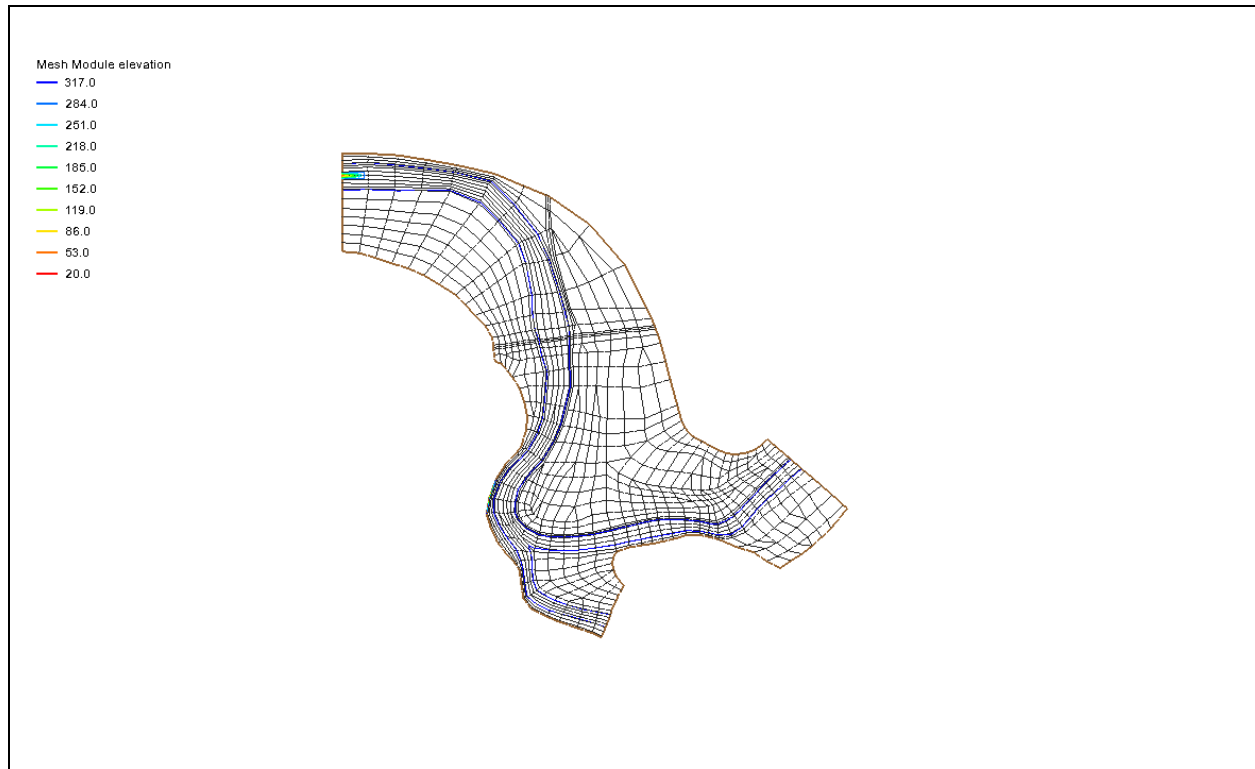


SMS 12.2 Tutorial

Generic 2D Mesh



Objectives

Learn how to create and use a generic model interface for models executed outside of SMS. Learn how to create a master generic model interface and how to use a previously saved generic model interface.

Prerequisites

- None

Requirements

- Map Module
- Mesh Module

Time



- 60–90 minutes

1	Getting Started	2
2	Specifying Model Units	3
3	Defining the Model Interface.....	3
	3.1 Global Parameters.....	3
	3.2 Boundary Conditions	6
	3.3 Material Properties.....	8
4	Protecting and Saving the Model Definition	9
5	Assigning Model Parameters.....	10
6	Assigning Boundary Conditions.....	11
	6.1 Creating Nodestrings	11
	6.2 Assigning Boundary Conditions	12
	6.3 Correlation and Activation Benefits of Boundary Conditions	14
	6.4 Boundary Condition Display Options.....	14
	6.5 Dynamic Boundary Conditions.....	15
	6.6 Deleting Boundary Conditions	16
7	Assigning Material Properties.....	16
8	Multiple Material Assignments	17
9	Dependencies.....	18
10	Gen2DM Model Check.....	18
11	Exporting the Gen2DM File and Running the Model	19
12	Conclusion.....	19



1 Getting Started

The instructions in this tutorial are provided as examples of how the various parameters can be set. They are provided as an example only, and the parameters used here may not be used—or used in this way—in every instance. Each use of the generic model will be different depending on the needs of the particular model executable to be used.

To start the creation of the master interface:

1. Using the **Create Feature Arc**  tool, click out a simple polygon. The number of nodes, number of arcs, shape, and size is not important for this exercise.
2. Select *Feature Objects* / **Build Polygons**.
3. Right-click on “ Area Property” and select *Type | Models* / **Generic Model**.
4. Right-click on the coverage again and select *Convert | Map* → **2D Mesh** to bring up the *2D Mesh Options* dialog.
5. Click **OK** to close the *2D Mesh Options* dialog, create a simple mesh, and bring up the *Mesh Name* dialog.

This enables the generic mesh model parameters that need to be set.

6. Click **OK** to accept the default name close the *Mesh Name* dialog.
7. Select “ Area Property Mesh” to make the  **Mesh Module** active.
8. Select *Data* | **Switch Current Model...** to bring up the *Select Current Model* dialog.
9. Select *Generic Model* and click **OK** to close the *Select Current Model* dialog.

2 Specifying Model Units

The units should be set appropriately for the model executable to be used. For this tutorial, set the units by doing the following:

1. Select *Display* | **Projection...** to bring up the *Display Projection* dialog.
2. In the *Horizontal* section, select *No projection* and select “Feet (U.S. Survey)” from the *Units* drop-down.
3. In the *Vertical* section, select “Feet (U.S. Survey)” from the *Units* drop-down.
4. Click **OK** to exit the *Display Projection* dialog.

3 Defining the Model Interface

Model interface parameters define various states and characteristics of a model. These model parameters may include those describing flow, channel roughness, and control structures.

Depending on the intentions and capabilities of the outside executable, parameters will be organized into groups and given suitable value ranges. Proper organization of parameters will increase the abilities of SMS as an interface, especially for executables that are designed for multiple simulation possibilities.

To begin defining the model interface for this example, do the following:

1. Select *Mesh* | **Define Model** to bring up the *Define Model* dialog.
2. Click **Model Parameters...** to bring up the *Mesh Model Parameters* dialog.
3. In the *Model Information* section, enter “Gen2DM” as the *Name*.

This renames the “ Area Property Mesh” to “ Gen2DM”.

4. Enter “minutes” as the *Time units* to be used in this example.

The time units can be anything appropriate to the model executable being used.

5. Click **OK** to close the *Mesh Model Parameters* dialog.

The name and time units will not be used in SMS, but it will be written to the interface file for reference. For the generic model interface, SMS uses the name entered here to identify related files.

Do not close the *Define Model* dialog at this time, as it is used in the following sections.

3.1 Global Parameters

To define the model parameters for this example, first create a parameter group:

1. Click **Global Parameters...** to bring up the *Gen2DM Global Parameters* dialog.
2. In the *Parameter Groups* section, enter “Hydrodynamic” in the blank field in the *Name* column.

3. On the same row, click **Define...** in the Parameters column to bring up the *Hydrodynamic Parameter Definition* dialog.
4. In the *Parameters* section, enter the first hydrodynamic parameter in the first row by entering “Time interval” in the *Name* column of the spreadsheet.
5. Press the *Tab* key to go to the drop-down in the *Type* column
6. Select “integer” from the drop-down in the *Type* column and press *Tab* to go to the *Default* column.
7. Enter “20” and press *Tab* to go to the *Minimum* column.

Remember that the Time units were set to “minutes” in the previous section. The model in this example accepts only positive time intervals, so this needs to be defined here.

8. Enter “0.0” in the *Minimum* column and leave the *Maximum* column blank to indicate no upper limit on the time.
9. Use the table below to enter the next three parameters for this example. Ignore the *Dependencies* column for this example:

Name	Type	Default	Minimum	Maximum
Velocity max (ft/sec)	float	75.0	0.0	100.0
Check for dry elements	boolean	(check the box)		
Element style	text	quadratic		

10. For the fifth parameter, enter “Critical scour velocity” as the *Name*, select “options” as the *Type*, and click the **Define opts...** button in the *Minimum* column to bring up the *Critical scour velocity Option Definition* dialog (Figure 1).

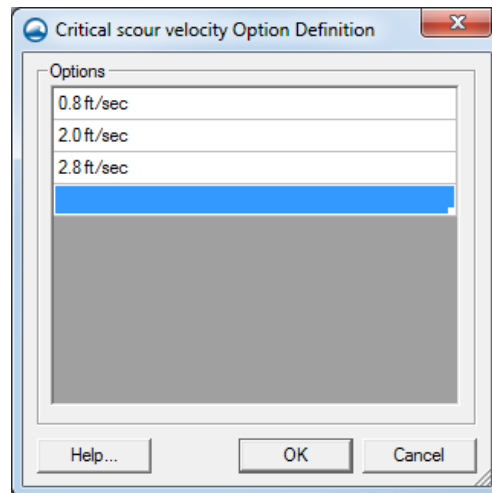


Figure 1 Critical scour velocity Option Definition dialog

11. Enter “0.8 ft/sec”, press *Tab*, enter “2.0 ft/sec”, press *Tab*, and enter “2.8 ft/sec”.
12. Click **OK** to close the *Critical scour velocity Option Definition* dialog.
13. Select “2.0 ft/sec” from the drop-down in the *Default* column.

14. For the sixth parameter, enter “Friction type” as the *Name*, select “options” as the *Type*, and click the **Define opts...** button in the *Minimum* column to bring up the *Friction type Option Definition* dialog.
15. Enter “Manning”, press *Tab*, and enter “Chezy”.
16. Click **OK** to close the *Friction type Option Definition* dialog.
17. Select “Manning” from the drop-down in the *Default* column.

The *Hydrodynamic Parameter Definition* dialog should resemble Figure 2.

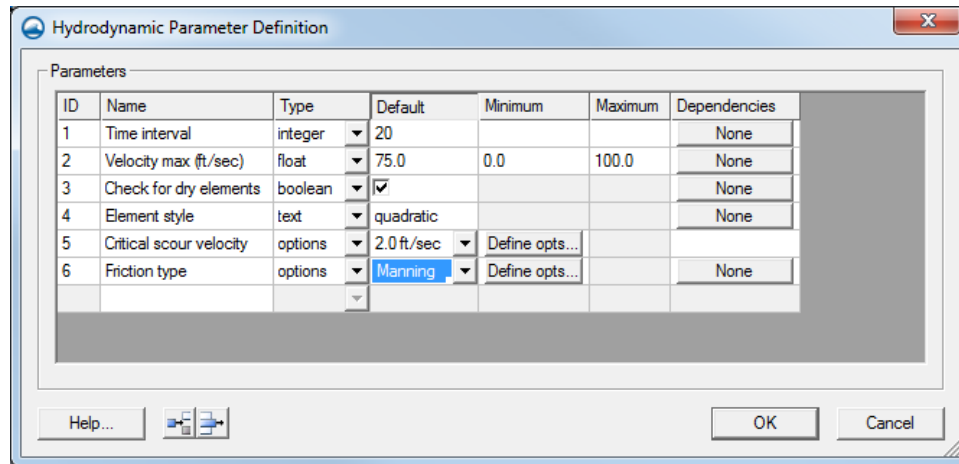


Figure 2 The completed *Hydrodynamic Parameter Definition* dialog

18. Click **OK** to close the *Hydrodynamic Parameter Definition* dialog.

Note that each name in a spreadsheet must be unique. Any line in a spreadsheet can be deleted by highlighting the name and pressing the *Delete* key.

19. Enter “Sediment transport” in the blank field below “Hydrodynamic” and click **Define...** to bring up the *Sediment transport Parameter Definition* dialog.
20. In the *Sediment transport Parameter Definition* dialog, use the information below to enter the parameter information for this example
 - *Name*: “Time interval”, *Type*: “integer”, *Default*: “10. 0”
 - *Name*: “Source X position”, *Type*: “float”, *Default*: “0.0”, no range
 - *Name*: “Source Y position”, *Type*: “float”, *Default*: “0.0”, no range
 - *Name*: “Source elevation”, *Type*: “float”, *Default*: “0.0”, no range
 - *Name*: “Parcel mass (slug)”, *Type*: “float”, *Default*: “0.5”, *Minimum*: “0.0001”, *Maximum*: leave blank
 - *Name*: “Particle mass (slug)”, *Type*: “float”, *Default*: “0.003”, *Minimum*: “0.0001”, *Maximum*: leave blank
 - *Name*: “Particle size (in)”, *Type*: “float”, *Default*: “0.05”, *Minimum*: “0.0”, *Maximum*: leave blank
 - *Name*: “Deviation”, *Type*: “float”, *Default*: “0.0”, no range

- Name: “Average density (slug/ft³)”, Type: “float”, Default: “3.0”, Minimum: “1.5”, Maximum: “6.0”
21. When done, click **OK** to exit the *Sediment transport Parameter Definition* dialog.
 22. There should now be two complete parameter groups defined for later use. Click **OK** to close the *Gen2DM Global Parameters* dialog and to save all data appertaining to global parameters of the model interface.

3.2 Boundary Conditions

Gen2DM (the SMS interface) allows boundary conditions to be specified for three entities: nodes, nodestrings, and elements (linear and quadratic triangles and quadrilaterals). The boundary conditions may be defined for general use, or correlated with a particular parameter group and hence its availability limited. To define a boundary condition:

1. In the *Define Model* dialog, click the **Boundary Conditions...** definition button. The *Gen2Dm Boundary Conditions* dialog should appear.
2. On the *Node* tab enter the *Name* “Water sink/source” and then press *Tab* on the keyboard.
3. Leave *Legal on interior* checked (This refers to whether this condition can be assigned within the mesh, in addition to along the mesh boundary).
4. Click the **Define** button to enter the *Water sink/source Definition* spreadsheet.
5. Add the following values:
 - Name: “Flow rate (cfs)”, Type: “float”, Default: “0.0”, Minimum: “0”, Maximum: leave blank
 - Name: “Water temperature (F)”, Type: “float”, Default: “65.0”, Minimum: “32.5”, Maximum: “100”
6. Click **OK** to save and exit the spreadsheet.
7. Continue adding the following boundary conditions and their parameters and limitations under the *Node* tab:
 - Name: “Ceiling (pressure flow)”, *Legal on interior*: no, *Values*:
 - Name: “Ceiling (ft above sea level)”, Type: “float”, Default: “0.0”, no range
 - Name: “Water surface observation gauge”, *Legal on interior*: yes, *Values*: none.
8. Add the following boundary conditions and their parameters and limitations under the *Nodestring* tab:
 - Name: “Water surface”, *Legal on interior*: no, *Values*:
 - Name: “Elevation”, Type: “float”, Default: “0.0”, no range

- Name: “Essential/Natural factor”, Type: “float”, Default: “0.0”, Minimum: “0.0”, Maximum: “1.0”
 - Name: “Vary along nodestring factor”, Type: “float”, Default: “1.0”, Minimum: “0.0”, Maximum: “10.0”
 - Name: “Flow”, Legal on interior: no, Values:
 - Name: “Flow rate (cfs)”, Type: “float/curve”, Default: click **Define**. In the *Define Float/Curve Type* dialog set both “Flow” and “Curve” to Default: “0.0”, Minimum: “0”, Maximum: leave blank. Click **OK** when done with the *Define Float/Curve Type* dialog.
 - Name: “Supercritical”, Legal on interior: no, Values: none
 - Name: “1D weir segment”, Legal on interior: yes, Values:
 - Name: “Discharge coefficient”, Type: “float”, Default: “1.0”, Minimum: “0”, Maximum: leave blank
 - Name: “Weir width (ft)”, Type: “float”, Default: “1.0”, Minimum: “0”, Maximum: leave blank
 - Name: “Crest level (ft above sea level)”, Type: “float”, Default: “0.0”, no range
 - Name: “Equation (off = water level / on = energy head)”, Type: “boolean”, Default: unchecked
 - Name: “Sediment trap”, Legal on interior: yes, Values: none
9. Add the following boundary condition and its parameters and limitations under the *Element* tab:
- Name: “2D weir”, Legal on interior: no, Values:
 - Name: “Discharge coefficient”, Type: “float”, Default: “1.0”, Minimum: “0”, Maximum: leave blank
 - Name: “Crest level (ft above sea level)”, Type: “float”, Default: “0.0”, no range
 - Name: “Equation (0 = water level / 1 = energy head)”, Type: “boolean”, Default: unchecked

The boundary conditions have been declared now for all entities to be utilized; however the nodestring boundary condition *Sediment trap* is only needed when a simulation depicts sediment transportation. To assign the *Sediment trap* boundary condition to the *Sediment transport* parameter group:

1. Select the *Nodestring* tab.
2. Check the *Specify parameter group correlation for boundary conditions* toggle at the bottom of the dialog. This enables a new column called *Corr. Param Group* for each tab in the spreadsheet.
3. For *Sediment trap*, under *Corr. Param. Group* (Correlated Parameter Group), select “Sediment transport” (this list includes the parameter groups created in section 3.1). Leave all other conditions as “(none)” to allow generality.

4. Click **OK** to save and exit this dialog.

The remaining portion of the model interface to define is material attributes.

3.3 Material Properties

The **Material Properties** button is used to set the attributes of the simulation's mesh. To define mesh property parameters:

1. Click the Material Properties... button.
2. Click the Define... button next to *Hydrodynamic*
3. In the *Hydrodynamic Parameter Definition* dialog, add the following material conditions:
 - *Name*: "Manning", *Type*: "options", *Default*: "constant", *Dependencies*: "Global."
 - Before completing the *Default* section, click the **Define opts** button to bring up an *Option Definition* dialog.
 - Enter "constant" and "vary by depth" then click **OK**.
 - Click **None** to set the *Dependencies*.
 - In the *Dependencies* dialog set:
 - *Dependency Level*: "Global"
 - *Parent option*: "Hydrodynamic→Friction type"
 - *Manning*: check *Chezy*: uncheck.
 - Click **OK** to close the *Dependencies* dialog.

Note: A dependency for Manning was just created. Manning is only available when the global parameter "friction type" is set to Manning. If Chezy is selected this parameter is unavailable. This will be demonstrated later in the tutorial.

- *Name*: "Manning n1", *Type*: "float", *Default*: "0.035", *Minimum*: "0.01", *Maximum*: "0.18", *Dependency*: "Local"
 - Click **None** to set the *Dependencies*.
 - In the *Dependencies* dialog set: *Dependency Level* "Local", *Parent option*: "Manning", *Constant*: check, *Vary by depth*: uncheck. Click **OK**.
- *Name*: "Manning n2", *Type*: "curve", *Default*: "Axis Title", no range, *Dependencies*: "Local"
 - To set the *Default*, click **Axis titles...**
 - In the *Axis Titles* dialog set: *X Axis Title* "curve depth", *Y Axis Title*: "y". Click **OK**.
 - Click **None** to set the *Dependencies*.

- In the *Dependencies* dialog set: *Dependency Level* “Local”, *Parent option*: “Manning”, *Constant*: uncheck, *Vary by depth*: check. Click **OK**.
 - *Name*: “Chezy”, *Type*: “float”, *Default*: “0.0”, no range, *Dependency*: “Global”
 - Click **None** to set the *Dependencies*.
 - In the *Dependencies* dialog set: *Dependency Level* “Global”, *Parent option*: “Hydrodynamic→Friction type”, *Manning*: uncheck, *Chezy*: check. Click **OK**.
 - Click **OK** to close the *Hydrodynamic Parameter Definition* dialog.
4. Click **OK** to save and exit the *Material Global Parameters* dialog, but don't close the *Define Model* dialog.

The master “*.2dm” file is now complete for this model interface.

4 Protecting and Saving the Model Definition

It is recommended to protect the model interface from accidental manipulation once it has been defined and to always have a version ready for new simulations. By saving a clean master “.2dm” for each outside model executable, SMS capabilities are enhanced and more efficient for the user. To guard and save the *Gen2DM* definition just created:

1. In the *Define Model* dialog, check the *Lock model definition* toggle.
2. Enter the password “sms-gen2dm” in the enabled *Key* field.
3. Click **Close** to finish.

The key is case sensitive. If the incorrect key is typed in when opening the *Define Model* dialog, the dialog will not open. The password is not protected or encrypted. It is written to the file and can be found easily by opening the file in a text editor. Please refer to section 10 *Gen2DM (*.2dm) File Format* for more information.

4. Select *File* | **Save As**. This will open the *Save As* dialog.
5. Explore to the *tutorial\SMS_Gen2DM* directory.
6. Enter “Master_Gen2DM” as the *File name*. Verify that the file type is “2D Mesh Files (*.2dm).”
7. Click **Save**.

To prevent the repetition of redefining the interface, always backup and store at least one copy of the initial master file or simulation. This should be in a “.2dm” file with mesh and assignment information.

5 Assigning Model Parameters

For the remainder of this tutorial use the “*.2dm” file with the *Gen2DM* model definition just created and a geometry. To close the file and open the next:

1. Select *File* | **Delete All**.
2. Click the **No** button to the message: “Do you wish to keep the Generic 2D Mesh model template(s)?”. It’s necessary to delete all.
3. If another deletion message appears, click the **Yes** button.
4. Select *File* | **Open**.
5. In the Data Files Folder for this tutorial, open the file “DoublePipe.2dm”. This file should be similar to the one just created except it will contain a mesh.

The geometry data will open, as shown in Figure 3. SMS will automatically be in the Gen2DM model after the file is opened and the menu *Gen2DM* will appear in the menu bar. It’s therefore known that this file is associated with the “Gen2DM” model executable outside of SMS (fictitious model executable; the name is only an example for this tutorial).

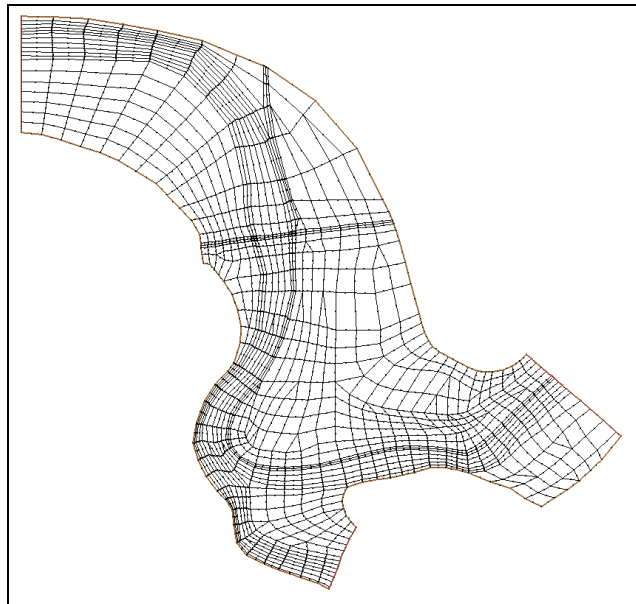


Figure 3 The mesh contained in the file *DoublePipe.2dm*

The Gen2DM file does not write out the coordinate system for later use, so redefine the system since a **Delete All** data command was performed.

6. Follow steps 1-3 in section 2 Specifying Model Units.
7. Select *Gen2DM* | **Define Model** and enter the password “sms-gen2dm”, the predefined model interface can be reviewed and edited. Click **OK**.
8. Close the *Define Model* dialog. Now to use this definition.

9. Select *Gen2DM* | **Global Parameters**. The *Gen2DM Global Parameters* dialog contains a tab for each parameter group defined in the model. In this case, there are two.
10. Select the *Hydrodynamic* tab. All variables have been initialized to the defined default values and *Activate parameter group* is checked. Also any boundary conditions that are correlated to this group are listed. Remember that any value entered in the *Hydrodynamic* tab or *Sediment transport* tab is subject to the constraints defined in the master interface.
11. Select **OK** to exit the *Global Parameters* dialog.

Global parameter values may be changed at any time by accessing the *Gen2DM Global Parameters* dialog. Upon saving, all current values are written out for use by the model executable.

6 Assigning Boundary Conditions


Before assigning boundary conditions, ensure the mesh composition complies with the outside SMS executable. Some models may only support certain element forms (triangular/quadratic) or advise against various mesh complexities. Since this model is conjured just as an example, the mesh will be assumed to be compliant. For adjustment of a mesh, use the options available in the *Nodes*, *Nodestrings* and *Elements* menus.

6.1 Creating Nodestrings

For most simulations, boundary conditions will be declared along nodestrings at the *open boundaries* of the mesh. Generally, a flow rate is specified across inflow boundaries and water surface elevation is specified across outflow boundaries for a simplistic run.

This example will have two inflow boundaries, one outflow boundary, and one interior boundary, so four nodestrings must be created. These boundaries are highlighted in Figure 4.

Nodestrings should be created from right to left when looking downstream and both nodestrings should span the entire river section. It does not matter which nodestring is created first. To create the outflow nodestring:

1. Choose the **Create Nodestrings**  tool from the *Toolbox*.
2. Start the nodestring by clicking on the upper node at the outflow boundary.
3. Hold the *SHIFT* key and double-click on the lower node at the outflow boundary to create and end the nodestring.
4. The inflow nodestring labeled 1 can be created likewise, making sure to create it right to left when looking downstream.
5. Create another nodestring for inflow labeled 2.
6. Create the final nodestring on the interior of the mesh.

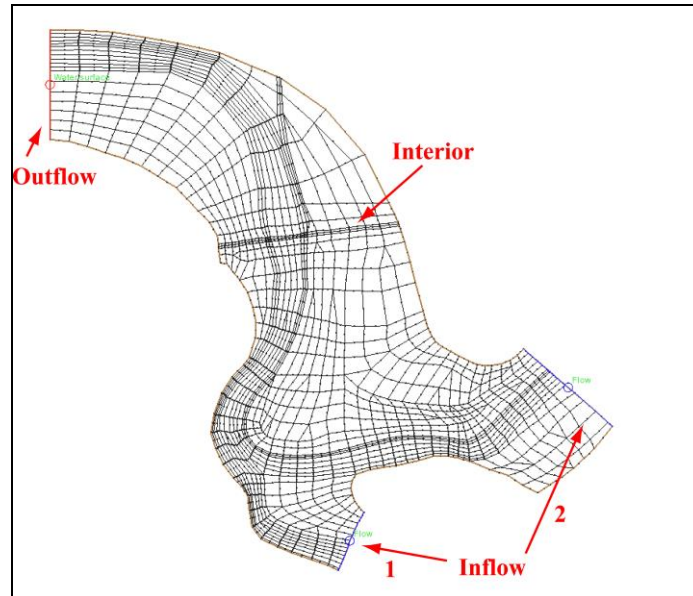




Figure 4 Position of the boundary nodestrings in the mesh

6.2 Assigning Boundary Conditions


To assign a boundary condition, choose the selection tool for an entity and select a desired entity. Assign conditions to the nodestrings, a node and an element:

Nodestrings:


1. Choose the **Select Nodestrings**  tool from the *Toolbox*. An icon appears at the center of each nodestring.
2. Select the inflow nodestring *1* by clicking on the icon.
3. Select *Gen2DM / Assign BC*. The *Gen2DM Nodestring Boundary Conditions* dialog should appear. Notice there are two tree item groups on the left side. *No Group* and *Sediment transport*. The *No Group* consists of all boundary conditions that are not correlated to a specific parameter group.
4. Click on *Flow* (without turning on the check). Notice *Flow* is displayed on the right screen but the options are disabled.
5. Click on *Flow* so a checkmark is turned on. Now notice that the options are enabled.
6. Make sure that *Constant* is toggled on and enter a *Flow rate* of “5000.0” cfs.
7. Click **OK** to assign the boundary condition. *Flow* has now been assigned as a boundary condition because it was checked on.
8. Select the inflow nodestring *2* by clicking on the icon.
9. Repeat steps 3-6 except give *Flow rate* a value of “8900.0” cfs. Click **OK**.

10. Choose the **Select Nodestrings**  tool from the *Toolbox*. Select the interior nodestring by clicking on the icon. This nodestring is located in the middle of the mesh.
11. Right-click and select **Assign BC**. This is an alternate way of assigning boundary conditions without using the *Gen2DM | Assign BC* menu command.
12. Look at the tree item options on the left. Notice that all of the conditions are gone except for *ID weir segment* and *Sediment trap*. That is because these boundary conditions have *Legal on interior* checked in the model definition (see section 3.2).
13. Click on *ID weir segment* so it has a check mark.
14. Click **OK** to assign the type *ID weir segment* using the default values.

Nodes:

1. Choose the **Select Mesh Nodes**  tool from the *Toolbox*.
2. Select a group of interior nodes by clicking and dragging a selection box.
3. Select *Gen2DM | Assign BC*.
4. Select *Water surface observation gauge* option so it has a check mark.
5. This condition does not contain any values to be entered, so select **OK** to assign the boundary condition to all selected nodes.


Elements:

1. Choose the **Select Elements**  tool from the *Toolbox*.
2. Select an interior element (not bordering the mesh boundary).
3. Select *Gen2DM | Assign BC*.
4. An error message should appear. This message is because the *Legal on interior* option has not been checked in the element boundary conditions definition. Click **OK** to this message.
5. Select an element along the inflow nodestring.
6. Select *Gen2DM | Assign BC*.
7. Notice the error message did not appear. Select **Cancel** because the weir should be assigned inside the mesh boundary.
8. Select *Gen2DM | Define Model* to bring up the *Define Model* dialog.
9. Enter the key, "sms-gen2dm" into the field. Click **OK**.
10. Click on the **Boundary Conditions** definition button.
11. Select the *Element* tab.
12. Check *Legal on interior* for 2D weir. Select **OK**.
13. Select the **Close** button.

14. Select an interior element (not bordering the mesh boundary) and assign it the *2D weir* boundary condition using default values.

6.3 Correlation and Activation Benefits of Boundary Conditions

The *Sediment transport* parameter group will not be used any further during this simulation assignment, so to simplify assigning the outflow boundary condition turn off the group.


1. Select *Gen2DM* / **Global Parameters** to bring up the *Global Parameters* dialog.
2. Select the *Sediment transport* tab.
3. Uncheck the *Activate parameter group* check box.
4. Click **OK**.
5. Choose the **Select Nodestrings**  tool from the *Toolbox*.
6. Select the outflow nodestring (on the left side of mesh).
7. Select *Gen2DM* / **Assign BC**.

If all parameter groups which have boundary condition correlations are inactive, the *Group* selector is replaced with the text (*none*). If some groups with correlations are active and others inactive, the inactive boundary conditions will not appear in the *Group* selector.

8. Select *Water surface* in the *Type* selector.
9. Enter “323.30” for *Elevation*.
10. Click **OK** to assign the boundary condition.

6.4 Boundary Condition Display Options

Notice that the entities with assigned conditions have symbols and labels, all of which are in black. To increase the visibility of certain assignments, change the attributes of each:

1. Click the *Display Options*  macro or select *Display* | **Display Options** or press CTRL+d. The *Display Options* dialog should appear.
2. The *2D Mesh* tab should be displayed on top. If not, select it.
3. Click the *Nodestrings BC Options* button. This will open the *Gen2DM Nodestring Display Options* dialog.
4. Every nodestring boundary condition defined in the model is represented in the spreadsheet. Turn on *Water Surface*. Click the middle of the line style button for *Water surface*.
5. Toggle *Solid* on and enter a width of 5.
6. Change the *Line Color* to red.
7. Click **OK**.
8. Repeat for *Flow*, but with bright blue.

9. Uncheck the toggle next to *1D weir segment*.
10. Change the *Unassigned nodestrings* to be orange.
11. Turn on and change *Nodestring labels* to be green by clicking on the down arrow beside the text preview. Make sure *Nodestring labels* is checked.
12. Click **OK** and follow the same procedure to adjust the display options for the *Nodal BC* options so that they are clearly discernable. Nodal BC symbols may be difficult to see if below the size of 3.
13. Click **OK** to close the *Display Options* dialog.

Gen2DM boundary condition nodestrings receive a hollow circle symbol displayed at their midpoints. The symbol receives the same color as the nodestring line. Inactive nodestrings receive a second smaller hollow circle as part of their symbol, as shown in Figure 5.

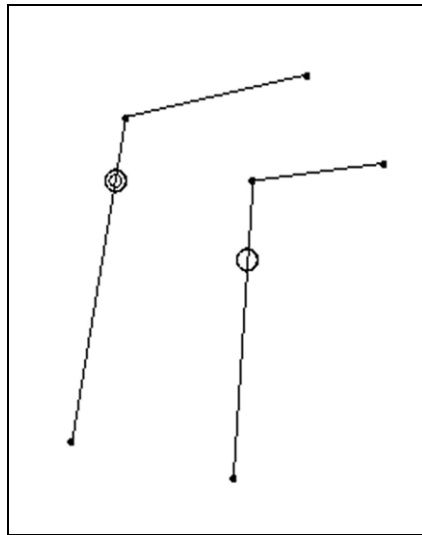


Figure 5 Symbols of inactive nodestrings (left) and active nodestrings

6.5 Dynamic Boundary Conditions

The boundary conditions of the model may be defined dynamically to allow for varying conditions by creating a curve for each dynamic value. To describe the inflow nodestring of type *Flow* as changing flow rates:


1. Select the inflow nodestring 1.
2. Select *Gen2DM | Assign BC*.
3. Place a checkmark in *Flow*.
4. Uncheck *Constant*.
5. Click the *Define* button.
6. Define the *Flow rate* curve in the *XY Series Editor* with the following values:
 - Time = 0.0, Value = 4000.0

- 20.0, 4500
 - 40.0, 5000
 - 60.0, 4500
 - 80.0, 4000
 - 100.0, 4000
7. Click **OK**.
 8. Click **OK**.

The data for each curve defined is stored by *SMS* by curve ID, but when written to file, each curve will be written out as a value for every time interval and given the entity and type ID it is describing.

6.6 Deleting Boundary Conditions


To delete a boundary condition:

1. Choose the **Select Nodestrings**  tool from the *Toolbar*.
2. Right-click on the interior nodestring found in the center of the mesh. Click on **Delete Selected**.
3. Click **Yes** in the dialog.

It is recommended that the model definition should remain untouched after creating a mesh. The manipulation of the boundary condition definitions (i.e. deleting types or type variables) or other model definition parameters may interfere with proper simulation set up and cause unforeseen problems.

7 Assigning Material Properties

Each element in the mesh is assigned a material type ID. This geometry has several material types. To see each of these materials:

1. Click the **Display Options**  macro or select *Display* | **Display Options** or press CTRL+d. This should bring up the *Display Options* dialog.
2. In the *2D Mesh* tab, turn on the *Materials* option.
3. Turn off *Nodes*, *Nodestrings*, and *Elements*.
4. Click the **OK** button to close the *Display Options* dialog.

The mesh boundary and materials should be on showing the sides and the main channel of the river as shown in **Error! Reference source not found.**

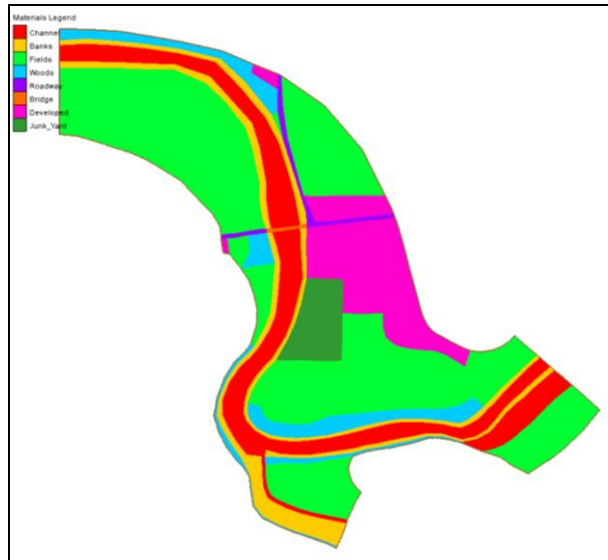


Figure 6 Material properties


The material properties define how water flows through the element. To edit the material parameters:

5. Select *Gen2DM* / **Material Properties**.
6. Click the General Material Properties button. The *Materials Data* dialog will appear. This dialog can also open by selecting *Edit* | Material Data.
7. Observe the different existing materials and their assigned pattern. Click OK.
8. Highlight *Channel*.
9. Enter “0.030” for *Manning n1*.
10. Click OK.

8 Multiple Material Assignments

It's possible to separate material assignments for each global parameter group that has been created. In this case, there are two: Hydrodynamic and Sediment transport.

1. Select *Gen2DM* / **Define Model** and enter “sms-gen2dm”.
2. Click on **Material Properties**.
3. Check the box that says: *Have a separate material assignment for each parameter group*. Click **OK**. Click **Close**.
4. Select *Gen2DM* / **Global Parameter**. The *Gen2DM Global Parameters* dialog should appear. Select *Sediment transport* tab. Check *Activate parameter group*. Click **OK**.
5. Select *Gen2DM* / **Set Active Material Group**. The *Active Material Groups* dialog should appear. Select *Sediment transport* and click **OK**.

6. Notice now that users are looking at the materials assigned to Sediment transport. Currently no materials have been assigned, so everything is disabled. Choose the **Select Elements**  tool from the *Toolbox*. Left-click somewhere in the mesh. Then right-click and select *Assign Material type*. The *Materials Data* dialog will appear. Select a material and click *Select*.
7. Select *Gen2DM / Set Active Material Group*. The *Active Material Groups* dialog will reappear. Select *Hydrodynamic* and click **OK**.
8. Before continuing, turn the *Materials* back off and the *Elements* back on in the *Display Options* dialog (*Display / Display Options*).

9 Dependencies

Earlier, users setup some dependency relationships with the Global parameter *Hydrodynamic* → *Friction type* (parent), with Material parameter *Hydrodynamic* → *Manning*, and *Hydrodynamic* → *Chezy*. *Hydrodynamic* → *Manning* (parent) then became the dependency for *Hydrodynamic* → *Manning n1* and *Hydrodynamic* → *Manning n2*. Now to demonstrate how this works.

1. Select *Gen2DM / Material Properties*. This will bring up the *Gen2DM Material Properties* dialog.
2. Select the *Hydrodynamic* tab.
3. Click on “Channel” on the left-side. By default the global *Friction type* is currently set to *Manning*. Because of this, users see *Manning* and *Manning n1*.
4. Now change *Manning* from *constant* to *vary by depth*. Because *Manning n2* is dependent upon *vary by depth* it now becomes available. Notice that *Manning n1* disappears. Now click **OK**.
5. Select *Gen2DM / Global Parameters*. The *Gen2DM Global Parameters* dialog should appear.
6. Change the *Friction type* to “Chezy” and click **OK**.
7. Select *Gen2DM / Material Properties*. The *Gen2DM Material Properties* dialog will reappear. Notice now that the *Manning* parameters are gone and *Chezy* is now available.
8. Click **OK**.

Dependencies can only be created from previously defined parameters. These parameters must be of the type “options” (not int, float, curve, etc.). Dependencies can be created for materials, global parameters and/or boundary conditions.

10 Gen2DM Model Check

SMS can detect anomalies within the mesh and model definitions, by performing a quick check. Not all invalid situations can be distinguished because Gen2DM is a user defined

interface for an executable outside SMS. However, SMS will look for basic mesh problems and missing model definitions. To run this check:

1. Select *Gen2DM* / **Check Mesh**.
2. The *Model Checker* dialog should appear and give a list of potential issues. The highlighted *Problem* will include a *Description* and a *Fix*. Read the description and follow the fix instructions if necessary by clicking the **Done** button and performing the required operations. Some problems may not need to be fixed such as changes to the model definition. If no problems are found, a message stating such will be displayed. In this case click **OK**.
3. Manually check model variables for validity as suggested by any documents included with the model executable.
4. Save the file as “DoublePipe-sim” by selecting *File* | **Save As**.
5. Change the *Save as type* to *2D Mesh Files (*.2dm)* and enter the name.
6. Click the **Save** button to save the simulation.

11 Exporting the Gen2DM File and Running the Model

Exporting the Gen2DM file consists of simply saving the file. All definitions and assignments are contained in it. Use the computer operating system to move or copy the “.2dm” file as necessary and run the model executable. Follow the instructions for the executable concerning location of the file and how to import to compute the model simulation. If a desired executable outside of SMS will not recognize the Gen2DM file format, contact the model distributor or designer for information on new versions or an additional executable to reformat the “.2dm” file into a compliant form.

12 Conclusion

This concludes the “Generic 2D Mesh Model” tutorial.