Practice Workbook

This workbook is designed for use in Live instructor-led training and for OnDemand self-study. OnDemand videos for this course are available through *CONNECT Advisor* and on the Bentley LEARNserver (*learn.bentley.com*).

Dynamic Analysis - SWMM

This course is suitable for:

OpenRoads Designer CONNECT Edition – Update 2 (10.02.00.60)

About this Practice Workbook...

- This PDF file includes bookmarks providing an overview of the document. Click on the bookmark to quickly jump to any section in the file.
- Both Imperial and Metric files are included in the dataset. Throughout this practice workbook Imperial values are specified first and the metric values second with the metric values enclosed in square brackets. For example: *12’ [3.4m]*
- This course workbook uses the *Training and Examples* workspace delivered with the software.
- The terms *Left-click*, *Click*, *Select* and *Data* are used interchangeably to represent pressing the left mouse button. The terms *Right-click* and *Reset* are also used interchangeably. If your mouse buttons are assigned differently, such as for left-handed use, you need to adjust accordingly.

Have a Question? Need Help?

If you have questions while taking this course, search in *CONNECT Advisor* for relates courses and topics. You can also submit questions to the Civil Design Forum on Bentley Communities where peers and Bentley subject matter experts are available to help.

Bentley Institute

TRNC03084-1/0001

Course Level: **Advanced**

DO NOT DISTRIBUTE – Printing for student use is permitted

Description and Objectives

Course description

This course teaches the user how to perform a Dynamic Analysis (SWMM).

Skills Taught

This workbook shows you how to:

- Understand the different solvers
- Review the starting data
- Create a new Scenario
- Create a new Rainfall Runoff Alternative
- Create a new Hydrology Alternative
- Activate CivilStorm
- Create a new Calculation option
- Compute the Scenario
- Inspect how flows vary over time

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Prerequisites

This training assumes that the user is familiar with OpenRoads Designer CONNECT Edition and its interface, or the OpenRoads technology (V8i SELECTseries 4 and its interface). It assumes that you:

- Are comfortable with OpenRoads Designer fundamentals and are comfortable with the 2D and 3D model duality of OpenRoads technology (setting a view to the 2D or 3D model). *QuickStart for Terrain Display* and *QuickStart for OpenRoads Designer Geometry*, available on Bentley LEARN, are an excellent introduction to these skills and more.
- Are familiar with Bentley Subsurface Utilities. The *QuickStart - Evaluating Subsurface Utilities in OpenRoads Designer* and the *QuickStart – Laying out a Drainage Network in OpenRoads Designer* classes are available on Bentley LEARN, are an excellent introduction to utility modeling.

In-Depth Follow-ups

Some side topics are touched on superficially in this class. There are many additional drainage-related classes available on the Learn Server.

Exercise 1: Reviewing Scenarios, Alternatives and Properties

Description

This exercise discusses Scenarios, Alternatives and Properties.

Skills Taught

- Activating the Product
- Understanding Scenarios and Alternatives
- Reviewing this files Scenario Structure

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

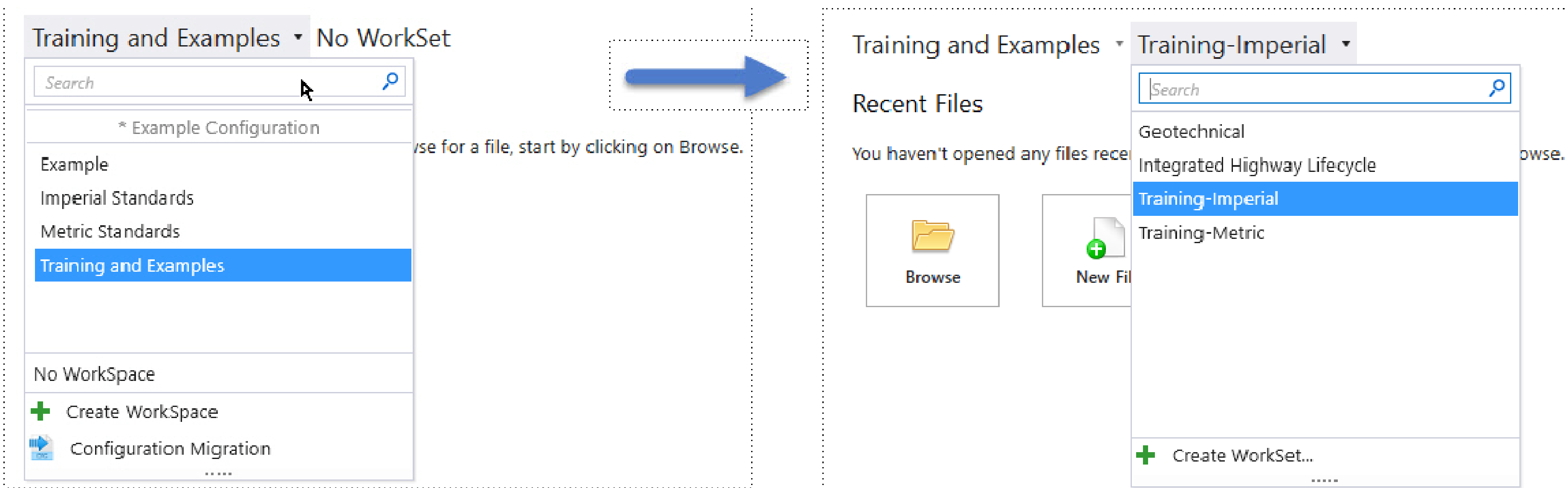
Open the DGN File with the Proper Workspace

1. Start the software.
2. Set the Workspace and Workset.

   The workspace and workset define standards that are used by the software. The Workspace and Workset used for this training are installed during the software installation.

   a. Select **Training and Examples** from the *Workspace* menu.

   b. Select **Training-Imperial** *[Training-Metric]* from the *Workset* menu



3. Browse to the folder where you unzipped the dataset files and select the file **_Drainage Design.dgn** *[_Drainage Design-Metric.dgn].*

Copyright © 2018 Bentley Systems, Incorporated

Understanding the different solvers

Description

We will start by discussing the different hydrology and hydraulics methodologies available in the software.

Calculating the hydraulics of wastewater collection systems is very difficult. It requires solving equations for partly full and full pipes with pumps and various control structures. Rainfall is not a constant intensity, although manual calculations assume so for simplicity. With the power of computer processing allowing us to more accurately model real-world behavior, a variety of hydrologic and hydraulic computation methodologies are used throughout the world.

Rational Method

In many of the other classes (*QuickStart - Evaluating and Creating Subsurface Utilities*, and *Managing Multiple Scenarios* are two examples) we have used the Rational method when we have computed our designs. This method – referred to as **GVF-Rational (StormCAD)** in the product, is known as a "solver" – in other words it is one method of computing, or solving, the hydrology and hydraulics.

The **GVF-Rational (StormCAD)** solver is used for storm water runoff from small areas in which the assumptions underlying the Rational method are valid. The assumptions include homogenous catchments up to a total area of *350 acres* [*150 hectares*]. These would be typical of subdivisions, industrial facilities, and commercial areas, and upstream of any ponds. It routes peak storm flows developed using the Rational method. Once flows have been determined, the hydraulic grade (including depth and velocity) is determined using gradually varied flow (GVF) methods, for both gravity and pressure pipes.

Major limitations of the Rational method include:

- Its use of the Peak Flow, results in conservative design. Pipes may be larger than necessary and inlets larger or more numerous than required.
- Its use of Peak Flows, rather than Hydrographs, hinder pond and pond outlet design.

Dynamic Methods

To produce a hydrograph of flow against time, a different solver must be used – one that can perform an unsteady analysis. For storm water systems, there are two choices – the **Explicit (SWMM Solver),** and the **Implicit (SewerGEMS Dynamic Wave)** solvers:

The **Implicit (SewerGEMS Dynamic Wave)** solver solves the full St. Venant equations using an implicit numerical method developed by Bentley based on the US National Weather Service FLDWAV model. It simultaneously solves for both flow and hydraulic grade and uses the same equations for gravity and pressure portions of the system. It only solves dynamic flows (no steady state). It can be applied to storm, sanitary and combined sewers.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

The **Explicit (SWMM Solver)** solves the full St. Venant equations using an explicit numerical method based on the EPA-SWMM model. In addition to **Explicit (SWMM Solver)** solving the St. Venant equations using a dynamic solution, it can also route flows using a kinematic wave solution and a uniform flow solution, which does not account for any backwater effects. The Explicit dynamic solver simultaneously solves for both flow and hydraulic grade and uses the same equations for gravity and pressure portions of the system. It can be applied to storm, sanitary and combined sewer, especially systems without a great deal of force mains or pressure sewers.

The EPA-SWMM model is well known worldwide, so it is the one that we will be using in this class.

Note: The **Implicit (SewerGEMS Dynamic Wave)** and **Explicit (SWMM Solver)** methods do not design a system – they analyze it – in other words they will not change conduit sizes, elevations etc. So, it's normal practice to use the GVF-Rational (StormCAD) solver to calculate conduit sizes, elevations etc., then to analyze the resulting system with one of these two solvers, if you need to route hydrographs or model ponds.

There is a fourth solver, which is used for sanitary (or foul) sewers – the **GVF-Convex (SewerCAD)** solver. It separates the hydraulic problem first into a set of gravity subnetworks and pressure subnetworks. For the gravity subnetworks, it divides the solution into a flow routing problem using convex routing (a hydrologic routing method) and then solves for hydraulic grade (including depth and velocity) using gradually varied flow (GVF) (backwater) equations. For the pressure subnetworks, it uses a true pressure solver as found in Bentley's WaterGEMS. It can perform extended period simulations and steady state simulations and can base steady simulations on an extreme flow factor method that reduces peaking factors as the flow increase moving downstream. The solution method was developed for the SewerCAD model. It can be applied to sanitary and combined sewers, especially those with a great deal of pumping or pressure sewers, and those which only require a steady state analysis.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Availability of Different Solvers

If your Civil product includes a license for drainage design functionality then, by default, you will be using a license of StormCAD for 100 inlets. This license gives you access to the **GVF-Rational (StormCAD)** solver. Other licenses are available, for products which give you access to the other solvers.

This table shows the available Solvers by product:

Solver	StormCAD	SewerCAD	CivilStorm	SewerGEMS
GVF-Rational (StormCAD)	Yes	No	Yes	Yes
GVF-Convex (SewerCAD)	No	Yes	No	Yes
Explicit (SWMM Solver)	No	No	Yes	Yes
Implicit (SewerGEMS Dynamic Wave)	No	No	Yes	Yes

This class uses the Explicit (SWMM Solver), which is available in either a CivilStorm or a SewerGEMS license. These licenses are not included by default, so activating them will log usage against the License Server.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Exercise 2: Review the Starting Data

Description

In this exercise you will review the data in the design file, looking at how Scenarios have been used to model different situations.

Skills Taught

- Reviewing Scenarios
- Reviewing Calculation options
- Reviewing IDF Rainfall Data
- Reviewing the Hydrology Alternative

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Reviewing the Scenarios

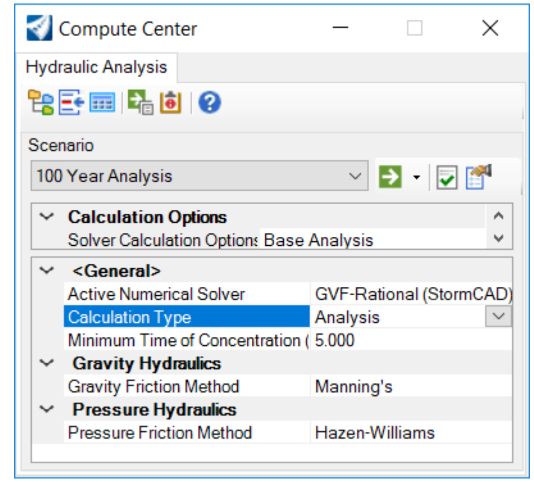
1. In Subsurface Utilities workflow/ribbon, click on **Analysis > Analysis Tools > Compute Center**:

2. In the *Compute Center* dialog, the **Scenarios** drop-down list may be the most important control in Subsurface Utilities. It lists the Current Scenario, which controls the structures that are displayed, the calculation results in the database, as well as all the Current Alternatives such as Physical (Sizes) and Design options – in short, it controls everything.

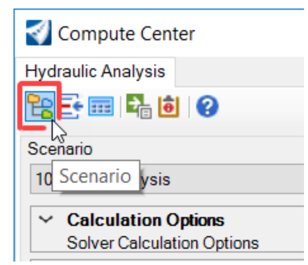
   Selecting a scenario from the drop-down list makes it Current, which updates database values, graphics, profile HGLs, FlexTables, etc.

   The *100 Year Analysis* scenario has its *Calculation Type* set to **Analysis**, which means that sizes and Physical properties will change.

   This functionality was covered in the *Hydraulic Analysis and Design* class.



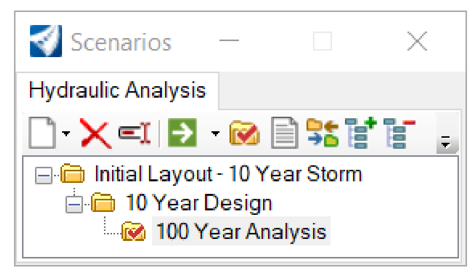
3. In *Compute Center*, click the **Scenario** icon:



   The *Scenarios* dialog shows the file's Scenarios and their dependencies.

   The *Scenarios* dialog shows three scenarios. The Current Scenario should be the *100 Year Analysis*. It is essentially a 10 Year Design with some upsized pipes, analyzed with a 100 Year Peak Flow Storm. We will use this as a starting point for the SWMM Analysis.

   Note: this scenario has the same settings as the final scenario in the Managing Multiple Scenario course. You can use that design file if you just completed that course. This book will refer to 100 Year Analysis instead of the more-specifically named Scenario last created in that course.



4. In the *Scenarios* dialog, click the **Compute** icon to compute the scenario. This ensures that the calculation numbers match the current settings in this file.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

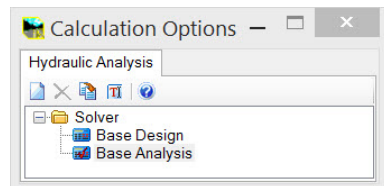
Reviewing Calculation Options

1. In the *Scenarios* dialog, double-click the **100 Year Analysis** scenario.

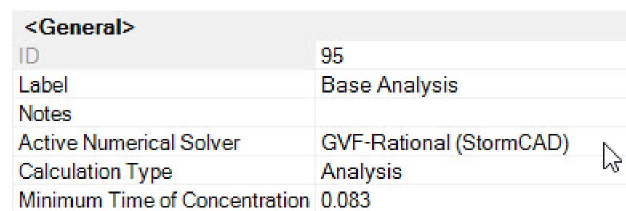
   At the bottom of the properties in the *Hydraulic Analysis* tab, review the *Solver Calculation Options*. This is set to **Base Analysis**, and it has been inherited from a parent scenario.

2. In the *Subsurface Utilities* workflow/ribbon, click on **Analysis > Calculation > Options**.

3. In the Calculation Options dialog, double-click **Base Analysis**.



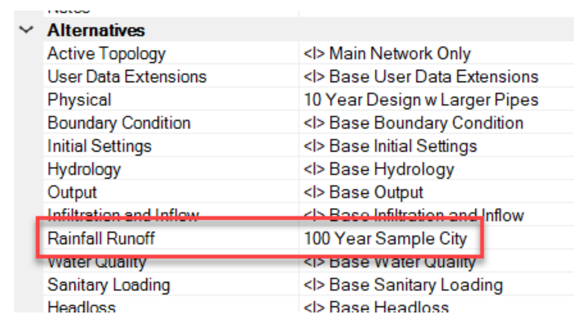
   The properties in the *Hydraulic Analysis* tab show that the *Active Numerical Solver* is set to **GVF-Rational (StormCAD).** We know that this is a peak flow solver, and we want to calculate runoff hydrographs and route them through the system.



Reviewing IDF Rainfall Data

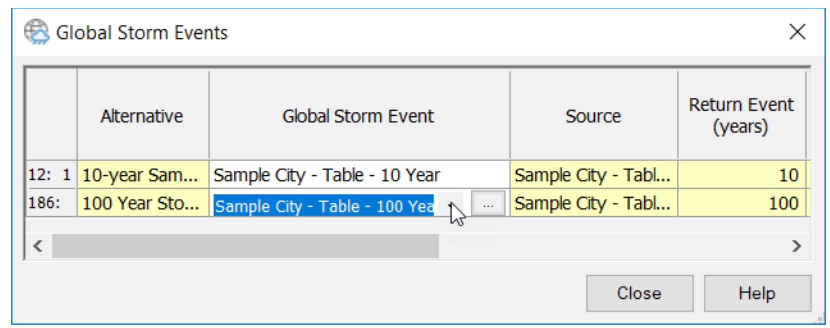
1. In the *Scenarios* dialog, double-click the **100 Year Analysis** scenario again.

The properties in the *Hydraulic Analysis* tab show that the *Rainfall Runoff Alternative* is set to **100 Year Sample City**.



2. Click on **Subsurface Utilities > Components > Common > Storm Data >> Global Storm Events**:

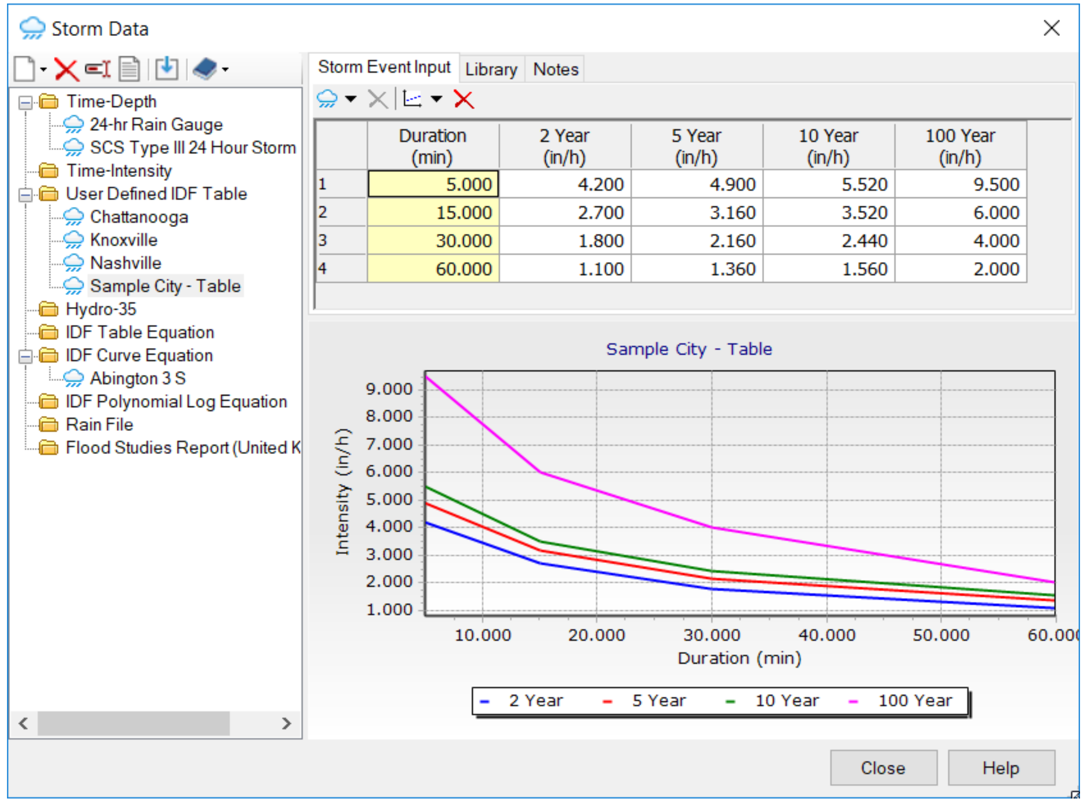
The *Global Storm Events* dialog shows that there are two Rainfall Runoff Alternatives, and it tells us the Global Storm Event that each one uses.



3. **Close** the *Global Storm Events* dialog.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

4. Click on **Subsurface Utilities > Components > Common > Storm Data >> Storm Data**:

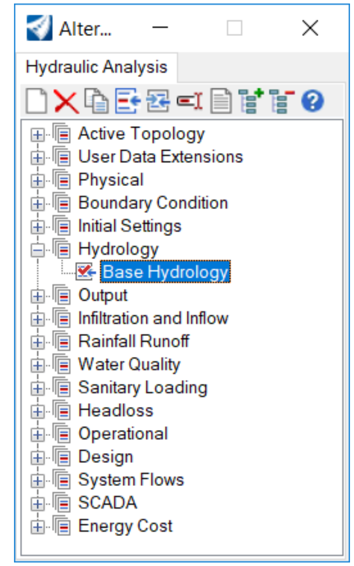


The *Storm Data* dialog shows us that the **Sample City – Table** data is in IDF format (Intensity Duration Frequency). This kind of data will not produce a runoff hydrograph.

5. Close the *Storm Data* dialog.

Reviewing the Hydrology Alternative

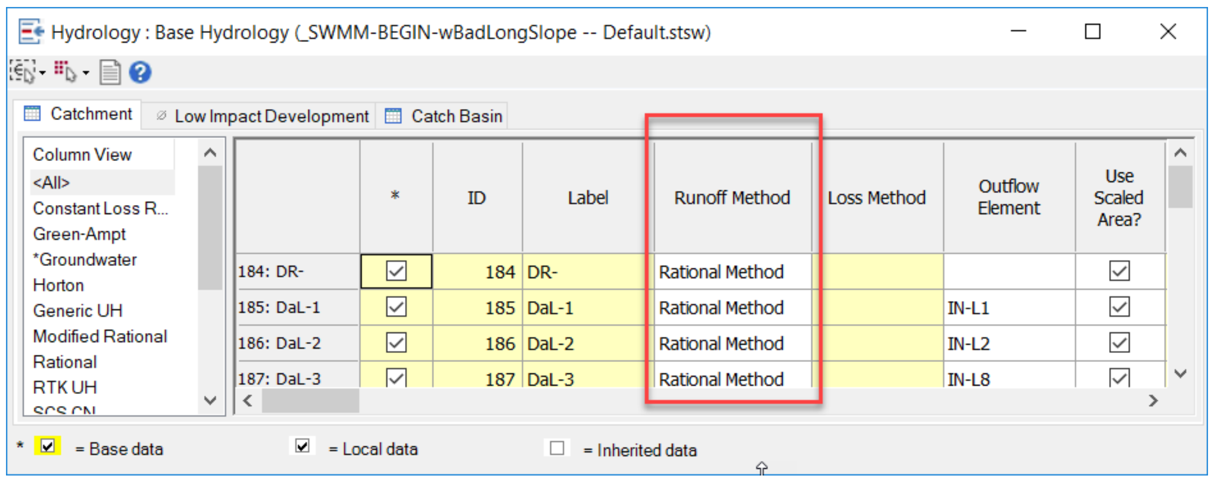
1. Click **Subsurface Utilities > Analysis > Calculation > Alternatives**
2. Expand the tree for the **Hydrology** Alternative:



At the moment, there is only one Hydrology Alternative, called Base Hydrology. Note the red tick mark, which shows that this is the alternative being used by the current scenario.

3. Double-click on the **Base Hydrology** alternative to open the *Headloss Alternative Editor* dialog.

Note that the *Runoff Method* for the catchments is set to **Rational Method**. Again, this will not produce runoff hydrographs.



4. **Close** the *Hydrology Alternative Editor* dialog.

We have reviewed one Scenario in the design file. Reviewing others would show the same sort of thing – they are all set up so that they will only route peak storm flows developed using the Rational method.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Exercise 3: Creating a New Scenario for SWMM

Description

In this exercise you will create a new Scenario, and set it up for the SWMM solver.

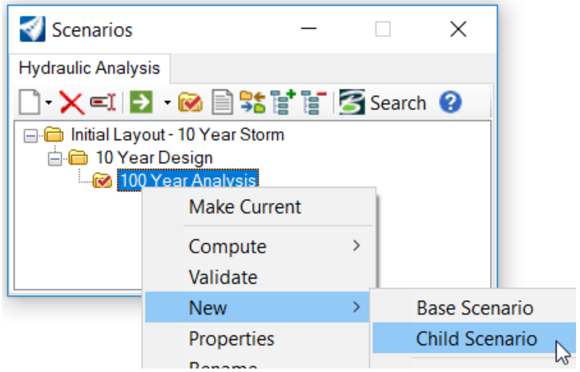
Skills Taught

- Creating a new Scenario
- Creating a new Rainfall Runoff Alternative
- Creating a new Hydrology Alternative
- Activating CivilStorm
- Creating a new Calculation Option

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

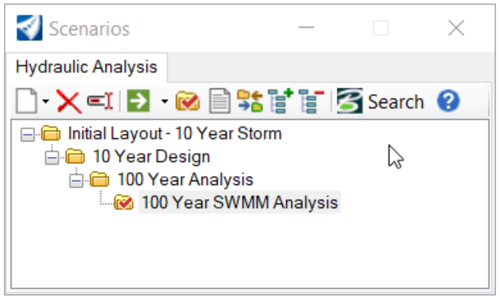
Creating a New Scenario

1. In the *Scenarios* dialog, right-click on *100 Year Analysis.* Select **New > Child Scenario**



2. Type the name **100 Year SWMM Analysis**, and press **return**.
3. Right-click on the new Scenario, and click **Make Current**.

   The *Scenarios* dialog should look like this:



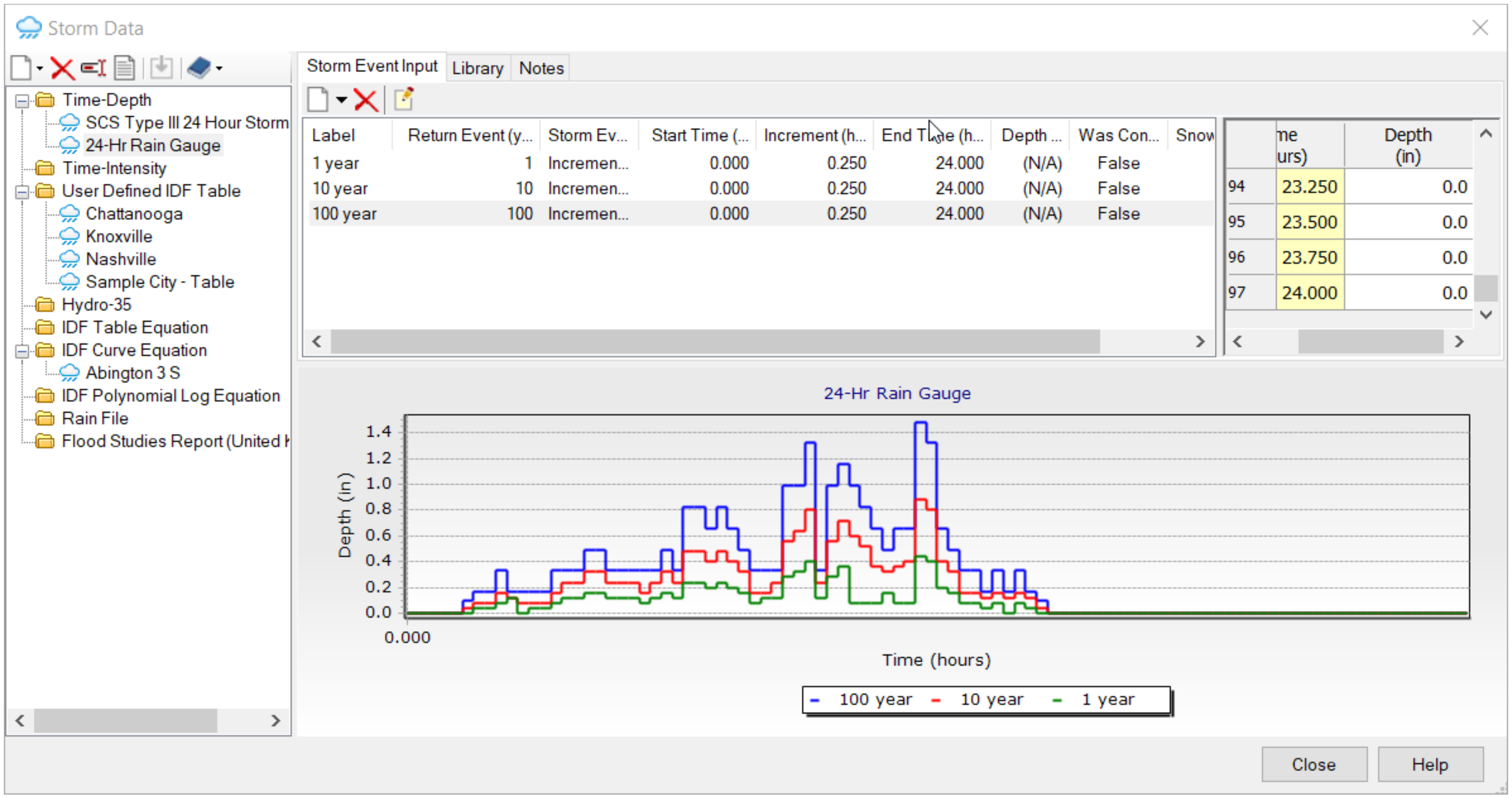
Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Reviewing Rainfall Data

Earlier, we looked at IDF data, which is being used by the Scenarios in the design file. As we discussed, this creates a peak flow for a given Time of Concentration. We need to model how the rainfall intensity or depth varies over time, which is called a Hyetograph.

1. Click on **Subsurface Utilities > Components > Common > Storm Data >> Storm Data**:

   The *Storm Data* dialog shows us that other kinds of rainfall data, as well as the IDF format which we have already seen, are available. In this class, we will use the **24-Hr Rain Gauge** storm, which is a graph of Time against Rainfall Depth.



2. In the Time-Depth group, click **24-Hr Rain Gauge** to display its data.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

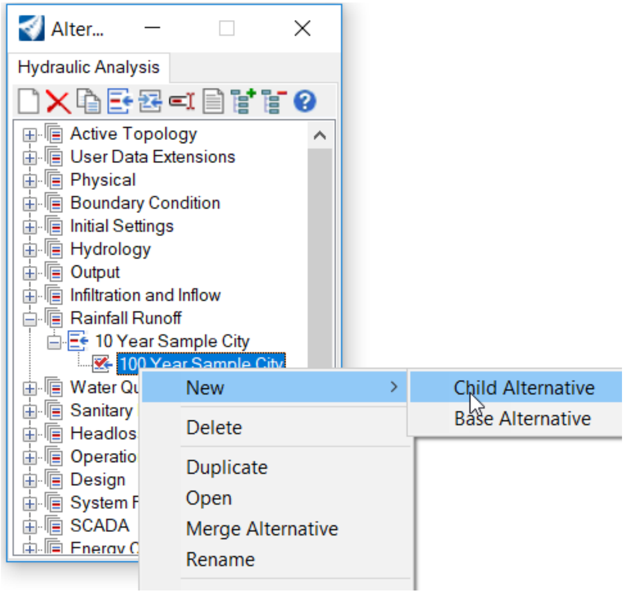
Note that there are three return periods set up here. The grid in the top right corner lists the time and depth for the selected event. The graph – a Hyetograph - at the bottom of the dialog shows us that the depth of rainfall varies over time.

3. **Close** the *Storm Data* dialog.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

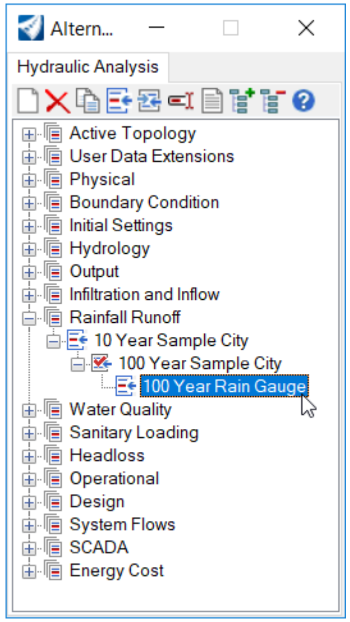
Creating a new Rainfall Runoff Alternative

1. In the *Alternatives* dialog, expand the **Rainfall Runoff** node. Right-click on **100 Year Sample City**, and select **New > Child Alternative.**



A new Alternative called **Rainfall Runoff Alternative – 1** is created.

2. Click on it, and type the name **100 Year Time-Depth**.

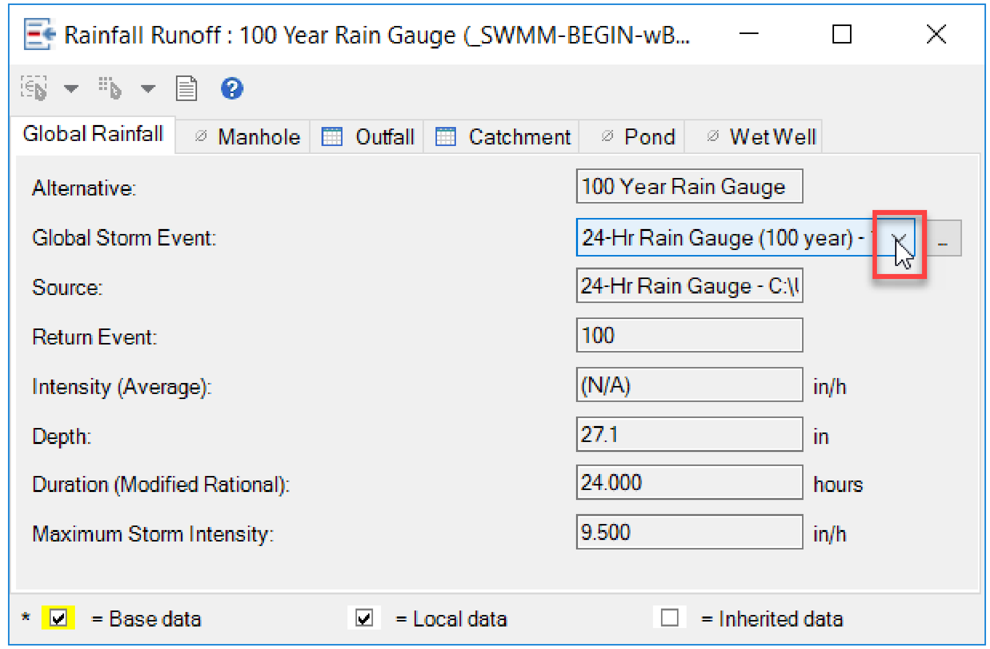


Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

3. Double-click on the **100 Year Rain Gauge** Alternative, to open the *Rainfall Runoff Alternative Editor* dialog.

4. Click the down arrow icon to the right of the *Global Storm Event* field.

5. Select **24 Hour Rain Gauge (100 year) – 100 Year** from the list.

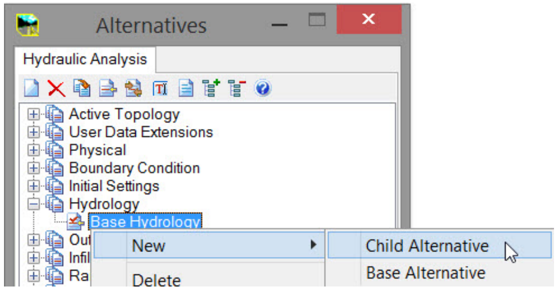


6. **Close** the *Rainfall Runoff Alternative Editor* dialog.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

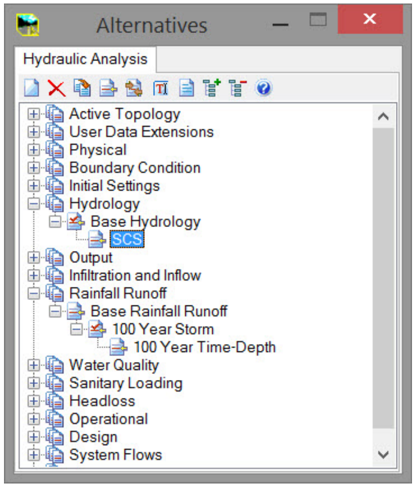
Creating a new Hydrology Alternative

1. In the *Alternatives* dialog, expand the **Hydrology** node. Right-click on **Base Hydrology**, and select **New > Child Alternative.**



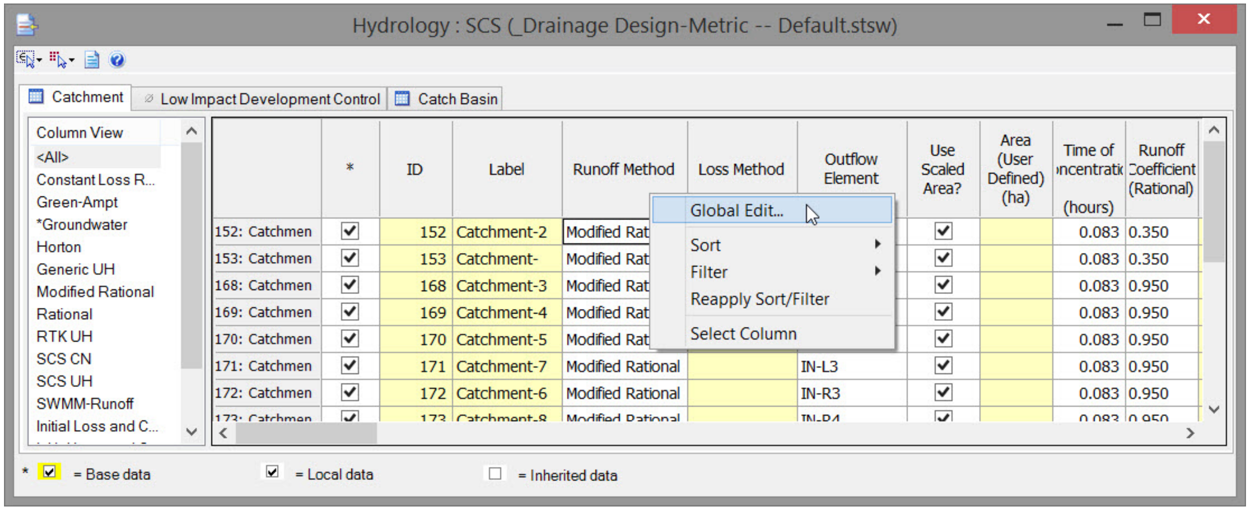
   A new Alternative called **Hydrology Alternative – 1** is created.

2. Click on it, and type the name **SCS**.



3. Double-click on the **SCS** Alternative, to open the **Hydrology Alternative Editor** dialog.
4. Find the column titled *Runoff Method*.
5. Right-click on the column title, and select **Global Edit**.

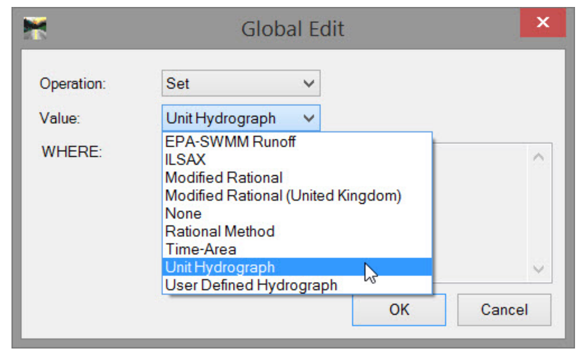
Copyright © 2018 Bentley Systems, Incorporated



6. In the *Global Edit* dialog, click the down arrow. Take a moment to read the runoff methods that are available.

7. Select **Unit Hydrograph**.

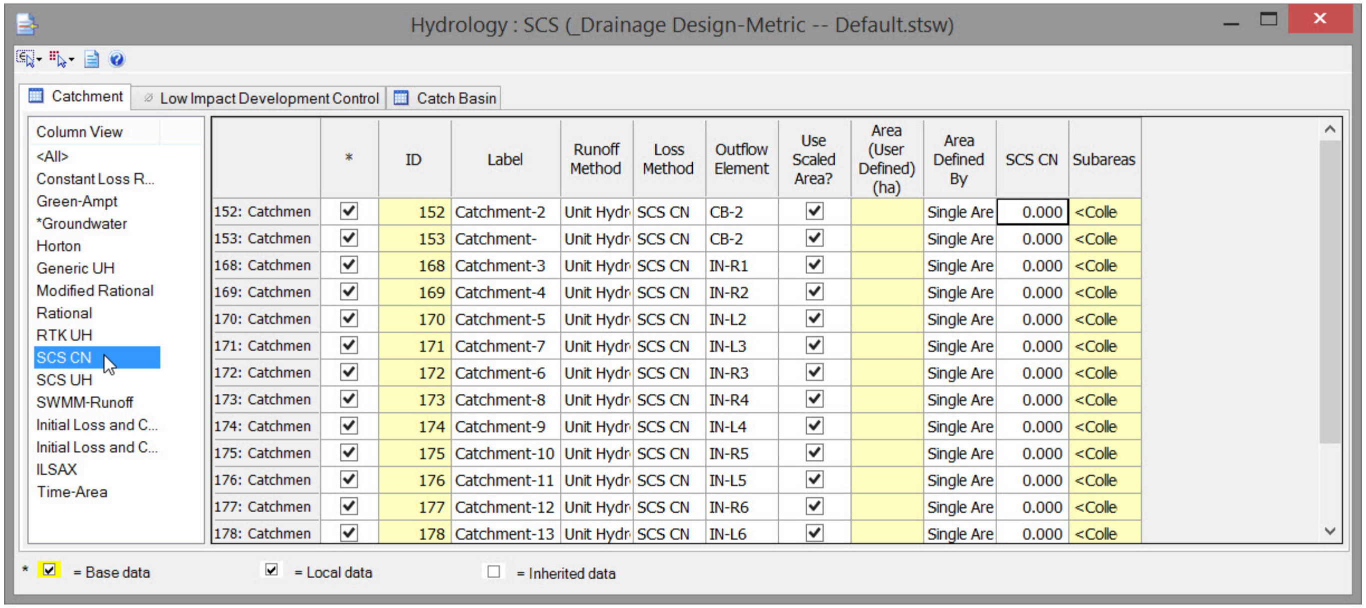
8. Click **OK** to close the dialog.



In the *Hydrology Alternative Editor* dialog, note the *Column View* down the left-hand side. You can use this to filter the columns that are displayed, reducing the list.

9. Click on **SCS CN** (which is short for **Soil Conservation Society – Curve Number**). This reduces the number of columns displayed.

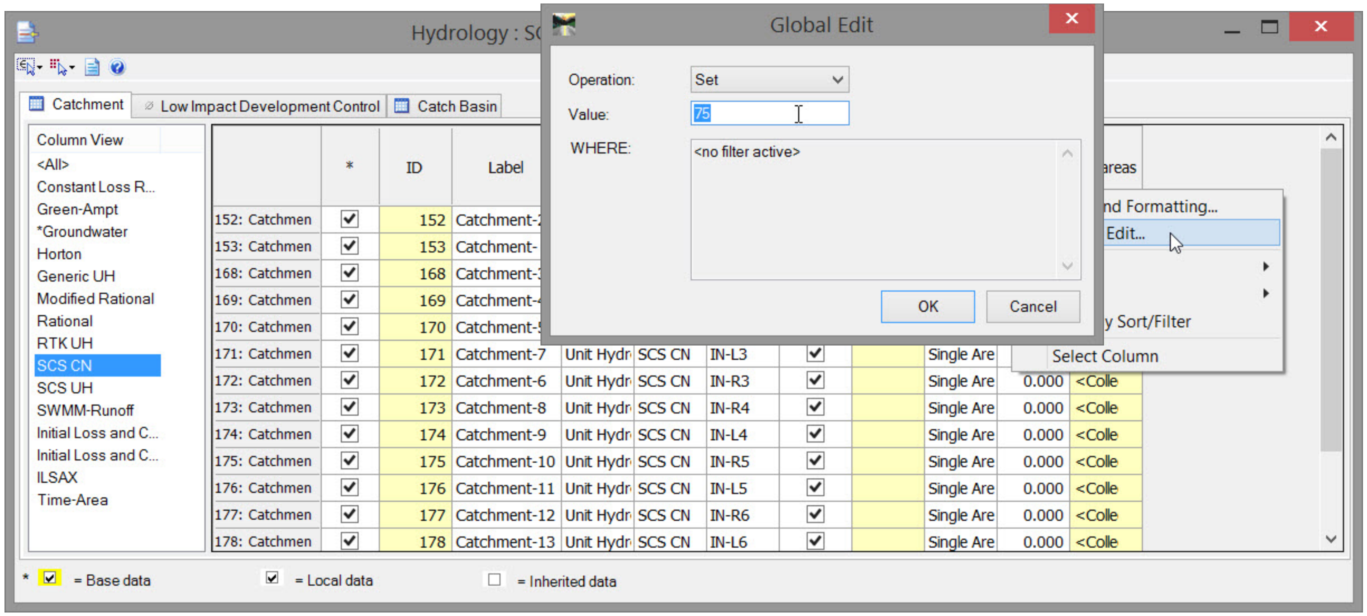
Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted



10. Find the column titled *SCS CN*.

Copyright © 2018 Bentley Systems, Incorporated

11. Right-click on the column title, and select **Global Edit**.



12. In the *Global Edit* dialog, type a value of **75**.

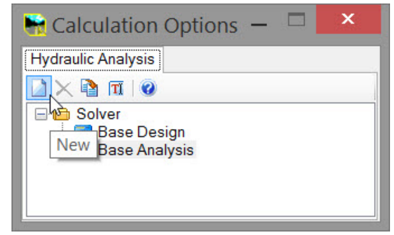
13. Click **OK** to close the dialog.

14. Close the **Hydrology Alternative Editor** dialog.

We have changed from the Rational method of calculating catchment runoff, which only gives a peak flow, to a unit hydrograph method, which will give a runoff hydrograph. The Curve Number is a measure of the porosity of the catchment surface – higher numbers mean less porosity, and therefore higher runoff.

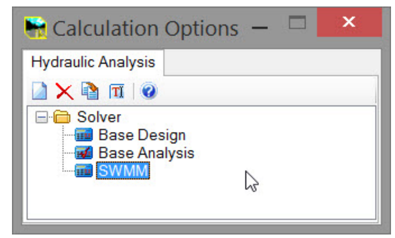
Creating a New Calculation Option

1. In the *Subsurface Utilities* workflow/ribbon, click on **Analysis > Calculation > Options**.
2. In the *Calculation Options* dialog, click **New**.

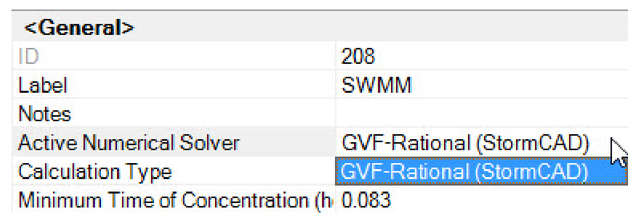


   A new *Calculation Option* called **New Solver – 1** is created.

3. Click on **New Solver – 1**, and type a new name of **SWMM.**



4. Double-click on **SWMM**.



5. The properties in the *Hydraulic Analysis* tab show that the *Active Numerical Solver* is set to **GVF-Rational (StormCAD).**

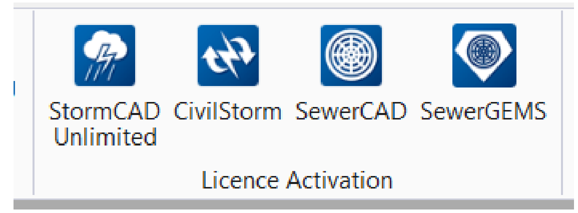
   We know that this is a peak flow solver, and we want to calculate runoff hydrographs and route them through the system, using the SWMM solver.

6. Click on the **GVF-Rational (StormCAD)** text, and click the down arrow.

The SWMM solver is not listed. This is because we are using StormCAD at the moment, which is the default product. We need to be using CivilStorm to make the SWMM solver available.

Activating CivilStorm

1. In the *Subsurface Utilities* workflow/ribbon, click the **Tools** tab.

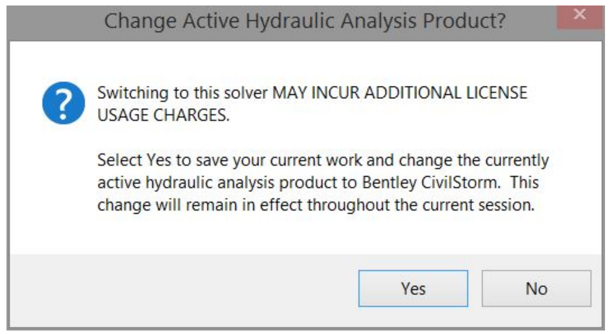


The License Activation group shows the available licenses.

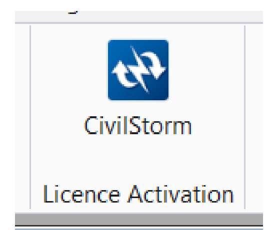
2. Click **CivilStorm**

Note: CivilStorm is not included by default in an OpenRoads Designer license. Activating CivilStorm will log usage against the License Server.

You will see a warning message about licensing and the potential to incur charges if you do not own the appropriate license.



3. Click **Yes**.
4. Once CivilStorm has been activated, many more tools become available, reflecting the extra functionality that is available in CivilStorm.
5. The License Activation group now only shows CivilStorm..



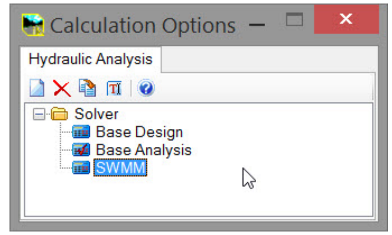
This shows us that the Current Hydraulic Analysis Product is CivilStorm.

Once you have activated a different product, that product will be used for the rest of the session – in other words it is activated until you exit the Civil product.

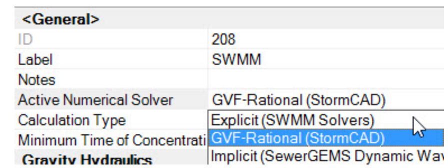
Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Create a New Calculation option

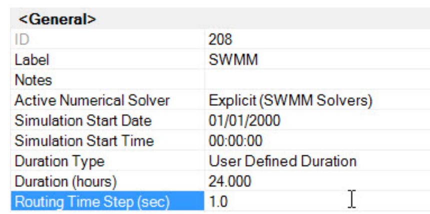
1. In the Calculation Options dialog, double- click SWMM.



2. In the *Hydraulic Analysis* tab, click on the **GVF-Rational (StormCAD)** text, and click the down arrow. More solvers are available now.
3. Select **Explicit (SWMM Solvers)** from the list.



4. Take a moment to review the settings that are available for this solver.
5. Set the **Routing Time Step** property to 1.0 seconds.



6. Close the *Calculation Options* dialog.

Computing the Scenario

We have now set up all of the Alternatives that we need for our SWMM Scenario. The final step before we can compute is to edit the Scenario so that it uses these Alternatives.

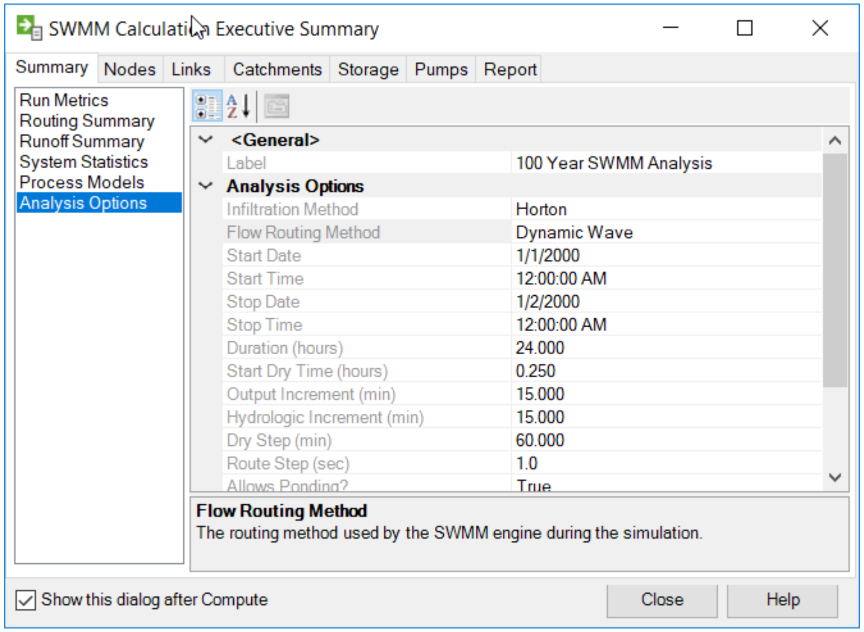
1. In the *Scenarios* dialog, double-click **100 Year SWMM Analysis**.
2. In the *Hydraulic Analysis* tab of *Properties*, change:
   c. The *Hydrology* Alternative to **SCS**
   d. The *Rainfall Runoff* Alternative to **100 Year Rain Gauge**
   e. The *Solver Calculation Options* to **SWMM**.
3. The *Hydraulic Analysis* tab of *Properties* should look like this now:

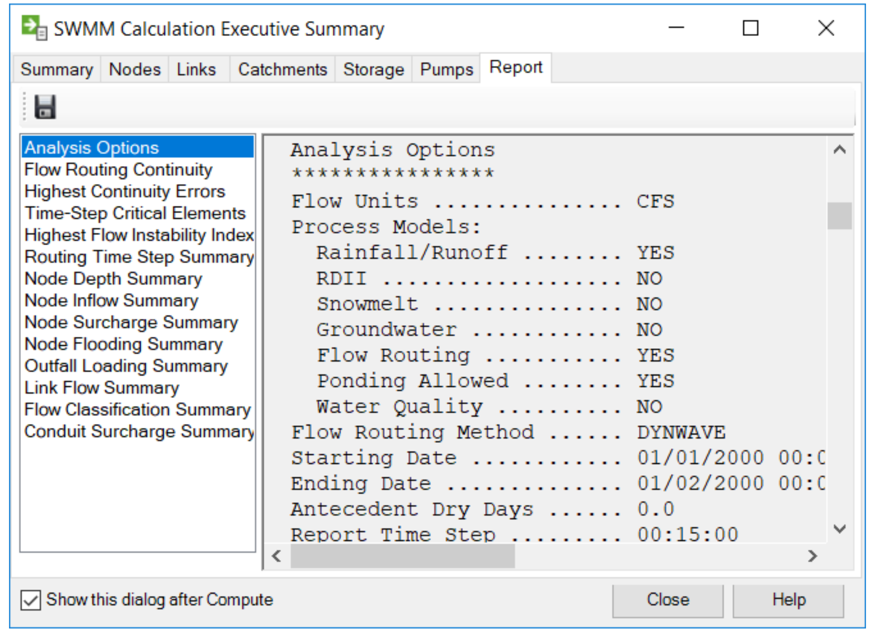
<General>	
ID	201
Label	100 Year SWMM Analysis
Notes	
Alternatives	
Active Topology	<I> Main Network Only
User Data Extensions	<I> Base User Data Extensions
Physical	<I> Larger Pipes
Boundary Condition	<I> Base Boundary Condition
Initial Settings	<I> Base Initial Settings
Hydrology	SCS
Output	<I> Base Output
Infiltration and Inflow	<I> Base Infiltration and Inflow
Rainfall Runoff	100 Year Time-Depth
Water Quality	<I> Base Water Quality
Sanitary Loading	<I> Base Sanitary Loading
Headloss	<I> Adjusted Coefficients
Operational	<I> Base Operational
Design	<I> Larger Spread Width
System Flows	<I> Base System Flows
SCADA	<I> Base SCADA
Calculation Options	
Solver Calculation Options	SWMM

4. In the *Scenarios* dialog, right-click on the **100 Year SWMM Analysis** Scenario, and click **Compute > Scenario**.

   A panel tells you how the calculations are progressing. If you watch it closely you will see that the unit hydrographs are computed. After a few moments the calculations will complete, and the *SWMM Calculation Executive Summary* dialog opens.

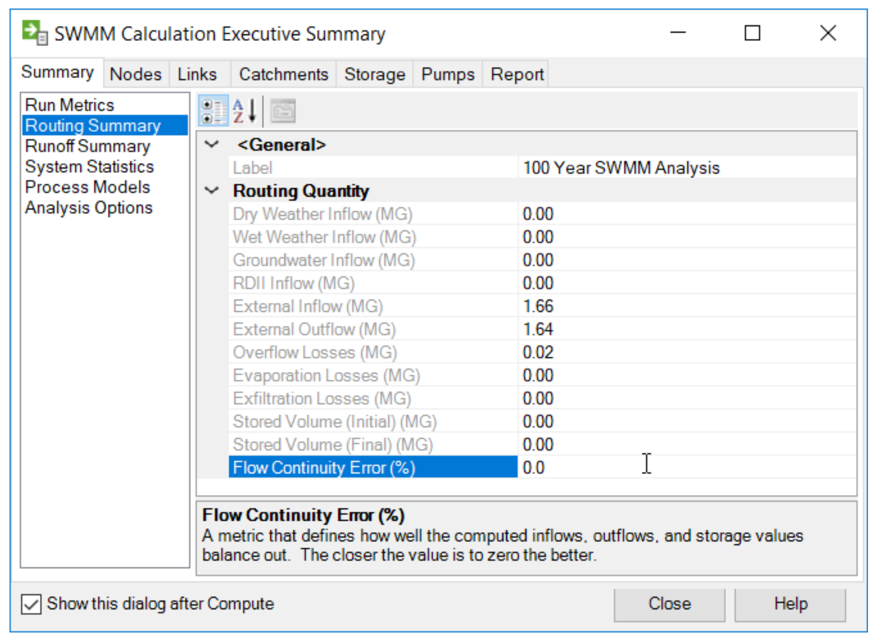
Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted





The tabs across the tops of the dialog provide access to General and Structure-specific Summaries and Reports.

5. Take a moment to review the content.
6. In the *Summary* tab the *Routing Summary* you will see that the **Continuity Error**. The lower the number, the better. If you see errors in the whole numbers, you may or may not have a problem. If the errors higher, you do have something in the system that needs cleaning up.
7. **Close** the dialog.



Exercise 4: Inspecting How Flows Vary Over Time

Description

In this exercise you will look at how to review the results of the SWMM analysis.

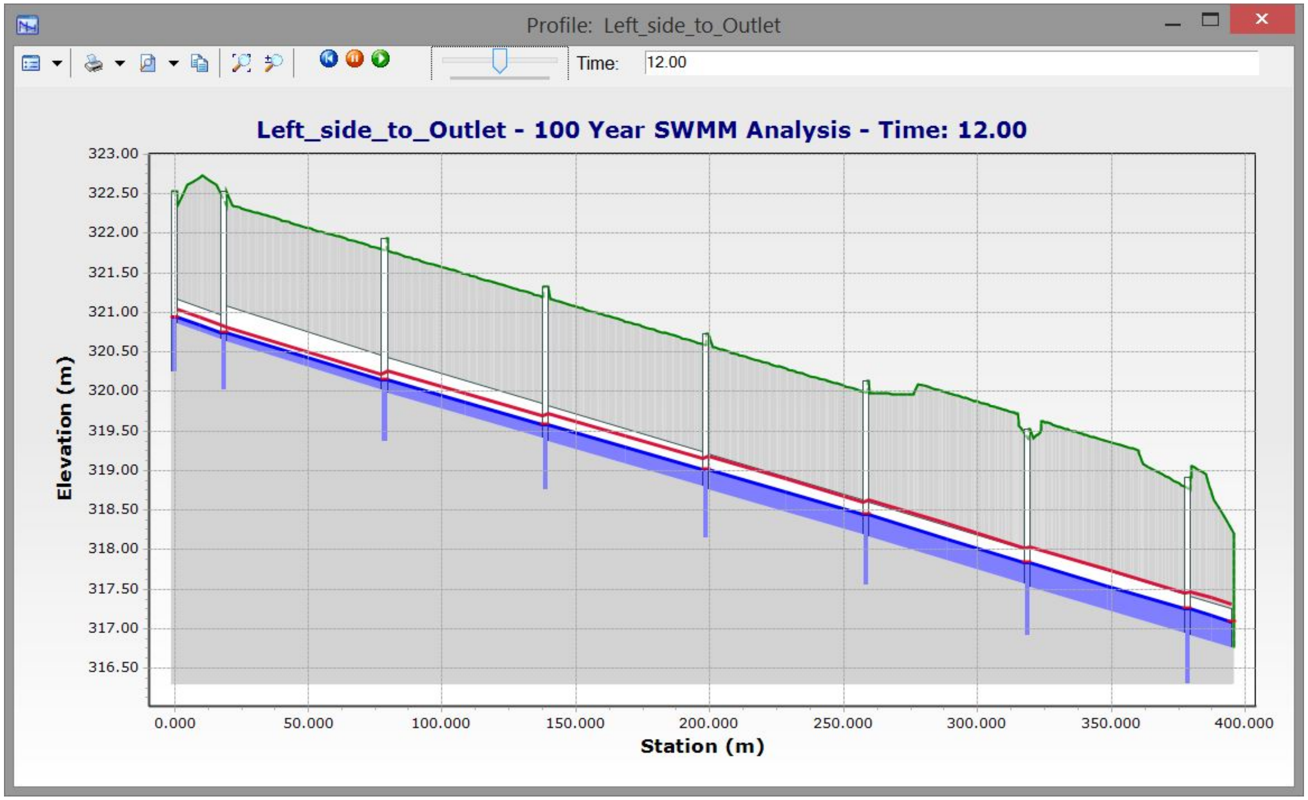
Skills Taught

- Reviewing the Analysis Profile

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Reviewing the Analysis Profile

1. Go to the Utility Model tab of *Project Explorer.* Expand the tree for the DGN, then expand the tree for Profile Runs.
2. Right click on **Left_side_to_Outlet**, and click on **Open Analysis Profile**.
3. Because we have routed runoff hydrographs through the system, and calculated how flow varies over time, we can click the green Play icon at the top of the dialog.



Reviewing the animation shows us that the peak flow occurs at 12 hours.

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted

Follow Up

This completes the process of analyzing a system using the SWMM solver.

For further exploration of the SWMM solver, and its use in designing retention ponds, we have additional calculations classes, including *Retention Ponds – Hydraulics.*

Copyright © 2018 Bentley Systems, Incorporated DO NOT DISTRIBUTE – Printing for student use is permitted