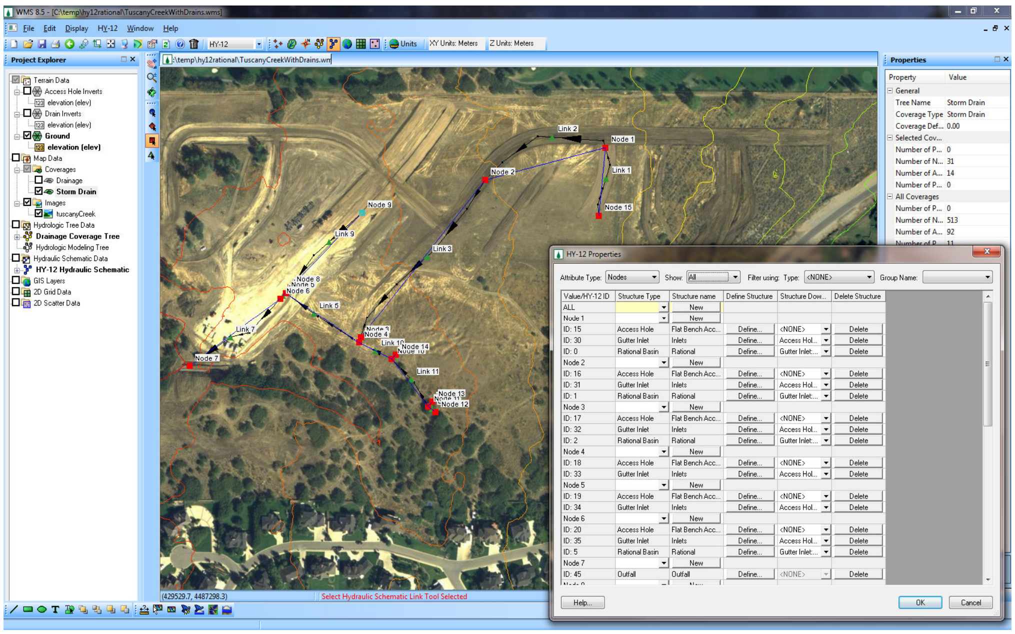
WMS

v. 9.1

WMS 9.1 Tutorial

Storm Drain Modeling – HY-12 Rational Design

Learn how to design storm drain inlets, pipes, and other components of a storm drain system using FHWA's HY-12 storm drain analysis software and the WMS interface



Objectives

Define a storm drain network and its associated data. Compute the storm drain data using WMS tools that assign elevations, slopes, lengths, and hydrologic parameters to the HY-12 storm drain model data. Then run the HY-12 model and view the results.

Prerequisite Tutorials

- Watershed Modeling – Advanced DEM Delineation Techniques
- Editing Elevations – Using TINs

Required Components

- Data
- Drainage
- Map
- Hydrology
- Hydrologic Models
- Storm Drain

Time

- 40-50 minutes

AQUAVEO™

1 Contents

2 Introduction

The US Federal Highway Administration's HY-12 is a DOS-based storm drain analysis program that can be used for designing inlets, pipes, and the general layout of a storm drain network. An HY-12 model can be generated by drawing the proposed pipe and inlet locations in a storm drain coverage. Then the map module locations are converted to a 1D schematic where the HY-12 model parameters are defined. Many of the HY-12 computations, such as channel calculations, curb and gutter calculations, and rational method computations, are based on computations in FHWA's Hydraulic Toolbox software. You can refer to the documentation in both the Hydraulic Toolbox and in HY-12 to learn about the specific computation methods used in HY-12.

This tutorial shows you how to model and design a storm drain network in a small suburban sub-basin. You learn how to assign pipe invert, access hole, ground, and inlet elevations to your model. You also learn how to associate computed hydrologic model data with rational method data so it can be used in HY-12. It is recommended that you be familiar with some of the more advanced watershed modeling techniques in WMS by following the advanced watershed modeling tutorial before attempting this one.

3 Objectives

Your task in this workshop is to design the storm drain network for Tuscany Creek, a small suburban development. In this exercise, you will learn how to use the WMS HY-12 interface to design a storm drain system in this suburban area. The following tasks will be demonstrated in this workshop:

- Reading an existing hydrologic model for a suburban area.
- Define a storm drain network.
- Define HY-12 structures in the storm drain network and assign parameters to the structures.
- Assign pipe invert, access hole, ground, and inlet elevations to your HY-12 model.
- Compute hydrologic model data and associate the computed hydrologic data with the HY-12 rational method computations.
- Run HY-12.
- View the HY-12 results.

4 Read and Convert Data

4.1 Read a Hydrologic Model

The first step in defining a storm drain simulation is to build or read an existing hydrologic model. Other tutorials describe the advanced watershed modeling process involved in defining a storm drain simulation for an urban or suburban watershed. First, you will read an existing hydrologic model.

1. From the File menu, select Open. Locate the HY12Rational folder in your tutorial files.
2. Find and open TuscanyCreek.wms. This is a hydrologic model that has already been defined. Notice that the areas and other geometric parameters have already been computed for this watershed.

4.2 Read and Define Storm Drain Network

In this section, you will read a shapefile defining the storm drain network and convert it to a schematic that can be used to define HY-12 link and node data.

1. Select File | Open again and open StormDrains.shp. This is an ArcInfo shapefile containing proposed storm drain locations in this watershed.

The storm drains have been defined using a shapefile. You need to convert these storm drains to pipes in a "Storm Drain" coverage in the map module to define an HY-12 schematic.

2. Right-click on the Coverages folder in the Project Explorer and select New Coverage.
3. Change the Coverage type to Storm Drain and select the OK button.

4. Select the StormDrains.shp file in the project explorer.
5. Select ***Mapping | Shapes -> Feature Objects.***
6. Select Yes to use all shapes in all the visible shapefiles for mapping.
7. Select *Next.*
8. Select *Next.*
9. Select *Finish.*
10. Hide the *StormDrains.shp* file, the Drainage coverage, the DEM, and the tuscanyCreek image by deselecting their check boxes in the Project Explorer.
11. Right-click on the Storm Drain coverage in the Project Explorer and select Zoom To Layer.
12. In the Storm Drain menu, select Map → 1D Schematic.
13. Change the model type to HY-12 and select OK.
14. You get a message saying that several links were assigned elevations using the DEM. Select OK to this message.

Your hydraulic schematic is now created. If you need to manually create arcs in a storm drain coverage defining your storm drain pipe network, you would create these arcs from upstream to downstream. The direction arrows on the storm drain arcs should be pointing downstream.

5 Define HY-12 Project Parameters

After you have created your hydraulic schematic, the next step is to define your project parameters. These parameters are global parameters that are used in your entire project.

1. Select the HY-12 Hydraulic Schematic in the Project Explorer.
2. Select HY-12 | Edit Project Parameters….
3. Enter "Tuscany Creek" for the project name.
4. Enter your name as the project designer.
5. Next to the material database, select the Select File… button.
6. Browse to and open the materialDB.txt file in your tutorial file folder.
7. Select the following options in the project parameters dialog:
    - HY12 Unit System: English Units
    - Error Reporting: Report Errors, Warnings, and Notices
    - HY12 Calculate Geometry: Specify length, angle, and elevations, compute Slope
    - Analyze or Design? Design: Size Pipes Only
    - Method to Design for Surcharge: Compare the HGL to the Surface Elevation
    - Freeboard in Design for Surcharge: 1.0 ft
    - Drop Allowed in an Access Hole: 1.0 ft

- Steady or Unsteady Flow: Steady Flow
- Use one IDF for Entire Project: ON
- Ignore Gutter Inlets: OFF
- Assume Gutter Inlets Capture All Flow: ON

8. Select Define… in the Project IDF column to define your IDF curve.
9. Select User Supplied Data, then the Define Storm Data… button.
10. Change the Recurrence Interval to 10 years.
11. Enter the following data for the 10-year storm (from NOAA Atlas 14) and select OK:

Duration (min)	Intensity (in/hr)
5	3.49
10	2.66
15	2.20
30	1.48
60	0.92

12. Select the 10-year recurrence interval line in the IDF curve table and select Done.
13. Select OK to close the HY-12 Properties dialog.

6 Define HY-12 Structures

The next step is to define HY-12 "structures" (inlets, access holes, pipes, etc.) at each of the links and nodes in your schematic. A structure represents a hydraulic or hydrologic computation that requires input and provides output. Some of the output, such as a discharge flow or hydrograph from a rational method computation, may be used in a structure located downstream in your model. One or more structures are defined at each link or node. Structures that *cannot* be represented by a line, such as an access hole, an inlet, or a rational method sub-basin, are defined at nodes. Structures that *can* be represented by a line, such as a pipe or a gutter, are defined at links. The following types of structures can be created at a node:

- Access Hole
- Gutter Inlet
- Junction
- Minor Loss
- Outfall
- Rational Method Basin
- Reservoir

- Transition

The following types of structures can be created at a link:

- Channel
- Gutter
- Pipe
- Pipe Storage

For more information about each of these structures, their computations, and their file formats, refer to the FHWA HY-12 and Hydraulic Toolbox documentation.

6.1 Define Access Holes

In this step, you will define access holes for each of your pipe junctions. Since most nodes in your model have access holes, you can define access hole data for all your nodes and then delete the access holes from nodes that do not have access holes. You will use this same approach to define other structures at your nodes.

1. Select the HY-12 Hydraulic Schematic in the Project Explorer.
2. Double-click on Node 15 to bring up the HY-12 Properties for this node.
3. In the HY-12 Properties dialog, change the "Show" field from Selected to All.
4. In the ALL row of the spreadsheet, change the Structure Type to Access Hole and select the New button (in the ALL row). Notice that access holes are created for all the nodes.
5. In the ALL row of the spreadsheet, select the Define Structure | Define… button.
6. Enter "3 Foot Manhole" for the Name and enter a width of 3.0 ft. Select OK to save the changes to all the nodes.
7. Node 7 is the outfall location. Select the Delete button for the Access Hole structure assigned to this node to delete this access hole. Node 6 is a connection between a pipe and channel. Delete the Access Hole at node 6.

6.2 Define Gutter Inlets

You will define gutter inlets in the same way as access holes–by assigning them to all the nodes, defining common data for all the inlets, and then deleting the inlets where they are not needed. There is an option at the top of the HY-12 Properties dialog to filter the structures that are displayed so only the selected structure types (such as Access Hole or Gutter Inlet) are displayed at each node.

1. In the HY-12 Properties dialog, change the "Filter using: Type" field from "<NONE>" to Gutter Inlet. Notice that the Access holes are no longer displayed.
2. In the ALL row of the spreadsheet, change the Structure Type to Gutter Inlet and select the New button. Notice that the Access Holes defined in a previous step are automatically assigned as the downstream structure from each of the inlets.
3. In the ALL row of the spreadsheet, select the Define Structure | Define… button.

4. Enter "2x4 Inlet" for the Name. Turn off the option to define gutter parameters since you will be assuming full capture at all the inlets. Select the Curb and Gutter Calculator button.
5. In the Curb and Gutter Analysis dialog, enter the following parameters:

Grate width: 2.0 ft

Grate length: 4.0 ft

Leave all other parameters at their default values.

6. Select OK on the Curb and Gutter Analysis dialog.
7. Enter a curb height of 0.75 ft.
8. Select OK to save the Gutter inlet changes to all the nodes.
9. Since Node 7 is the outfall location, select the Delete button for the Gutter Inlet structure assigned to this node. There is no inlet at the outfall location. There are also several other nodes that have an access hole but no gutter inlet. Select the Delete button to delete the Gutter Inlets at the following nodes: Node 4, Node 5, Node 6, Node 10, and Node 11.

6.3 Define Rational Basins

In this simulation, you will use the rational method to compute the discharge and assume full capture at each of the inlet locations. A hydrographic simulation could be run that considers the entire hydrograph from the Rational Method simulations.

The rational method requires a runoff coefficient, a rainfall intensity, and an area for each sub-basin in the watershed. In WMS, a composite runoff coefficient is computed using a runoff coefficient coverage. You can compute the intensity by defining storm intensities for a certain recurrence interval for several storm durations (an Intensity-Duration-Frequency curve). The time of concentration for each sub-basin determines the intensity from this curve. The area for each sub-basin is computed when the watershed is delineated and its geometric data is computed. Since this is a steady-state simulation, only the peak flow from the Rational Method is used. You compute and assign these hydrologic and geometric parameters for the Rational Method computations later in this tutorial. You only assign the rational method computations to each node here.

1. In the HY-12 Properties dialog, change the "Filter using: Type" field to Rational Method Basin.
2. In the ALL row of the spreadsheet, change the Structure Type to Rational Method Basin and select the New button. Notice that the Gutter Inlets defined in a previous step are automatically assigned as the downstream structure from each of the Rational Basin computations (some rational basins have access holes as the downstream computations…these will be deleted later since these access holes are not associated with basins).
3. In the ALL row of the spreadsheet, select the Define Structure | Define… button.
4. Enter "Rational" for the Name. You will assign the remaining rational method parameters later in this tutorial.
5. Select OK to save the Rational Basin properties to all the nodes.

6. There are several nodes that will not be associated with any sub-basin. You need to delete the Rational Basin structures at these nodes. Select the Delete button to delete the Rational Basins at the following nodes: Node 4, Node 5, Node 6, Node 7, Node 10, and Node 11.

6.4 Define Outfall

You will define the outfall of the storm drain network in this section.

1. In the HY-12 Properties dialog, change the "Filter using: Type" field to Outfall.
2. Change the structure type to Outfall for Node 7 and select the New button for this structure.
3. In the row containing the outfall structure just added, select the Define Structure | Define… button.
4. Change the name of the structure to "Primary Outfall" and select the OK button to save the outfall properties.

6.5 Define Pipes

Now that all the point-based structures are defined at the nodes, you can define the arc-based structures and their parameters. The line-based structures include the pipes, channels, and gutters in the storm drain network.

1. In the HY-12 Properties dialog, change the "Attribute Type" field to Links.
2. In the ALL row of the spreadsheet, change the Structure Type to Pipe and select the New button (in the ALL row). Notice that pipes are created for all the nodes.
3. In the ALL row of the spreadsheet, select the Define Structure | Define… button.
4. Enter "12 Inch Pipe" for the Name. Select a “Circular” Shape Type and verify that the selected shape has a diameter of 1.0 ft. Enter a Manning's n of 0.012. Select OK to save the changes to all the pipes. The other pipe parameters will be computed later; the thickness is computed as the diameter/12.0.
5. Link 7 is a channel instead of a pipe. Select the Delete button for the Pipe structure assigned to this link to delete this pipe.

6.6 Define Channels

A channel will be constructed that runs from the last sub-basin inlet (Node 6) to the outfall (Node 7). This is labeled as Link 7 in the project. The final step in defining the HY-12 structures is to define this channel.

1. In the HY-12 Properties dialog, change the "Filter using: Type" field to Channel.
2. Change the structure type to Channel for Link 7 and select the New button for this structure.
3. In the row containing the channel structure just added, select the Define Structure | Define… button.
4. Change the name of the structure to "Outfall Channel" and select the Channel Calculator button.

5. Enter the following data for the channel analysis:

    Size slope 1 (Z1): 2.0 H:1V

    Size slope 2 (Z2): 2.0 H:1V

    Channel width (B): 5.0 ft

    Longitudinal slope: 0.09 ft/ft

    Manning's roughness: 0.06 (we will install large-diameter riprap encased in a gabion mattress-a rock-filled wire container)

6. Select OK to close the Channel Analysis dialog.
7. Enter a Manning's roughness of 0.06.
8. Select the OK button to save the channel properties.
9. Select OK to close the HY-12 Properties dialog.

7 Compute and Assign Data to HY-12 Structures

Now that you have set up some of the parameters for each of the structures, you can use some of the geometric and hydrologic calculators available in WMS to assign elevations, areas, and hydrologic parameters to each of the HY-12 structures.

7.1 Assign Surface and Inlet Elevations

1. Select File | Open again and open Elevations.tin. This is a set of 3 TINs that will be used to define the ground elevations, the access hole invert elevations, and the storm drain invert elevations for this model.
2. Select the TIN labeled "Ground" in the project explorer to make it active. Then select the HY-12 Hydraulic Schematic in the project explorer to activate the HY-12 menu.
3. Select the HY-12 | Assign Elevations | To Ground menu command.
4. In the Select Elevation Source dialog that appears, select Ground as the elevation source and select the OK button.
5. You should get an information message saying that several nodes were assigned elevations using the active TIN. Select OK to this message.

7.2 Assign Storm Drain Invert Elevations

1. Select the HY-12 | Assign Elevations | To Inlets menu command.
2. In the Select Elevation Source dialog that appears, select Ground as the elevation source and select the OK button.
3. You should get an information message saying that several nodes were assigned elevations using the active TIN. Select OK to this message.

7.3 Assign Access Hole Invert Elevations

1. Select the TIN labeled "Access Hole Inverts" in the project explorer to make it active. Then select the HY-12 Hydraulic Schematic in the project explorer to activate the HY-12 menu.
2. Select the HY-12 | Assign Elevations | To Access Holes menu command.
3. In the Select Elevation Source dialog that appears, select Access Hole Inverts as the elevation source and select the OK button.
4. You should get an information message saying that several nodes were assigned elevations using the active TIN. Select OK to this message.

7.4 Define Outfall Elevation

1. Select the HY-12 | Assign Elevations | To Outfalls menu command.
2. In the Select Elevation Source dialog that appears, select Access Hole Inverts as the elevation source and select the OK button.
3. You should get an information message saying that Node 7 (the outfall) was assigned elevations using the active TIN. Select OK to this message.

7.5 Assign Channel Elevations

1. Select the TIN labeled "Drain Inverts" in the project explorer to make it active. Then select the HY-12 Hydraulic Schematic in the project explorer to activate the HY-12 menu.
2. Select the HY-12 | Assign Elevations | To Channels menu command.
3. In the Select Elevation Source dialog that appears, select Drain Inverts as the elevation source and select the OK button.
4. You should get an information message saying that Link 7 (the outfall channel) was assigned elevations using the active TIN. Select OK to this message.

7.6 Assign Pipe Elevations

1. Select the HY-12 | Assign Elevations | To Pipes menu command.
2. In the Select Elevation Source dialog that appears, select Drain Inverts as the elevation source and select the OK button.
3. You should get an information message saying that several links were assigned elevations using the active TIN. Select OK to this message.

7.7 Assign Pipe Lengths and Orientations

In this section, assign the lengths to the pipes and the channel and the orientation, or inlet angle, to the pipes.

1. Select the HY-12 | Assign Lengths and Orientations menu command.

2. You should get an information message saying that several links were assigned lengths, then another message saying that several links were assigned orientation. Select OK to both messages.

7.8 Assign Areas to Rational Basins

1. Select the HY-12 | Link Outlets to Inlets… menu command.
2. In the Link Storm Drain and Drainage Nodes dialog, enter a tolerance of 5.0 (feet) and select the Auto Link button.
3. Notice that all the nodes are assigned outlet points except the 4 nodes located at pipe junctions and not associated with an inlet. Select OK to close the link Outlets to Inlets dialog.
4. Select the HY-12 | Assign Hydraulic Data | Areas command. This command assigns the computed basin areas upstream from the outlet points you have linked to the rational method computations at each node.
5. You should get an information message saying that several Rational Method structures at nodes were assigned areas using the linked outlet points. Select OK to this message.

7.9 Compute Runoff Coefficients and Assign to Rational Basins

In this section, you will read a polygon shapefile defining runoff coefficients in your watershed and compute runoff coefficients from the runoff coefficient polygons.

1. Select File | Open again and open RunoffCoefficients.shp. This is an ArcInfo shapefile of polygons defining runoff coefficients in the area that overlays your storm drain and watershed models.

You need to convert this shapefile data to polygons in a "Runoff Coefficient" coverage in the map module to compute composite runoff coefficients for each of your sub-basins.

2. Right-click on the Coverages folder in the Project Explorer and select New Coverage.
3. Change the Coverage type to Runoff Coefficient and select the OK button.
4. Select the RunoffCoefficients.shp file in the project explorer.
5. Select ***Mapping | Shapes -> Feature Objects.***
6. Select Yes to use all shapes in all the visible shapefiles for mapping.
7. Select *Next*. Notice that the RUNOFFC field is mapped to various Runoff Coefficient values.
8. Select *Next.*
9. Select *Finish.*
10. Hide the *RunoffCoefficients.shp* file and the Runoff Coefficient coverage in the Project Explorer. Show the Drainage coverage and select this coverage so this is the active coverage.
11. Select the Drainage Coverage Tree in the project explorer to activate the hydrologic calculators menu. Then select Calculators | Compute GIS Attributes.

12. In the Compute GIS Attributes dialog, change the computation type to Runoff coefficients and select OK. Notice that the computed runoff coefficients are displayed.
13. Select the HY-12 Hydraulic Schematic in the project explorer to activate the HY-12 menu.
14. Select HY-12 | Assign Hydrologic Data | Runoff Coefficients. This command area-weights the runoff coefficients for the sub-basins assigned to each outlet and assigns the area-weighted runoff coefficient to the rational method computations at each node.
15. You should get an information message saying that several Rational Method structures at nodes were assigned runoff coefficients using the linked outlet points. Select OK to this message.

7.10 Compute Time of Concentrations and Assign to Rational Basins

1. Select the Drainage Coverage Tree in the project explorer to activate the hydrologic calculators menu. Then select Rational | Run Simulation.
2. In the Rational Method dialog, change the Display Type field to Basins and the Show field to All.
3. Compute the Tc using the Basin Data method for each sub-basin by selecting the Compute… button in the 'Compute TC – Basin Data' row, selecting the Kirpich Method for overland flow on bare earth method (this should be the default method), and selecting the OK button for each of the basins.
4. Select OK to save your changes and close the Rational Method dialog.
5. Select the HY-12 Hydraulic Schematic in the project explorer to activate the HY-12 menu.
6. Select HY-12 | Assign Hydrologic Data | Time of Concentration. This command computes the average time of concentration for the sub-basins assigned to each outlet and assigns this time of concentration to the rational method computations at each node.
7. You should get an information message saying that several Rational Method structures at nodes were assigned times of concentration using the linked outlet points. Select OK to this message.
8. Select the Storm Drain coverage in the project explorer to make it the active coverage. Turn off the display of the Drainage coverage. Select the HY-12 Hydraulic Schematic in the project explorer to activate the HY-12 menu.

8 Run HY-12

You have now finished setting up your HY-12 model. To run the model, do the following steps:

1. Select HY-12 | Run HY-12.
2. If prompted, find and select the hy12 executable.

3. Verify that the material database exists and was read correctly. Verify that the remaining file locations will work on your system.
4. Select *Run Simulation*.
5. Once the model stops running, you will be prompted to select a program to view a log file of the HY-12 run and the HY-12 report file. The report can be customized by changing the material database.
6. Review the HY-12 log and report file and close or minimize the files when you are done with them. If you need to view any of the files used to run HY-12 or written from HY-12, you can open these files from the Run HY12 Simulation dialog.
7. Close the Run HY12 Simulation dialog.

9 View HY-12 Output

Whether or not the model run was successful, HY-12 generates a report file. If the run was successful, WMS reads the results, which include the energy and hydraulic grade line (EGL, HGL) elevations at each node in the model. For hydrographic simulations, HY-12 computes a hydrograph at each node in the model and you can view a plot of the EGL or the HGL for a node at each time step in your model. Both of these results are read into WMS after an HY-12 run is completed. This section will show how you can view the results in the HY-12 output file and graphically in WMS.

9.1 View Detailed Output

1. Select the Frame Image button.
2. Select a node 1.
3. In the HY-12 menu, select the View Detailed Link/Node Output… menu command.
4. If prompted, select a text editor and select the OK button. A text file describing the detailed link and node computation results appears.
5. Review the HY-12 output file and close or minimize the file when you are done with it.

9.2 View HGL and EGL Plots

You can view plots of HGL and EGL by selecting one or more links in your model. The selected links cannot branch. The plot will show all pipes and access holes between the selected links and nodes.

1. Select nodes 1 and 15 in your model.
2. Select the HY-12 | View EGL and HGL Plots… menu command.
3. In the HGL and EGL Profiles dialog, observe the HGL, EGL, and ground surface elevation plots. Select OK to close the HGL and EGL Profiles dialog.

You have now completed the HY-12 tutorial. This tutorial has shown you how to build an HY-12 model, link the model to hydrologic parameters, run HY-12, and view the results of the HY-12 run.