

# **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.



TRNC03084-1/0001

Course Level: Advanced

## **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

## **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.

## Description

This exercise discusses Scenarios, Alternatives and Properties.

## **Skills Taught**

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

# **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

Training and Examples • No Wor	<set< th=""><th>Training and Examples 🔹</th><th>Training-Imperial 🔹</th></set<>	Training and Examples 🔹	Training-Imperial 🔹
Search P * Example Configuration Example Imperial Standards	/se for a file, start by clicking on Browse.	Recent Files You haven't opened any files rece	Training-Imperial
Metric Standards Training and Examples		Browse New F	Training-Metric il
No WorkSpace			
<ul> <li>Create WorkSpace</li> </ul>			
Configuration Migration			Create WorkSet

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

#### 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.

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.

<u>Note:</u> The <u>Implicit (SewerGEMS Dynamic Wave)</u> and <u>Explicit (SWMM Solver)</u> 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.

# **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.

## 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

# **Reviewing the Scenarios**

0

#### 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:

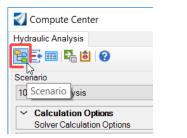
The Scenarios dialog shows the file's Scenarios and their dependencies.

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 **D** to compute the scenario. This ensures that the calculation numbers match the current settings in this file.

4	Compute Center	- 🗆 🗡	<
Hyd	raulic Analysis		
te	📑 🎫 🛃 🙆 😯		
Sce	nario		
100	) Year Analysis		
~	Calculation Options		^
	Solver Calculation Option: Bas	e Analysis	Y
~	<general></general>		
	Active Numerical Solver	GVF-Rational (StormCA	١D
	Calculation Type	Analysis	$\sim$
	Minimum Time of Concentration	n ( 5.000	
$\sim$	Gravity Hydraulics		
	Gravity Friction Method	Manning's	
	Pressure Hydraulics		
$\sim$	Flessule Hyuldulics		



		×
· 🐼 📄	55 ₿	
	r Storm	
esign		
ar Analy	rsis	
	- 10 Yea esign	• 😥 📄 <table-cell-rows> 🖥 - 10 Year Storm esign ear Analysis</table-cell-rows>

# **Reviewing Calculation Options**

High

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.

 Calculation Options

 Solver Calculation Options
 <I> Base Analysis

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

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

Hydraulic Analysis	
🗋 🗙 🖪 🔳 🎯	
E Colver	
- 📶 Base De	sign
🚽 🖬 Base An	alveie

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.

<general></general>		
ID	95	
Label	Base Analysis	
Notes	and and the electronycles on a second second second	
Active Numerical Solver	GVF-Rational (StormCAD)	
Calculation Type	Analysis	3
Minimum Time of Concentration	0.083	

# **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**.

$\sim$	Alternatives	
	Active Topology	<l> Main Network Only</l>
	User Data Extensions	<i>Base User Data Extensions</i>
	Physical	10 Year Design w Larger Pipes
	Boundary Condition	<i>Base Boundary Condition</i>
	Initial Settings	<i>Base Initial Settings</i>
	Hydrology	<l> Base Hydrology</l>
	Output	<l> Base Output</l>
	Infiltration and Inflow	A Base Infiltration and Inflow
	Rainfall Runoff	100 Year Sample City
	water Quality	<i>Base water Quality</i>
	Sanitary Loading	<l> Base Sanitary Loading</l>
	Headloss	<i> Base Headloss</i>

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.

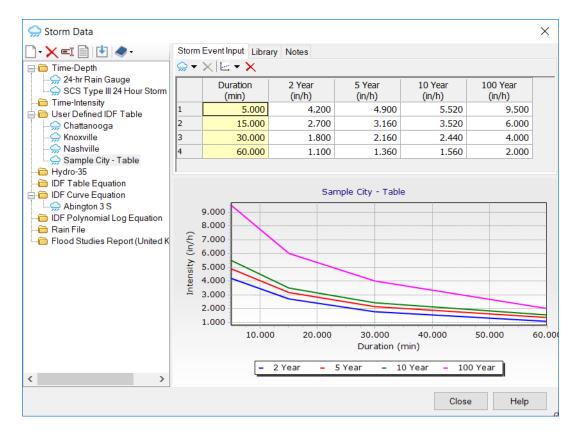
🛞 Glo	obal Storm Ever	nts		×
	Alternative	Global Storm Event	Source	Return Event (years)
12: 1	10-year Sam	Sample City - Table - 10 Year	Sample City - Tabl	10
186:	100 Year Sto	Sample City - Table - 100 Yea ү 💮	Sample City - Tabl	100
<		45		>
				_
			Close	Help

3. Close the Global Storm Events dialog.

#### 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.

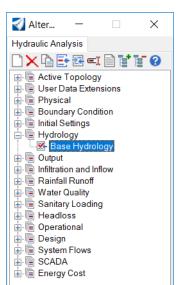


# **Reviewing the Hydrology Alternative**

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

Ş

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.
- E Hydrology : Base Hydrology (\_SWMM-BEGIN-wBadLongSlope -- Default.stsw) Х 🗄 - 🌇 - 🗎 🕜 📰 Catchment 🛛 🖉 Low Impact Development 🔄 Catch Basin Column View ~ Use <All> Outflow \* ID Label Runoff Method Loss Method Scaled Constant Loss R.. Element Area? Green-Ampt \*Groundwater 184: DR- $\checkmark$  $\checkmark$ 184 DR-Rational Method Horton  $\checkmark$  $\checkmark$ 185: DaL-1 185 DaL-1 Rational Method IN-L1 Generic UH Modified Rational  $\checkmark$  $\checkmark$ 186: DaL-2 186 DaL-2 Rational Method IN-L2 Rational  $\checkmark$  $\checkmark$ 187: DaL-3 187 DaL-3 Rational Method IN-L8 RTKUH < SUS UN \* 🗹 = Base data = Local data = Inherited data

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.

## Description

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

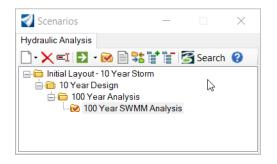
## **Skills Taught**

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

# **Creating a New Scenario**

- 1. In the Scenarios dialog, right-click on 100 Year Analysis. Select New > Child Scenario Scenarios \_ Х Hydraulic Analysis 🗋 • 🗙 🛋 🔁 • 🞯 📄 🚼 🐩 📰 🌌 Search 💡 🖃 🛅 Initial Layout - 10 Year Storm 📄 🫅 10 Year Design 100 Year Analysis Make Current Compute > Validate New Base Scenario Properties Child Scenario Done
- 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:



# **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.

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.

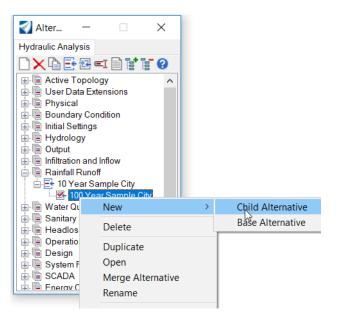
3. Close the Storm Data dialog.

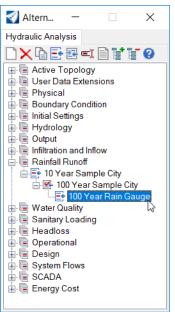
# **Creating a new Rainfall Runoff Alternative**

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**.





- 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.

Duration (Modified Rational): Maximum Storm Intensity:

6. Close the Rainfall Runoff Alternative Editor dialog.

📑 Rainfall Runoff : 100 Year F	Rain Gauge (_SWMM-E	BEGIN-wB —	
🚯 🕶 🖏 💌 🖹 🔞			
Global Rainfall Ø Manhole	Outfall 🛄 Catchment	⊘ Pond ⊘ WetWe	
Alternative:		100 Year Rain Gauge	
Global Storm Event:		24-Hr Rain Gauge (100	year) - 🔀 💷
Source:		24-Hr Rain Gauge - C:\l	N
Return Event:		100	
Intensity (Average):		(N/A)	in/h
Depth:		27.1	in
Duration (Modified Rational):		24.000	hours
Maximum Storm Intensity:		9.500	in/h
* 🔽 = Base data	🗹 🛛 = Local data	= Inherited	data

# **Creating a new Hydrology Alternative**

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

 Alternatives
 —
 ×

 Hydraulic Analysis
 —
 •

 Hydraulic Analysis
 —
 •

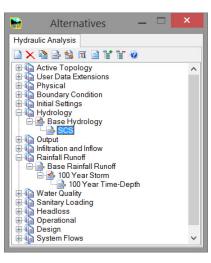
 Hydraulic Analysis
 •
 •

 Hydraulic Analysical
 •
 •

 Boundary Condition
 •
 •

 Hital Settings
 •
 •

 Hydrology
 •
 •
 •

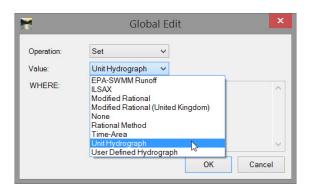


- A new Alternative called Hydrology Alternative 1 is created.
- 2. Click on it, and type the name SCS.

- 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.

Catchment Ø Lo	w Imj	oact Developmer	t Control	Catcl	n Basin								
Column View <all> Constant Loss R</all>	^		*	ID	Label	Runoff Method	Loss Method	Outflow Element	Use Scaled Area?	Area (User Defined)	Time of incentratio	Runoff Coefficient (Rational)	,
Green-Ampt *Groundwater							Global Edit			(ha)	(hours)		
Horton		152: Catchmen	~	152	Catchment-2	Modified Rat	Sort		✓		0.083	0.350	
Generic UH		153: Catchmen	✓	153	Catchment-	Modified Rat	Filter		✓		0.083	0.350	
Modified Rational		168: Catchmen	~	168	Catchment-3	Modified Rat		· · · ·	✓		0.083	0.950	
Rational		169: Catchmen	~	169	Catchment-4	Modified Rat	Reapply Sort/	Filter	~		0.083	0.950	Ī
RTKUH		170: Catchmen	~	170	Catchment-5	Modified Rat	Select Column	n j	~		0.083	0.950	Ť
SCS CN		171: Catchmen	~	171	Catchment-7	Modified Rational		IN-L3	~		0.083	0.950	t
SCS UH		172: Catchmen	~	172	Catchment-6	Modified Rational		IN-R3	~		0.083	0.950	t
SWMM-Runoff Initial Loss and C		173: Catchman			Catchment-8	Modified Dational		TNLD/			580.0		`

- 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.

"⊳+ 📄 🥝													
Catchment 🧭 Low Im	npact Developmen	t Control	Catcl	h Basin									
Column View <all> Constant Loss R</all>		*	ID	Label	Runoff Method	Loss Method	Outflow Element	Use Scaled Area?	Area (User Defined) (ha)	Area Defined By	SCS CN	Subareas	
Green-Ampt *Groundwater	152: Catchmen	✓	152	Catchment-2	Unit Hydr	SCS CN	CB-2	✓		Single Are	0.000	<colle< td=""><td></td></colle<>	
Horton	153: Catchmen	~	153	Catchment-	Unit Hydr	SCS CN	CB-2	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
Generic UH	168: Catchmen	~	168	Catchment-3	Unit Hydr	SCS CN	IN-R1	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
Modified Rational	169: Catchmen	~	169	Catchment-4	Unit Hydr	SCS CN	IN-R2	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
Rational	170: Catchmen	~	170	Catchment-5	Unit Hydr	SCS CN	IN-L2	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
RTKUH	171: Catchmen	~	171	Catchment-7	Unit Hydr	SCS CN	IN-L3	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
	172: Catchmen	~	172	Catchment-6	Unit Hydr	SCS CN	IN-R3	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
SWMM-Runoff	173: Catchmen	~	173	Catchment-8	Unit Hydr	SCS CN	IN-R4	✓		Single Are	0.000	<colle< td=""><td></td></colle<>	
Initial Loss and C	174: Catchmen	~	174	Catchment-9	Unit Hydr	SCS CN	IN-L4	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
Initial Loss and C	175: Catchmen	~	175	Catchment-10	Unit Hydr	SCS CN	IN-R5	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
ILSAX	176: Catchmen	~	176	Catchment-11	Unit Hydr	SCS CN	IN-L5	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
Time-Area	177: Catchmen	~	177	Catchment-12	Unit Hydr	SCS CN	IN-R6	~		Single Are	0.000	<colle< td=""><td></td></colle<>	
	178: Catchmen	~	178	Catchment-13	Unit Hydr	SCS CN	IN-L6	~		Single Are	0.000	<colle< td=""><td></td></colle<>	

10. Find the column titled SCS CN.

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

• "lov 📄 🥝	Impact Developmer	nt Control	Catc	h Basin	Operation Value:		Set	T						
Column View <all> Constant Loss R</all>		*	ID	Label	WHERE:	-	<no acti<="" filter="" td=""><td>ive&gt;</td><td></td><td></td><td>1</td><td>are</td><td></td><td></td></no>	ive>			1	are		
Green-Ampt *Groundwater	152: Catchmen	~	152	Catchment-2									Formatting	
Groundwater Horton	153: Catchmen	~	153	Catchment-								EC	lit 🖓 🚽	
Generic UH	168: Catchmen	~	168	Catchment-:		8					~			
Modified Rational	169: Catchmen	~	169	Catchment-4					0	к	Cancel			
Rational	170: Catchmen	~	170	Catchment-								y S	Sort/Filter	
RTKUH SCS CN	171: Catchmen	~	171	Catchment-7	Unit Hydr	SCS CN	IN-L3	~		Single Are	Se	elect Co	lumn	
SCS UH	172: Catchmen	~	172	Catchment-6	Unit Hydr	SCS CN	IN-R3	~		Single Are	0.000	<colle< td=""><td></td><td></td></colle<>		
SWMM-Runoff	173: Catchmen	~	173	Catchment-8	Unit Hydr	SCS CN	IN-R4	~		Single Are	0.000	<colle< td=""><td></td><td></td></colle<>		
nitial Loss and C	174: Catchmen	~	174	Catchment-9	Unit Hydr	SCS CN	IN-L4	~		Single Are	0.000	<colle< td=""><td></td><td></td></colle<>		
nitial Loss and C	175: Catchmen	~	175	Catchment-10	Unit Hydr	SCS CN	IN-R5	~		Single Are	0.000	<colle< td=""><td></td><td></td></colle<>		
LSAX Time-Area	176: Catchmen	~	176	Catchment-11	Unit Hydr	SCS CN	IN-L5	~		Single Are	0.000	<colle< td=""><td></td><td></td></colle<>		
Time-Area	177: Catchmen	~	177	Catchment-12	Unit Hydr	SCS CN	IN-R6	~		Single Are	0.000	<colle< td=""><td></td><td></td></colle<>		
	178: Catchmen	~	178	Catchment-13	Unit Hydr	SCS CN	IN-L6	~		Single Are	0.000	<colle< td=""><td></td><td></td></colle<>		

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.

4. Double-click on SWMM.

IIO

- A new Calculation Option called New Solver 1 is created.
- 3. Click on New Solver 1, and type a new name of SWMM.

Solver is set to GVF-Rational (StormCAD).

 <General>

 ID
 208

 Label
 SWMM

 Notes
 Active Numerical Solver

 GVF-Rational (StormCAD)
 GVF-Rational (StormCAD)

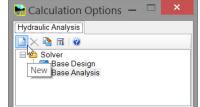
Minimum Time of Concentration (h 0.083

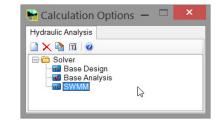
We know that this is a peak flow solver, and we want to calculate runoff hydrographs and route them through the system, using the SWMM solver.

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

5. The properties in the Hydraulic Analysis tab show that the Active Numerical

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**

TIN

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

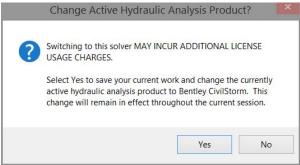
The License Activation group shows the available licenses.

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.

## **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.

<general></general>	
ID	208
Label	SWMM
Notes	
Active Numerical Solver	Explicit (SWMM Solvers)
Simulation Start Date	01/01/2000
Simulation Start Time	00:00:00
Duration Type	User Defined Duration
Duration (hours)	24.000
Routing Time Step (sec)	1.0

🐂 Calculation Options 🗕

📾 Base Design 😼 Base Analysis

22

Hydraulic Analysis

6. Close the Calculation Options dialog.



<General>

# **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:

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

Summary Nodes	Links Catchments Storage Pun	nps Report		Summary Nodes Links Cat	chments Storage Pumps Report			
Run Metrics Routing Summary Runoff Summary	<mark>₽</mark> 2↓							
	✓ <general></general>		^	Analysis Options	Analysis Options			
System Statistics	Label	100 Year SWMM Analysis		Flow Routing Continuity	****			
Process Models Analysis Options	✓ Analysis Options	Hadaa		Highest Continuity Errors	Flow Units	CFS		
	Infiltration Method Flow Routing Method	Horton Dynamic Wave		Time-Step Critical Elements Highest Flow Instability Index	Process Models:			
	Start Date	1/1/2000		Routing Time Step Summary		YES		
	Start Time	12:00:00 AM		Node Depth Summary		NO		
	Stop Date	1/2/2000		Node Inflow Summary Node Surcharge Summary Node Flooding Summary Outfall Loading Summary Link Flow Summary Flow Classification Summary Conduit Surcharge Summary		NO		
		12:00:00 AM				NO		
		24.000				YES		
		0.250			Ponding Allowed			
	Output Increment (min)	15.000			Water Quality			
	Hydrologic Increment (min)	15.000			Flow Routing Method			
	Dry Step (min)	step (min) 60.000 st	Starting Date 01/01/2000 00					
	Route Step (sec) Allows Ponding?	1.0 True	~		Ending Date 01/02/2000 0			
	Flow Routing Method	The			Antecedent Dry Days			
		WMM engine during the simulation.			Report Time Step		>	

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.

Summary	Nodes	Links	Catchments	Storage	Pumps	Report			
Run Metric Routing S			2↓ 🖻						
Runoff Sur		~	<general></general>						
System Statistics			Label 100 Year SW			r SWMM Analys	sis		
Process Models		~	Routing Quantity						
Analysis Options		Dry Weather In	nflow (MG)		0.00				
			Wet Weather I	nflow (MG	)	0.00			
			Groundwater I	nflow (MG	)	0.00			
			RDII Inflow (M	G)		0.00			
		External Inflov	v (MG)		1.66				
		External Outfl	ow (MG)		1.64				
			Overflow Loss			0.02			
			Evaporation L		1	0.00			
		Exfiltration Los			0.00				
			Stored Volume	· · · · ·	1	0.00			
		Stored Volume	× 7 ×		0.00	r			
		Flow Continuit	y Error (%)	)	0.0	T			
			w Continuity netric that defin		I the con	nputed inflows	, outflows, and s	torage value	es
		bal	ance out. The o	closer the v	alue is to	zero the bette	er.	2	

## Description

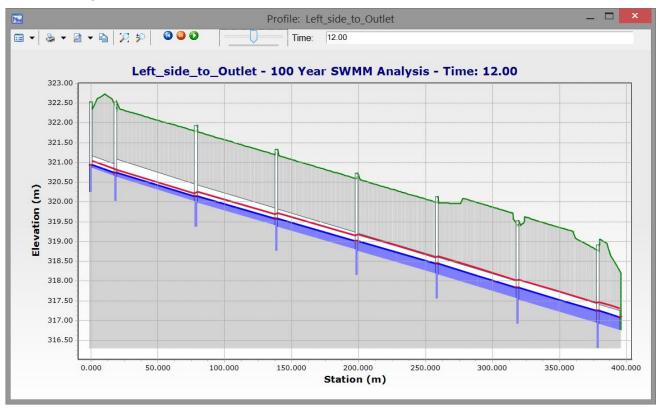
In this exercise you will look at how to review the results of the SWMM analysis.

## **Skills Taught**

• Reviewing the Analysis Profile

## **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.

# **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.*