowns** and also **Match Inverts**.

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.

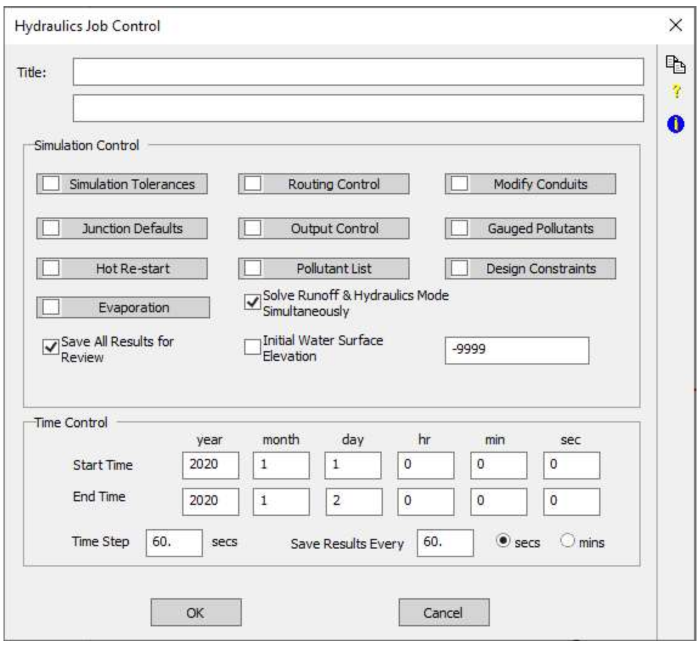
1. What is the ground elevation of Node 5/1? _____
2. What is the invert elevation of Node 3/2? _____
3. What is the length of Link 6? _____ ft
4. What is the peak user inflow at Node 5/1? _____ cfs

Job Control Settings & Running the Model

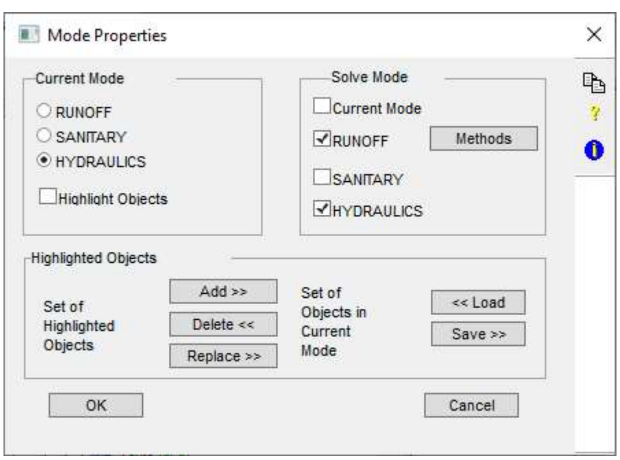
Settings for running the model are managed in the Job Control dialog. This part reviews some of the Job Control settings in the Hydraulic mode.

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
   ```

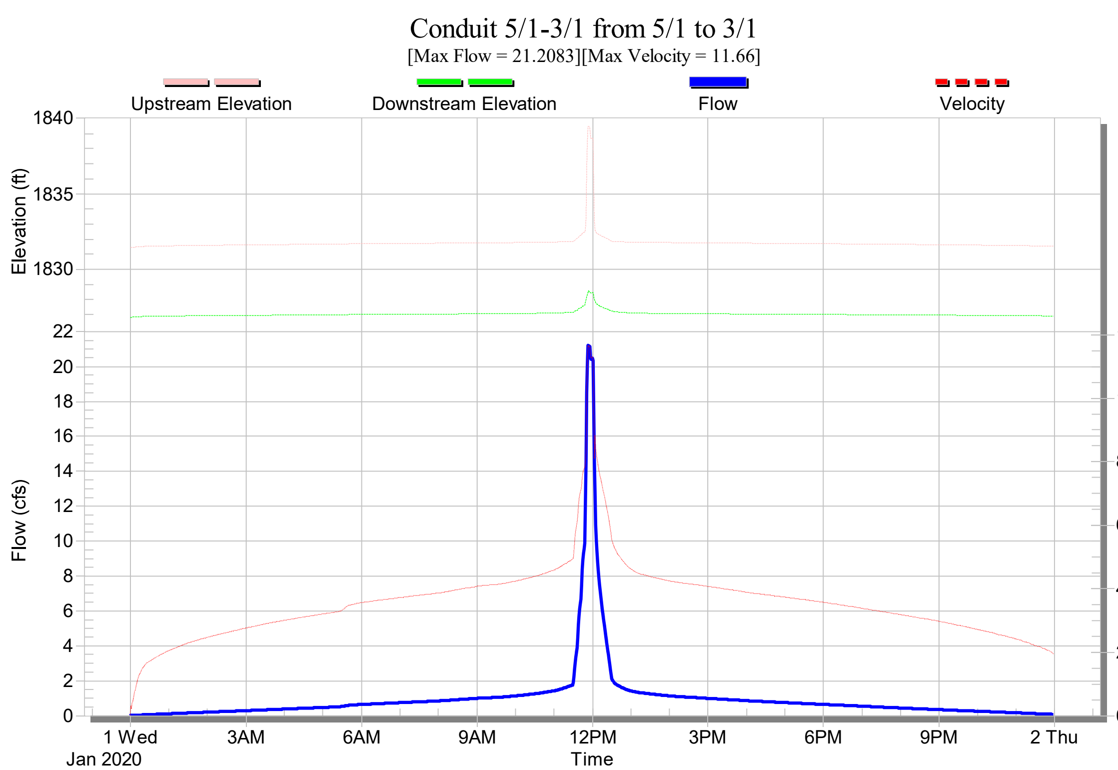
   We can proceed with the warning at this point. The warning means that all short pipes less than the default minimum will be simulated using the minimum length. As the warning indicates the default minimum can be altered using a configuration parameter.

Reviewing the Model

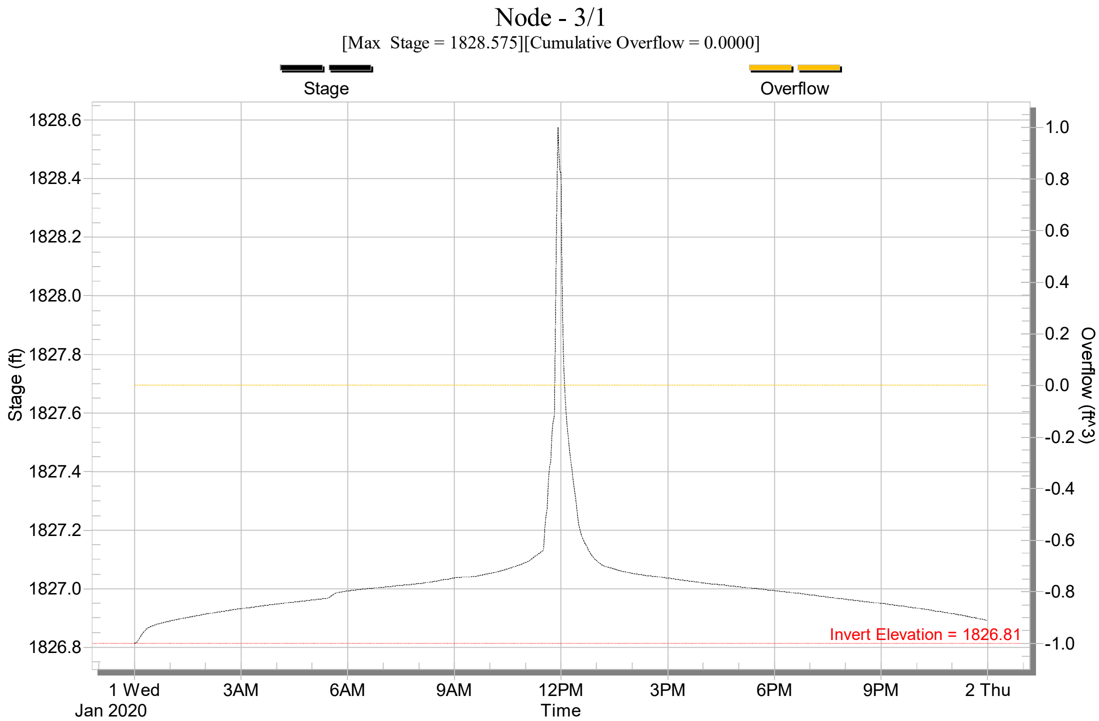
A variety of tools are available for examining model results. In Workshop Module 2, we explored the output from the Runoff Mode. In this example, we will explore output options in the Hydraulics Mode.

Examining graphical results

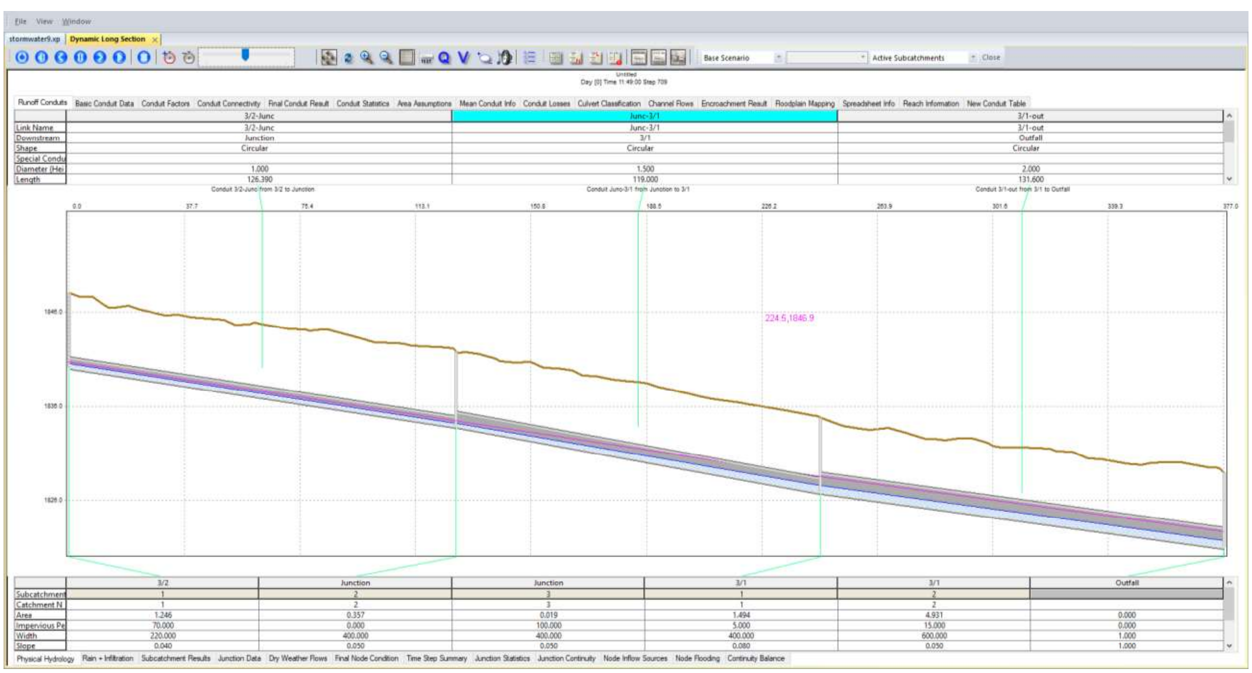
1. To get graphical results, you can right click on a node or link and select review results or you can left click to select the node or link and select the review results icon, or with the node or link selected, click on the review results line or you can use the **F7** button on the keyboard. Now select **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	Continuity Error % of Node	Continuity Error % 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? ______