SURFACE WATER MODELING SYSTEM

Generic 2D Mesh Model

1 Introduction

This lesson teaches you how to create and use a generic model interface for models executed outside of SMS. Sections 1 to 4 instruct how to create a master generic model interface and sections 5 to 12 instruct how to use a previously saved generic model interface. To start the creation of the master interface:

1. Make sure the Mesh Module is active and select Data | Switch Current Model.

2. Select to use a Generic model.

3. Click OK.

2 Specifying Model Units

Before continuing, make sure the units are as desired. To do this:

1. Select *Edit* | *Projection*.

2. Make sure the *Horizontal* and *Vertical System* and *Units* are appropriate for the model executable outside of SMS. For this example, set the *Horizontal Projection* to *Local* and units to *U.S Survey Feet*, *Vertical System* to *Local* and *Vertical Units* set to *U.S. Survey Feet*.

3. Click *OK* to exit the dialog.

3 Defining the Model Interface

Model interface parameters will define various states and characteristics of a model. These model parameters may include items such as those needed to describe 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:

1. Select *Mesh* | *Define Model*

2. Click the *Model Parameters* button

3. Under *Model Information* enter "Gen2DM" (for this example) for the *Name*. Upon exiting this and the main *Define Model* dialog, the menu item previously titled *Mesh* will be labeled *Gen2DM*.

4. For *Time Units*, enter the desired unit the model will be using. In this example enter "minutes."

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. By giving the model interface the model name, files can be opened in *SMS* and quickly recognized as pertaining to a particular model.

3.1 Global Parameters

To define the model parameters, in the *Define Model* dialog, first create a parameter group:

1. Click the *Global Parameters* button

2. Type the name "Hydrodynamic" for the first *Parameter Group* and then press *Enter* on the keyboard. The *Define…* button now becomes active.

3. Click the *Define…* button. This will open the *Hydrodynamic Parameter Definition* spreadsheet.

4. Enter the first *Hydrodynamic* parameter, "Time interval". When editing the name is complete, press the *Tab* key on the keyboard. The accompanying fields will be enabled.

5. Select *integer* in the *Type* column.

6. Set the *Default* to be 20 minutes (the time unit was previously declared as minutes).

7. The fictitious model will accept only positive time intervals, consequently declare the *Minimum* for *Time interval* as zero and leave the *Maximum* as blank.

8. Continue adding the following parameters with their limitations:

   - "Velocity max (ft/sec)", float, default of 75.0, range of 0.0 to 100.0
   - "Check for dry elements", boolean, on (checked)
   - "Element style", text, "quadratic"
   - "Critical scour velocity", options, Click the *Define opts* button and enter "0.8 ft/sec", "2.0 ft/sec", and "2.6 ft/sec" then select the default option to be "2.0 ft/sec".
   - "Friction type", options, Click the *Define opts* button and enter "Manning" and "Chezy". Select the default option as "Manning".

9. Click OK to exit the *Hydrodynamic Parameter Definition* dialog.

Any line in a spreadsheet can be deleted by highlighting the name and pushing the *Delete* key. Also, each name in a spreadsheet must be unique.

1. Create another *Parameter Group* called "Sediment transport" just after "Hydrodynamic".

2. Press *Tab* on the keyboard and click on the *Define* button and enter the following information:

   - "Time interval", integer, 10, 0, leave blank
   - "Source X position", float, 0.0, no range
   - "Source Y position", float, 0.0, no range
   - "Source elevation", float, 0.0, no range
   - "Parcel mass (slug)", float, 0.5, 0.0001, leave blank
   - "Particle mass (slug)", float, 0.003, 0.0001, leave blank
   - "Particle size (in)", float, 0.05, 0, leave blank
   - "Deviation", float, 0.0, no range
   - "Average density (slug/ft^3)", float, 3.0, 1.5 to 6.0

3. Click *OK*.

You should now have two complete parameter groups defined for later use. Click *OK* 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. Click the *Boundary Conditions…* Definition button.

2. On the *Node* tab enter the name "Water sink/source", 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 value "Flow rate (cfs)", float, 0.0, 0 and leave blank .

6. Add "Water temperature (F)", float, 65.0, 32.5 to 100

7. Click *OK* to save and exit the spreadsheet.

8. Continue adding the following boundary conditions and their parameters and limitations under the *Node* tab:
   - "Ceiling (pressure flow)", not legal, value:
     - o "Ceiling (ft above sea level)", float, 0.0, no bounds
     - o Click *Ok.*
   - "Water surface observation gauge," legal , no defined values

4. Add the following boundary conditions and their parameters and limitations under the *Nodestring* tab:
   - "Water surface," not legal, values:
     - o "Elevation", float, 0.0, no bounds
     - o "Essential/Natural factor", float, 0.0, 0.0 to 1.0
     - o "Vary along nodestring factor", float, 1.0, 0.0 to 10.0
     - o Click *OK.*
   - "Flow," not legal, value:
     - o "Flow rate (cfs)", float, 0.0, 0, leave blank
     - o Click *OK.*
   - "Supercritical," not legal, no defined values
   - "1D weir segment," legal, values:

- “Discharge coefficient”, float, 1.0, 0, leave blank
    - “Weir width (ft)”, float, 1.0, 0, leave blank
    - “Crest level (ft above sea level)”, float, 0.0, no bounds
    - “Equation (off = water level / on = energy head)”, boolean, unchecked
    - Click *OK*.
- “Sediment trap,” legal, no defined values

5. Add the following boundary condition and its parameters and limitations under the Element tab:

    - “2D weir,” not legal, values:
        - “Discharge coefficient”, float, 1.0, 0, leave blank
        - “Crest level (ft above sea level)”, float, 0.0, no bounds
        - “Equation (0 = water level / 1 = energy head)”, boolean, unchecked
        - Click *OK*.

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. Add the following material conditions:

- “Manning”, options, Click the *Define opts* button and enter “constant” and “vary by depth”, then select the default option to be “constant”. Click the *Define…* button under Dependencies. Select “Global” for dependency level and “Friction type” for Parent options. Uncheck “Chezy” so only “Manning” is checked and click *OK*.

Note: We have now just created a dependency for Manning. 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.

- “Manning n1”, float, .035, .01, 0.18. Click the *Define…* button under Dependencies. Select “Local”, “Manning” and uncheck “vary by depth” so only “constant” is checked”. Click *OK*.

- “Manning n2”, curve. Click the *Axis titles…* and enter “curve depth for the x axis title, and “y” for the y axis title then click *OK*. Click the “Define…” button under Dependencies. Select “Local”, “Manning” and check “vary by depth” and uncheck “constant”. Click *OK*.

- “Chezy”, float, 0.0 then blank. Click the “Define…” button under Dependencies. Select “Global”, “Friction type”, uncheck “Manning” and check “Chezy”. Click “*Ok*”.

- Click *OK*.

4. Click *OK* to save and exit this 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 you type in the incorrect key 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*.

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 back up and store at least one copy of the initial master file or a simulation ("*.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. You want 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 we just created except it will contain a mesh.

The geometry data will open, as shown in Figure 1. *SMS* will automatically be in the Gen2DM model after the file is opened and the menu *Gen2DM* will appear in the menu bar. You therefore know 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).

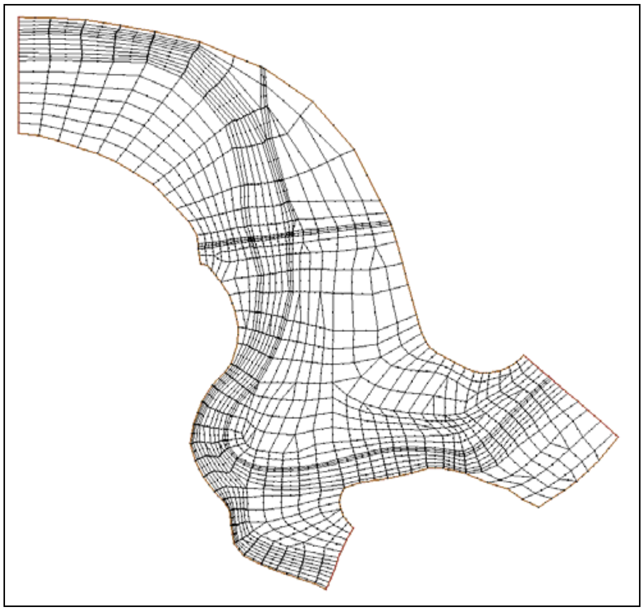

Figure 1. The mesh contained in the file DoublePipe.2dm.

The Gen2DM file does not write out the coordinate system for later use, so redefine the system since you have performed a delete all data SMS command. Follow steps 1-3 in section 2 Specifying Model Units.

If you select *Gen2DM | Define Model* and enter the password "sms-gen2dm," the predefined model interface can be reviewed and edited. You will now be using this definition.

1. Select *Gen2DM | Global Parameters*. The dialog contains a tab for each parameter group defined in the model. In our case we have two.

2. 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 you enter in the *Hydrodynamic* tab or *Sediment transport* tab is subject to the constraints defined in the master interface.

3. Select *OK* to exit.

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 our 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.

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 one inflow boundary and one outflow boundary so two nodestrings must be created. These boundaries are highlighted in Figure 2.

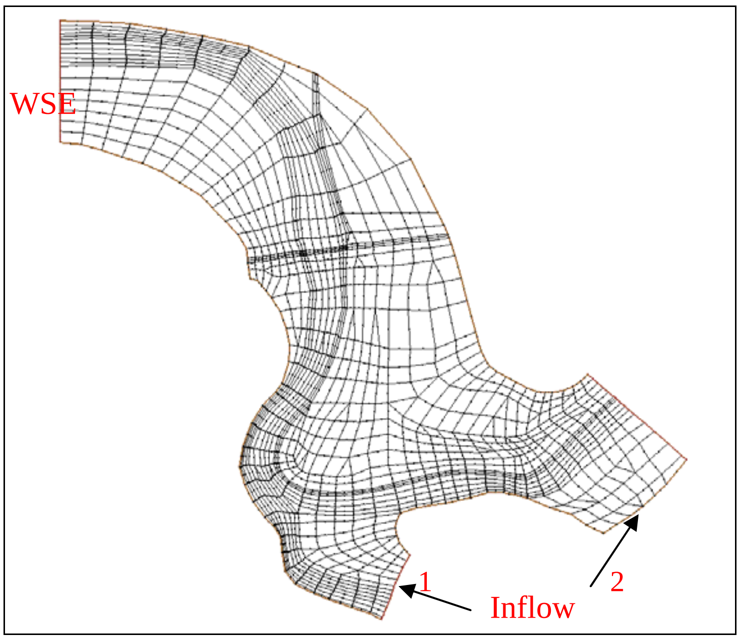

Figure 2. Position of the boundary nodestrings in the mesh.

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 inflow nodestring labeled 1:

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.

The inflow nodestring can be created likewise, making sure you create it right to left when looking downstream. Create another nodestring for inflow labeled 2.

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*.

You'll 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. Click on *Flow* (without turning on the check). You'll notice *Flow* is displayed on the right screen but the options are disabled.

4. Click on *Flow* so a checkmark is turned on. You'll now notice that the options are enabled.

5. Enter a *Flow rate* of 5000.0 cfs.

6. Click *OK* to assign the boundary condition. *Flow* has now been assigned as a boundary condition because it was checked on.

7. Select the inflow nodestring 2 by clicking on the icon.

8. Repeat steps 3-6 except give Flow rate a value of 8900.0 cfs. Click *OK*.

9. 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.

10. Right click and select *Assign BC*. This is an alternate way of assigning boundary conditions without using the Select *Gen2DM | Assign BC* menu.

11. Look at the tree item options on the left. Notice that all of our conditions are gone except for *1D weir segment* and *Sediment trap*. That is because these boundary conditions have *Legal on interior* checked in the model definition (see section 3.2).

12. Click on *1D weir segment* so it has a check mark.

13. Click *OK* to assign the type *1D 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 will appear. Click *OK* to this message.
5. Select an element along the inflow nodestring.
6. Select *Gen2DM* | *Assign BC*.
7. Notice now we don't get the error message. Select *Cancel* because the weir should be assigned inside the mesh boundary.
8. Select *Gen2DM* | *Define Model*.
9. Enter the key, "sms-gen2dm" into the field.
10. Click on the *Boundary Condition Definition* button.
11. Select the *Element* tab.
12. Check *Legal on interior* for *2D weir*.
13. Select *OK*.
14. Select the *Close* button.
15. 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*.
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

You may have noticed 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*.
2. The *2D Mesh* tab should be displayed on top, if not, select it.
3. Click the *Nodestrings Options* button.
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 3.

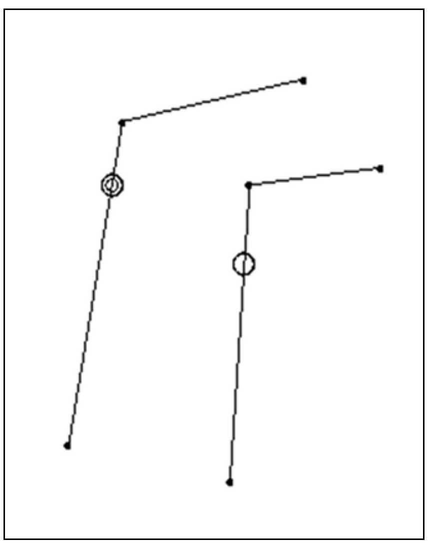

Figure 3. Symbols of inactive nodestrings (left) and active nodestrings.

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. Click the *Define* button.
5. 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
6. Click *OK*.
7. 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.
3. Click on *Delete Selected*.
4. Click *Yes* on 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*.
2. In the *2D Mesh* tab, turn on *Materials*.
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 Figure 4.

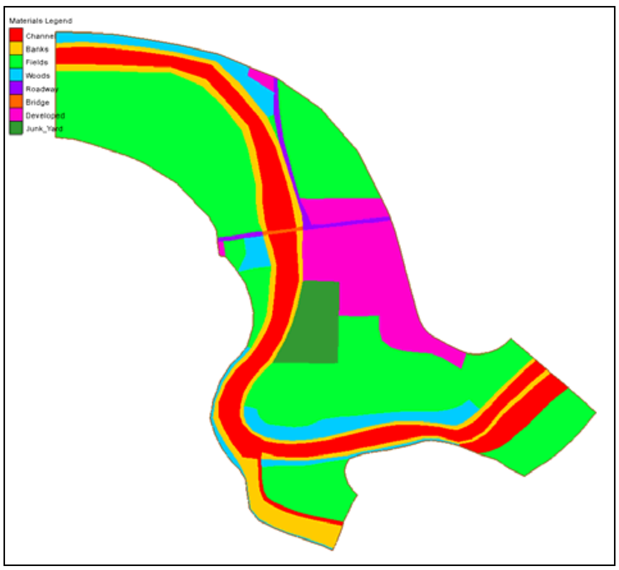

Figure 4 Material properties

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

1. Select *Gen2DM* | *Material Properties*.
2. Click the *General Material Properties* button. You can also open this dialog by selecting *Edit* | *Material Data*.
3. Observe the different existing materials and their assigned pattern.
4. Highlight *Channel*.
5. Select *constant* as the value for *Manning*. Enter 0.030 for *Manning n1*.

6. Click *OK*.

8 Multiple Material Assignments

You can have separate material assignments for each global parameter group that has been created. In our case we have 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*. Select *Sediment transport* tab. Check *Activate parameter group*. Click *OK*.

5. Select *Gen2DM | Set Active Material Group.* Select *Sediment transport* and click *OK*.

6. Notice now that we 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*. Select a material of your choosing and click *Select*.

7. Select *Gen2DM | Set Active Material Group.* Select *Hydrodynamic* and click *OK*.

8. Before continuing, turn the *Materials* back off and the *Elements* back on in the *Display Options (Display | Display Options).*

9 Dependencies

Earlier we 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 we will demonstrate how this works.

1. Select *Gen2DM | Material Properties*.

2. Select the *Hydrodynamic* tab.

3. Click on *Channel* on the left-side. By default our global F*riction type* is currently set to *manning*. Because of this, we see *Manning* and *Manning n1*. Now change *Manning* from *constant* to *vary by depth*. Because *Manning n2* is dependant upon *vary by depth* it now becomes available. Notice that *Manning n1* disappears. Now click *OK*.

4. Select *Gen2DM | Global Parameters*.

5. Change the *Friction type* to *Chezy* and click *OK*.

6. Select *Gen2DM | Material Properties*. Notice now that the *Manning* parameters are gone and *Chezy* is now available. 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 Gen2DM (*.2dm) File Format

SMS uses a generic format that is extremely flexible to write out "*.2dm" file. The file could be opened in a text editor (that does not trim line length) and easily edited with basic knowledge of the *cards* used. The file is organized by *key words at the beginning of each line* followed by data describing attributes. If you opened *Conestoga_River_Sim.2dm* using *Microsoft ® Notepad*, you would see the following format (cards shown in proper order with data and description), but probably not the same values:

MESH2D

- Denotes the beginning of the mesh data – no data

E3T 94 1103 1101 1102 2

- A linear triangular element (3 vertices) – element ID (integer), 1st vertex node ID (integer), 2nd vertex node ID (integer), 3rd vertex node ID (integer), element material type ID (integer)

E4Q 95 103 388 387 104 2

- A linear quadratic element (4 vertices) – element ID (integer), 1st vertex node ID (integer), 2nd vertex node ID (integer), 3rd vertex node ID (integer), 4th vertex node ID (integer), element material type ID (integer)

E6T 17 7 1254 25 1255 8 1256 2

- A quadratic triangular element (3 vertices, 3 mid points) – element ID (integer), 1st vertex node ID (integer), 1st mid point node ID (integer), 2nd vertex node ID (integer), 2nd midpoint node ID (integer), 3rd vertex node ID (integer), 3rd mid point node ID (integer), element material type ID (integer)

E8Q 18 21 1248 7 1256 8 1257 26 1258 2

- A quadratic quadrilateral element (4 vertices, 4 mid points) – element ID (integer), 1st vertex node ID (integer), 1st mid point node ID (integer), 2nd vertex node ID (integer), 2nd midpoint node ID (integer), 3rd vertex node ID (integer), 3rd mid point node ID (integer), 4th vertex node ID (integer), 4th mid point node ID (integer), element material type ID (integer)

E9Q 50 28 1315 53 1316 52 1317 27 1260 4630 2

- A quadratic quadrilateral element (4 vertices, 4 mid points, an element centered node) – element ID (integer), 1st vertex node ID (integer), 1st mid point node ID (integer), 2nd vertex node ID (integer), 2nd midpoint node ID (integer), 3rd vertex node ID (integer), 3rd mid point node ID (integer), 4th vertex node ID (integer), 4th mid point node ID (integer), the element centered node ID (integer), element material type ID (integer)

Note: The *E3T*, *E4Q*, *E6T*, *E8Q* and *E9Q* cards may be intermixed in the written order. The mesh is written by starting with one element and listing neighboring elements. The node IDs for these cards are in counterclockwise order around the element (ending with the centroid node for *EQ9s*). Each card will be written only if an element of the type is present in the mesh.

ND 3260 -7.62822790e+001 4.00306200e+001 7.40765873e+001

- A node – node ID (integer), x position (real number), y position (real number), z position (real number)

Note: The xyz positions are positive/negative one digit with eight decimal places followed by "e" (times ten to the…) positive/negative power real numbers.

NS 567 2489 568 2545 594 2626 630 2702 -663

- A nodestring – nodestring head node ID, 1st vertex node ID (if any),… last vertex node ID (if any), nodestring tail node ID

Note: The nodestring tail is denoted by a negative sign in front of the tail node ID. A nodestring may consist of more than ten nodes which constitute a file line, consequently a nodestring may extend multiple *NS* cards. Each sequential line should be read until the negative tail node ID is found, ending the nodestring definition. If no nodestrings are present in the mesh, this card will not be written.

Note: Node IDs for all cards above are limited to six digits (i.e. 999999 maximum).

BEGPARAMDEF

- Denotes the end of the mesh data (begun by *MESH2D*) and the beginning of the Gen2DM model definition – no data

GM "Gen2DM"

- Gen2DM model name – model name (text)

Note: Text is always delimited by quotation marks.

SI 0

- Using International System of Units – 0 (false) or 1 (true)

DY 1

- Dynamic model – 0 (false) or 1 (true)

TU "minutes"

- Time units –time unit name (text)

TD 20 1000

- Dynamic time data – time step length (integer/real number), maximum model time (integer/real number)

❖ Note: *TD* data is written as an integer when possible or written as a real number if decimal places are not all zeroes.

KEY "sms-gen2dm"

- Gen2DM model definition informal security password – key (text)

Note: This card is only written when the model definition is locked at time of saving.

GP 1 "Hydrodynamic" 1

- Global parameter group – id, group name (text), activity state: 0 (false) or 1 (true)

GP_DEF 1 1 "Time interval" 1 20 0 2147483657

- Global parameter definition – global group id, parameter id, parameter name (text), parameter type (integer), …
    - If the parameter type is zero (boolean type): …default value: 0 (false) or 1 (true)
    - Type is one (integer type): …default value (integer), minimum value (integer), maximum value (integer)
    - Type is two (real number type): …default value (real number), minimum value (real number), maximum value (real number)
    - Type is three (text type): …default value (text)
    - Type is four (option type): …default selection (text)

Note: The GP_OPTS card is only written when the previous GP_DEF card has a parameter type of four. The default value of the option parameter should be one of the options list in the GP_OPTS card line.

Note: Maximum/minimum integer values of 2147483648 and -2147483647 are understood as no bounds. The integer limits are characterized by four bytes. Maximum/minimum real number values of 1.79769e+308 and -1.79769e+308 are understood as no bounds. The real number limits are characterized by eight bytes.

GP_OPTS 1 7 "0.8 ft/sec" "2.0 ft/sec" "2.6 ft/sec"

- Parameter option definitions for the global group id 1, parameter id 7 followed by the options. 1st option (text), 2nd option (text) (if any), …last option (text). These particular options are for "Critical scour velocity".

NUME 3

- Number of entities – number of entities available: nodes, nodestrings and elements (integer)

BCPGC 1

- Boundary condition parameter group correlation – whether allowing boundary conditions to be correlated to parameter groups: 0 (false) or 1 (true)

DISP_OPTS entity 0 0 0 0 0 0 0 0

Display option for active nodes/elements or nodestrings

DISP_OPTS inactive 0 0 0 0 1 0 0 0

Display options for the inactive nodes/elements or nodestrings

DISP_OPTS multiple 0 0 0 0 1 0 0 0

Display options for multiple assigned boundary conditions for nodes/elements or nodestrings

- Boundary entity display options – entity type ID (integer), entity label color: red value (integer), entity label color: green value (integer), entity label color: blue value (integer), whether entity label is active: 0 (false) or 1 (true), pattern attributes (integer), size of the symbol (integer), and symbol attributes (integer)

BEFONT 1 -19 0 0 0 700 255 0 0 0 3 2 1 49 "Courier New"

- Boundary entity label font – entity type ID (integer), font information (integers and text) (if necessary)

Note: The font information may be represented by a single integer or represented by multiple integers and a font name.

Note: The *BEDISP* and *BEFONT* cards are written before the entity's boundary condition and values (if any) are written. The next instance of these cards will begin the next entity. The entity type ID equals zero for node, one for nodestring and two for element.

BC 0 "Water sink/source" 1 0 1 -1 "(none)"

- Boundary condition definition – entity type id (integer), condition name (text), condition id (integer), 0 – (not used for backwards compatibility of older versions), legal on the interior of the mesh: 0 (false) or 1 (true), -1 (always constant), name of parameter group this condition is correlated with (text)

Note: A group correlation name of *(none)* means the condition is a global condition and not parameter group specific.

BC_DEF 1 1 “Flow rate (cfs)” 2 0 0 1.79769e+308

(This card is similar to GP_DEF, expect we are defining definitions for boundary conditions)

- Boundary condition definition – condition id, parameter id, name (text), parameter type (integer), and the values associated with the type: default value, minimum value, maximum value (see GP_DEF)

BC_DEF_OPTS 1 1 “Flow rate (cfs)” 2 0 0 1.79769e+308

This card defines boundary condition options. See GP_DEF_OPTS for an explanation of this card.

BC_DISP_OPTS 3 0 0 0 0 1 0 1 1

- Same as DISP_OPTS expect the first parameter is the boundary condition parameter id(3). All remaining fields are the same as DISP_OPTS.

MAT 1 “Channel”

- Material definition. Id, name (text).

MAT_MULTI 0

MAT_MULTI 1 1

- Multiple material assignments on/off. If on (1), then the selected parameter group id

MAT_DEF 1 1 "Manning" 2 0.035 0.01 0.18

- Similar to GP_DEF and BC_DEF. See GP_DEF for more information.

MAT_DEP 2 1 3 "PARENT_LOCAL" "Manning" 0 "constant" 0 "vary by depth" 1

MAT_DEP 2 1 4 "PARENT_GLOBAL" "Friction type" 1 "Manning" 0 "Chezy" 1

- Dependency card. Material id (integer), group id (integer), parameter id (integer), dependency type (parent_global, parent_local, parent_self, parent_none), parent name (string), active (Boolean), 1st option name (string), 1st option active (boolean), 2nd option name, 2nd option active…for each option.

ENDPARAMDEF

- Denotes the end of the Gen2DM model definition (begun by *BEGPARAMDEF*) – no data

BEG2DMBC

- Denotes the beginning of the Gen2DM model assignment – no data

GP_VAL 1 1 40

GP_VAL 1 6 “Chezy”

- Global parameter assignment – global group id (integer), parameter id (integer), assigned value (float, int, text, etc)

MAT_VAL 1 1 2 0.05

- Material assignment – material id (integer), group parameter id (integer), parameter id (integer), and value

BC_VAL E 1013 1 1 1

BC_VAL S 5 4 1 6.7

BC_VAL N 3226 2 1 7.7

- Boundary condition assignment – entity type (N = node, E = element, S = nodestring), entity id (node id, element id or nodestring id), group parameter id, parameter id, value (float, int, text, etc)

Note: These cards are only written if the data that exists is different than the default data.

END2DMBC

- Denotes the ending of the Gen2DM model assignment (begun by *BEG2DMBC*) – no data

This concludes the “*.2dm” file format.

12 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 your 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.

13 Conclusion

This concludes the *Generic 2D Mesh Model* tutorial.

