Introduction

Welcome to the *DSM2 Version 7 tutorial*. This document will detail the development of a model of a simple channel system for use with HYDRO and QUAL. The directions for building the model consist of six tutorials, with each building in complexity from its predecessor. The reader is strongly encouraged to review the *DSM2 User's Manual* before attempting these tutorials. For setup and configuration instructions after installation, please refer to the *Quick Setup Guide*.

The interface for the DSM2 database will hereafter be referred to as the GUI. The directory where you installed DSM2 will be referred to as *{DSM2_home}*. E.g., if you accepted the default install directory, *{DSM2_home}* would be *d:\delta\dsm2*.

The first tutorial is called *Channels*, and involves setting up the channel grid, adding parameters, setting boundary conditions, and listing output locations. The second tutorial is called *Reservoir_Gate_Transfer*, and involves adding these components to the simple channel system.

The third tutorial is called *Layering*. The section provides instruction for modifying existing model information in the database by adding new data layers. Layers are key to the DSM2 management system. They allow input items to be grouped in logical bundles, and allow changes to be brought into an old simulation without erasing or altering archived items.

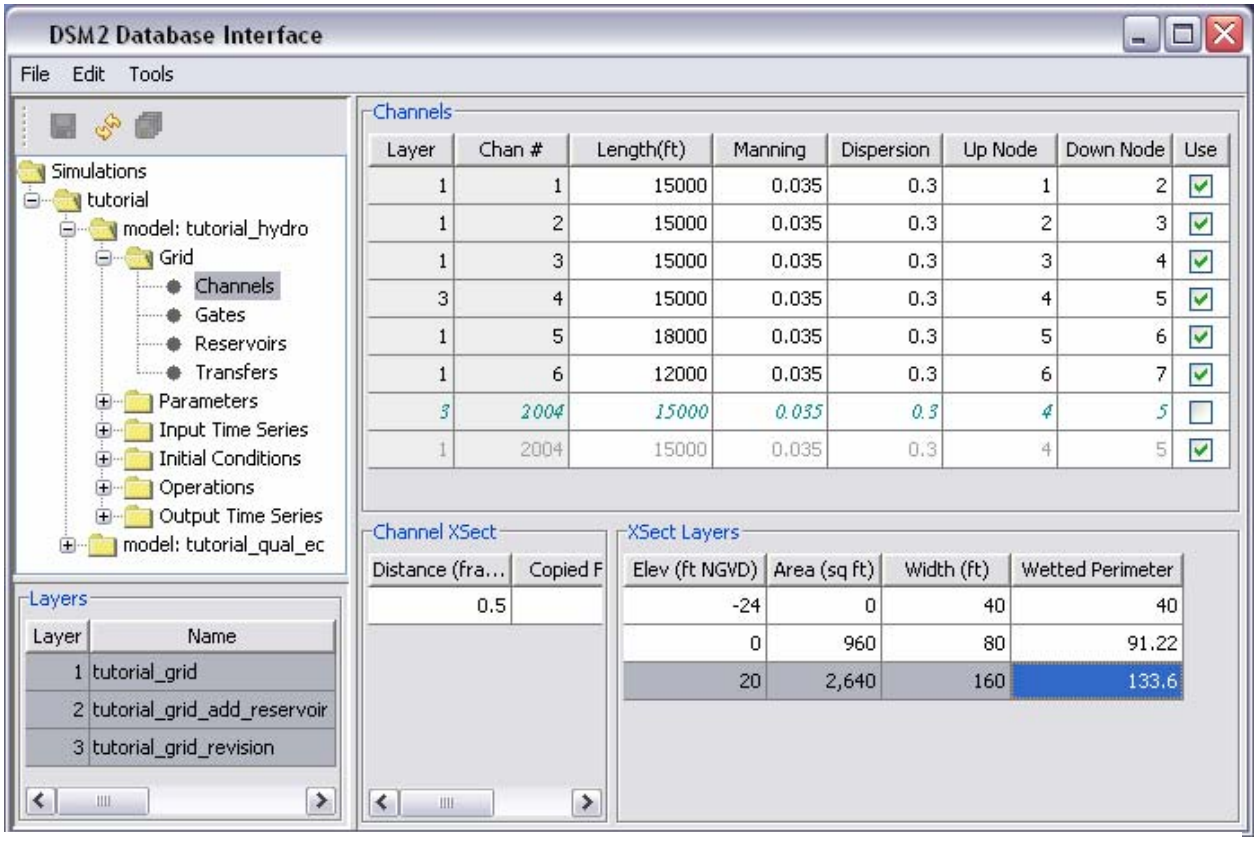
The fourth tutorial is called *Timevar*, and demonstrates the addition of time-varying information to the model. In the previous sections, all boundary conditions and gate timings were set as constant, and no input files were needed. In this section, the model is set to read time-varying information stored in DSS files.

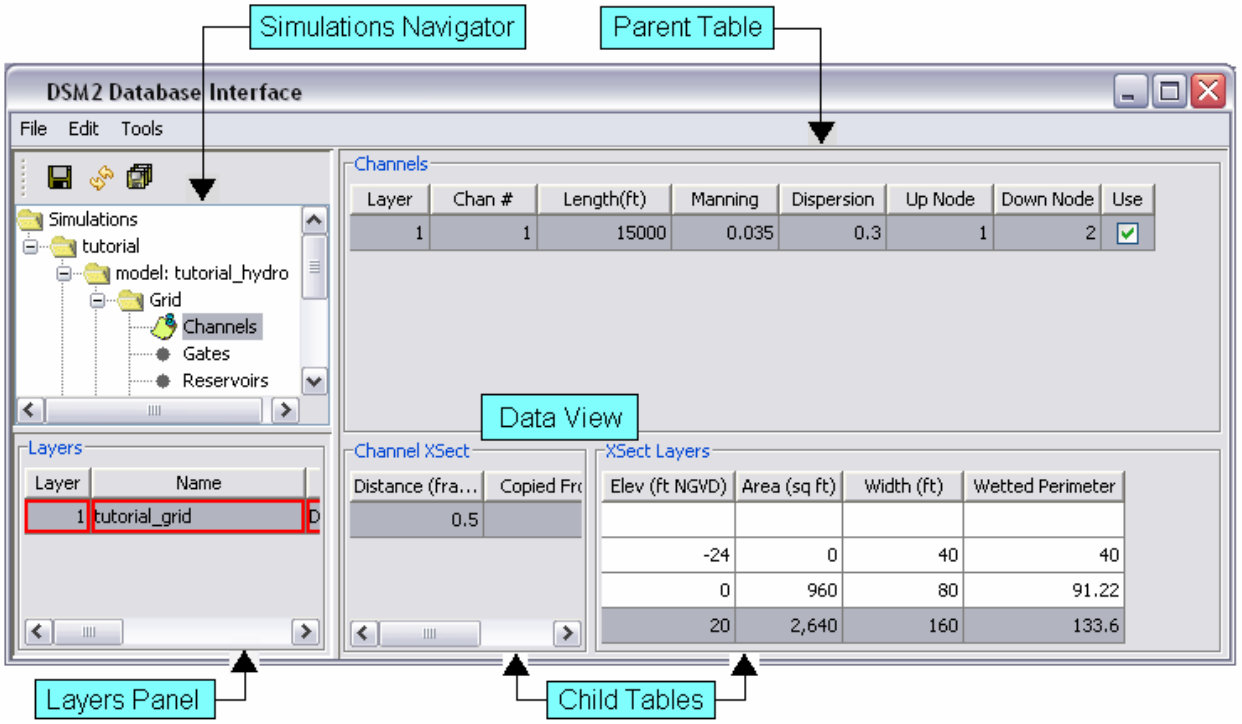
The fifth tutorial is called *Output*, and covers advanced output options. The first part involves modifications to the text input file, *hydro.inp*. The second part describes the use of *groups* in the GUI. With *groups*, the user can track constituents in QUAL from different sources.

The sixth tutorial is called *Oprule*, and covers the use of Operating Rule Language (ORL) statements to set gate operations. In the text version of DSM2, the text files were needed to explicitly state the operations of gates. With the GUI, expressions can

be used to make the model operate gates on-the-fly. E.g., a gate can be directed to automatically close when salinity conditions reach a certain threshold.

Figure 1 provides the terminology used in the GUI. The upper-left box is the *Simulations Navigator*. Here, the user may access the various folders associated with a simulation. One or more simulations may be loaded into the navigator at a given time.

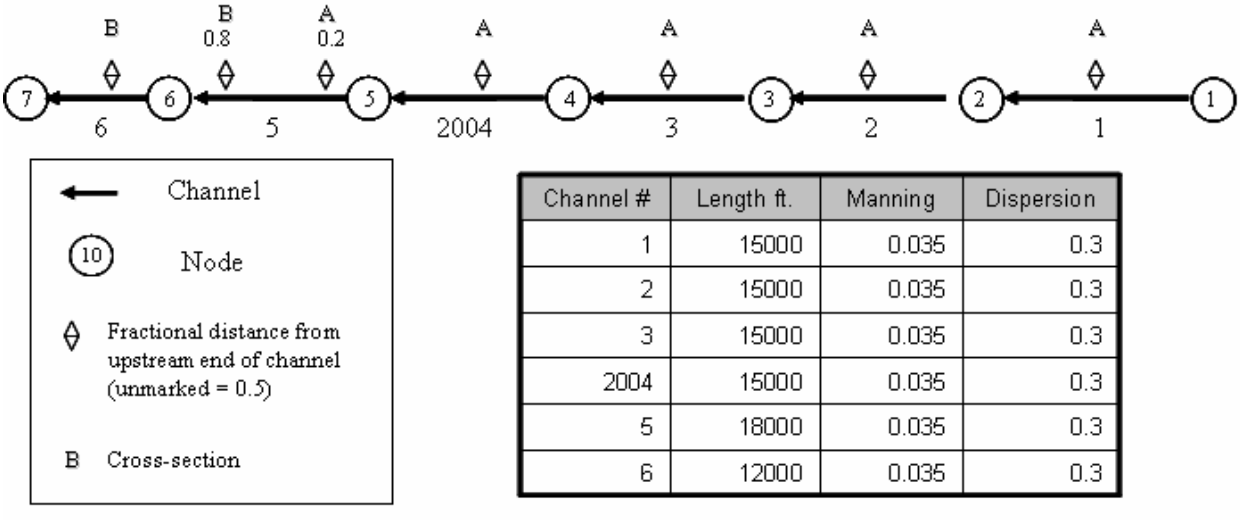




When a simulation is selected, the navigator displays only the top-level simulation folder. E.g., in Figure 1, the simulation name is *tutorial*, and all model information is contained in this folder. When the *tutorial* folder is expanded, it displays model folders. There will be one HYDRO model folder, and one or more QUAL model folders.

Tutorial 1: Channels

The purpose of this tutorial is to set up a simple channel-only grid with simple constant boundary conditions. The channels have the following configuration and specifications:



Channel #	Length ft.	Manning	Dispersion
1	15000	0.035	0.3
2	15000	0.035	0.3
3	15000	0.035	0.3
2004	15000	0.035	0.3
5	18000	0.035	0.3
6	12000	0.035	0.3

Figure 1 - Simple channel configuration and specifications.

The following steps will instruct you on how to create these channels and enter these and other specifications into the GUI.

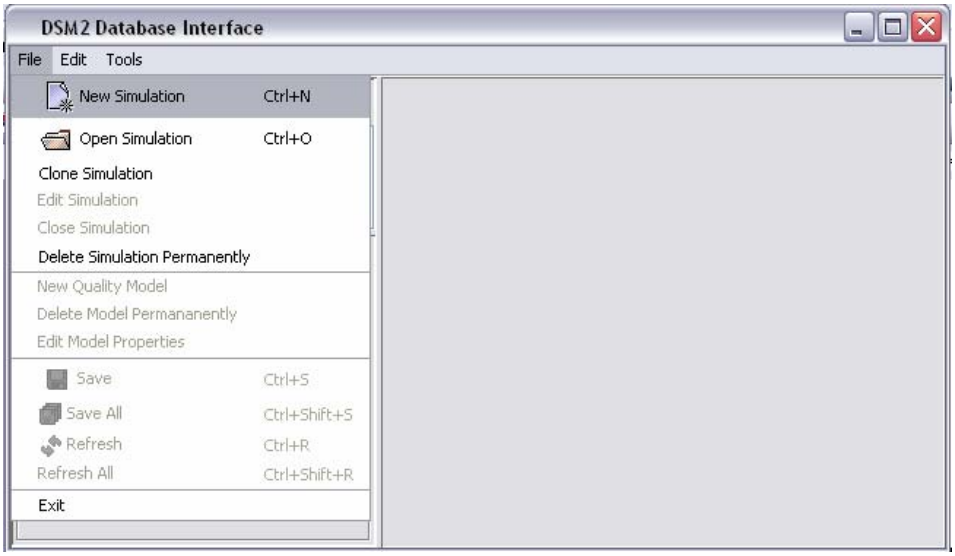
1. **Launch the GUI:**
   a. If you requested a desktop icon, double-click the *DSM2* icon.
   b. Otherwise, the GUI can be launched by navigating to the *{DSM2_home}\bin* directory, and double-clicking *gui.bat*.

2. **Check/Modify the Database Connection:**
   a. In the GUI:
      1) Click on the *Tools* menu.
      2) Select *Configure Database*.
      3) Make certain that the *ODBC source (named in system files)* option is selected.
      4) In the *ODBC DSN* text box, make certain it says: *dsm2input_access*.
   b. In Windows Explorer:

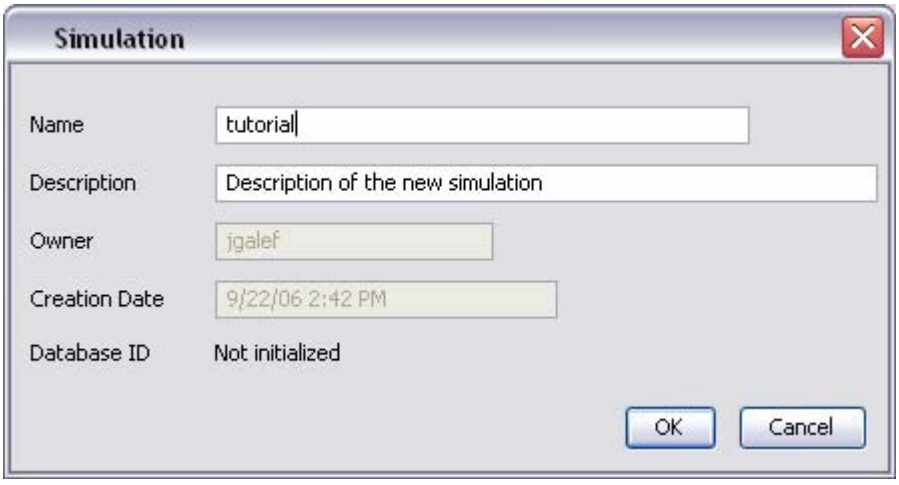
1) Navigate to the directory:
   \{DSM2_home}\tutorial\simulations\simple\t1_channels.
2) Open the file, *hydro.inp*.
   i) Locate the SCALAR section.
   ii) Locate the variable, *dbase_in_tutorial*.
   iii) Change this value to *dsm2input_access*.
3) Repeat this process for the *qual.inp* file.

3. Create a simulation called *tutorial*.

a. Go to *file* and select *New Simulation*.



b. Name the new simulation, *tutorial*, and add a description.



4. In HYDRO, add the Parameter information:

a. In the *Simulations Navigator*:

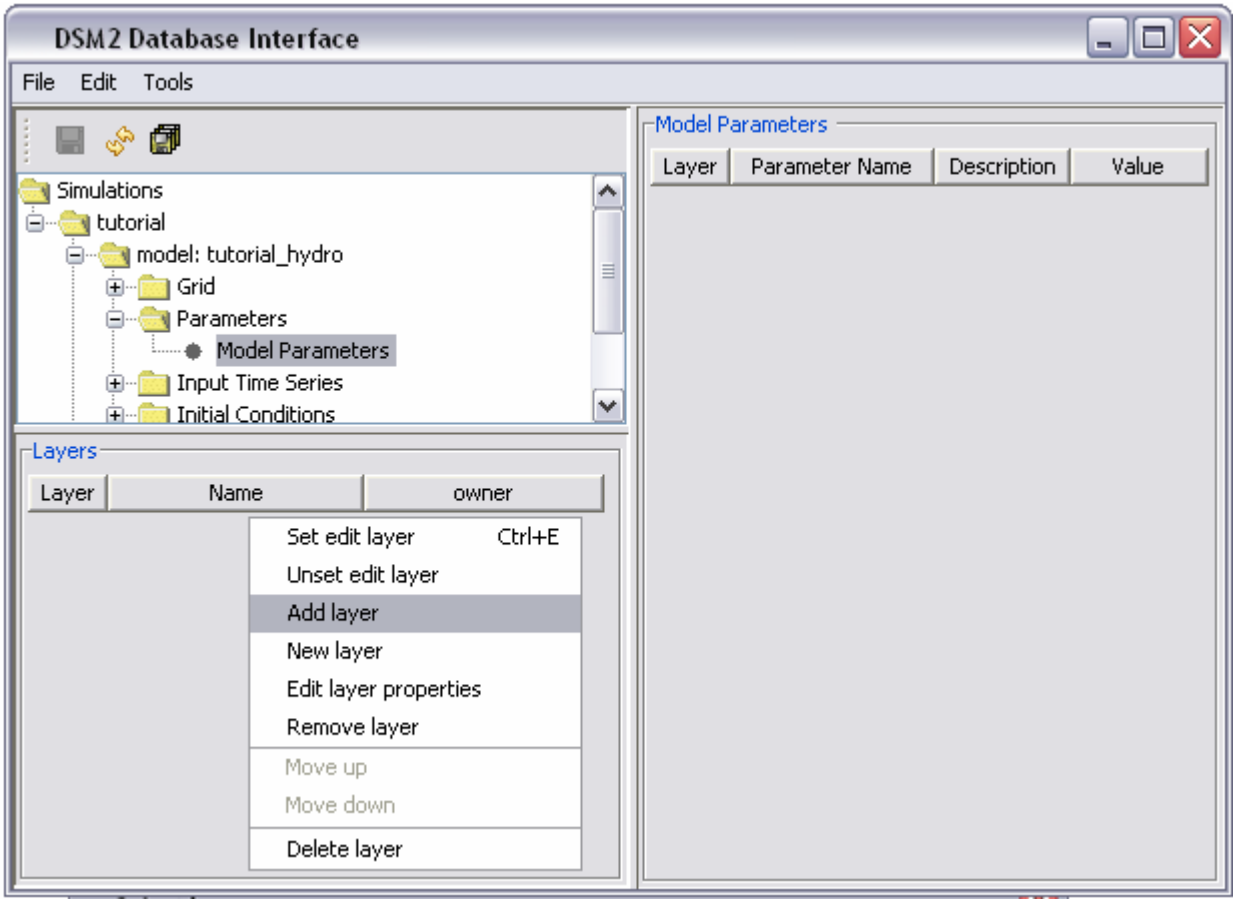
1) Expand the *tutorial* folder.

2) Expand the *model: tutorial_hydro* folder.

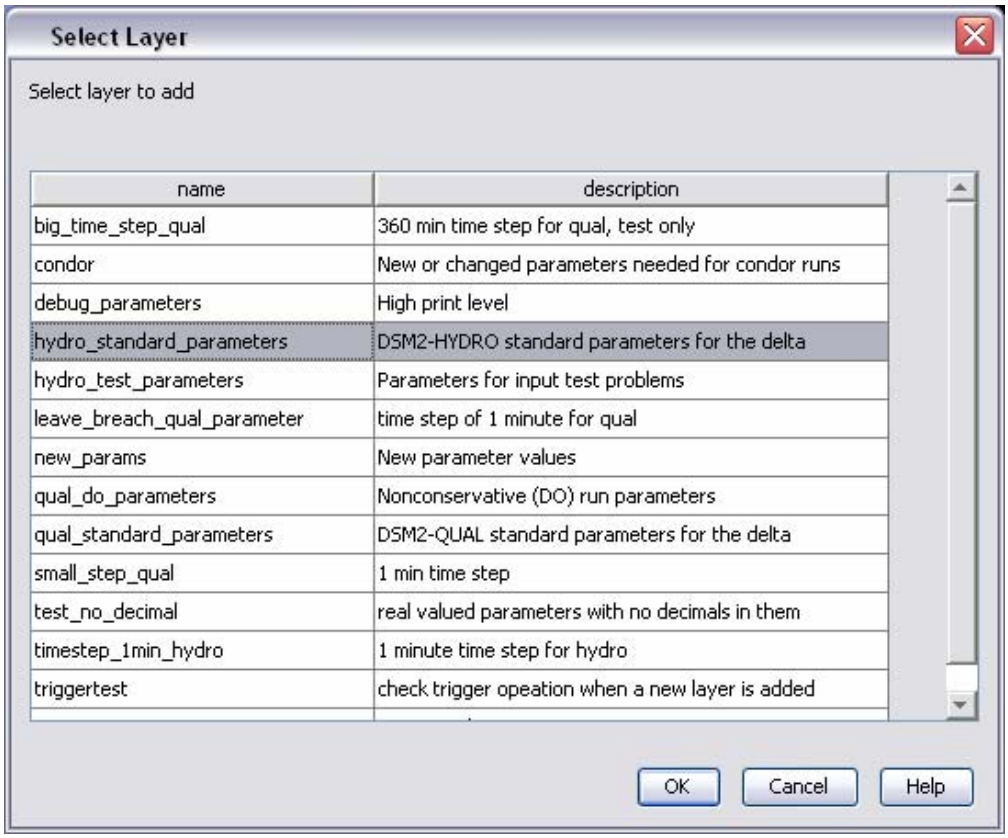
3) Expand the *Parameters* folder.

4) Double-click on *Model Parameters*.

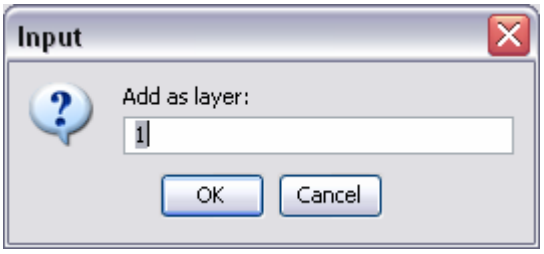
b. In the *Layers Panel*, right-click and select *Add layer* from the menu.



c. In the *Select Layers window*, double-click the *hydro_standard_parameters* layer.



d. In the *Input window*, add as layer *1*.



5. In HYDRO, add Channel information:

a. In the *Simulations Navigator*:

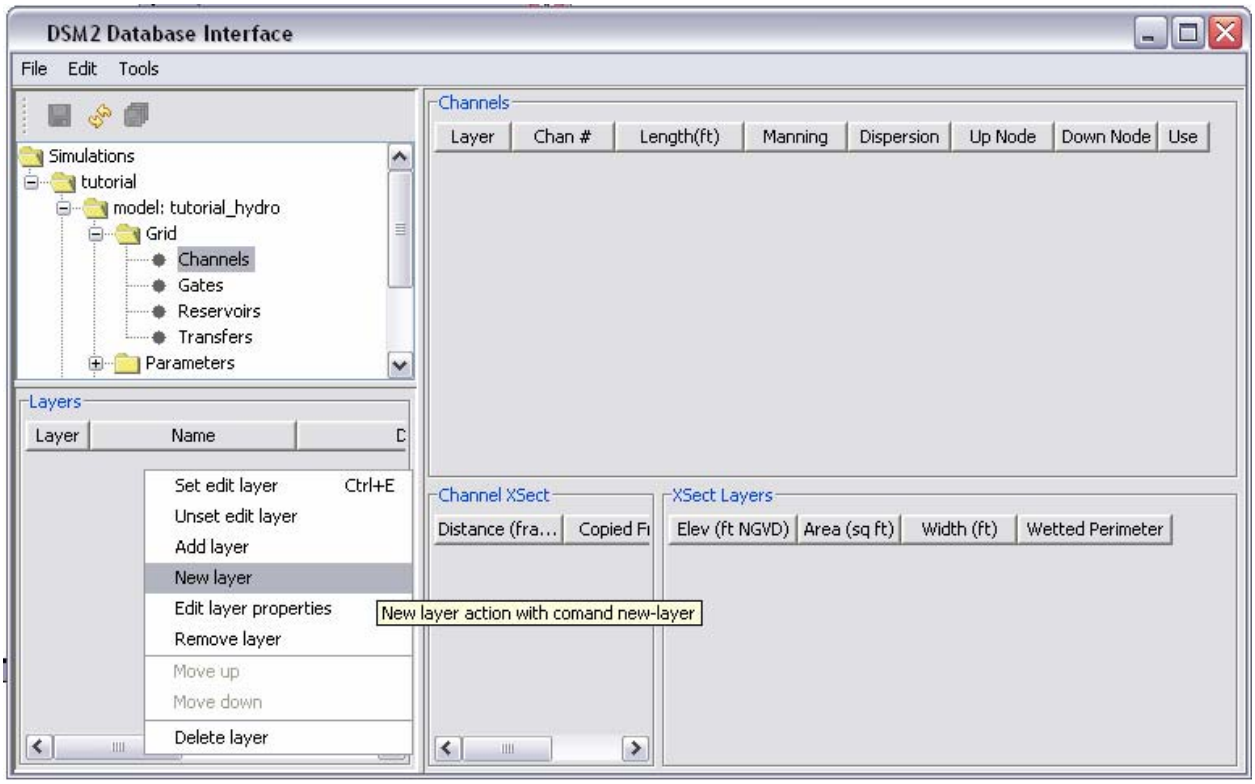
1) Collapse the *Parameters* folder [optional].

2) Expand the *Grid* folder.

3) Double-click on *Channels*.

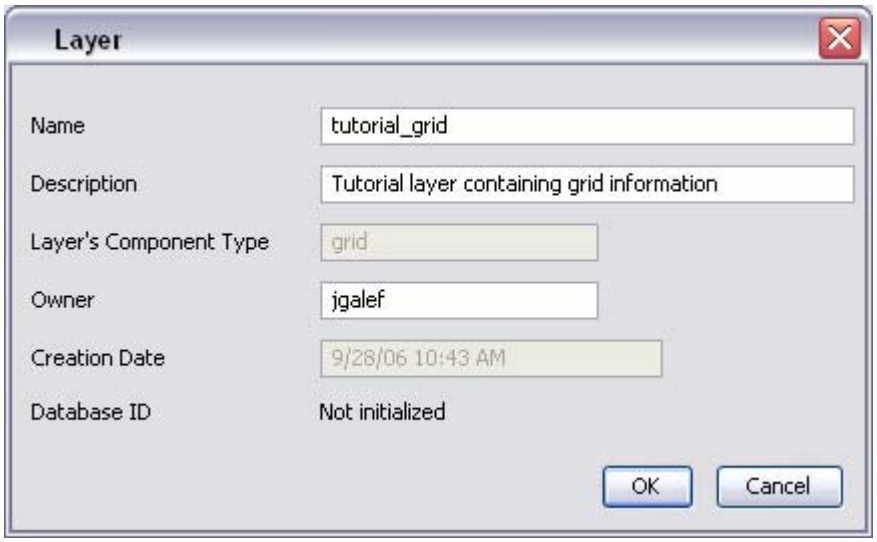
b. Add a Channels Layer:

1) In the *Layers Panel*, right-click and select *New layer* from the menu.



2) In the *Confirm refresh window*, select *Yes*.

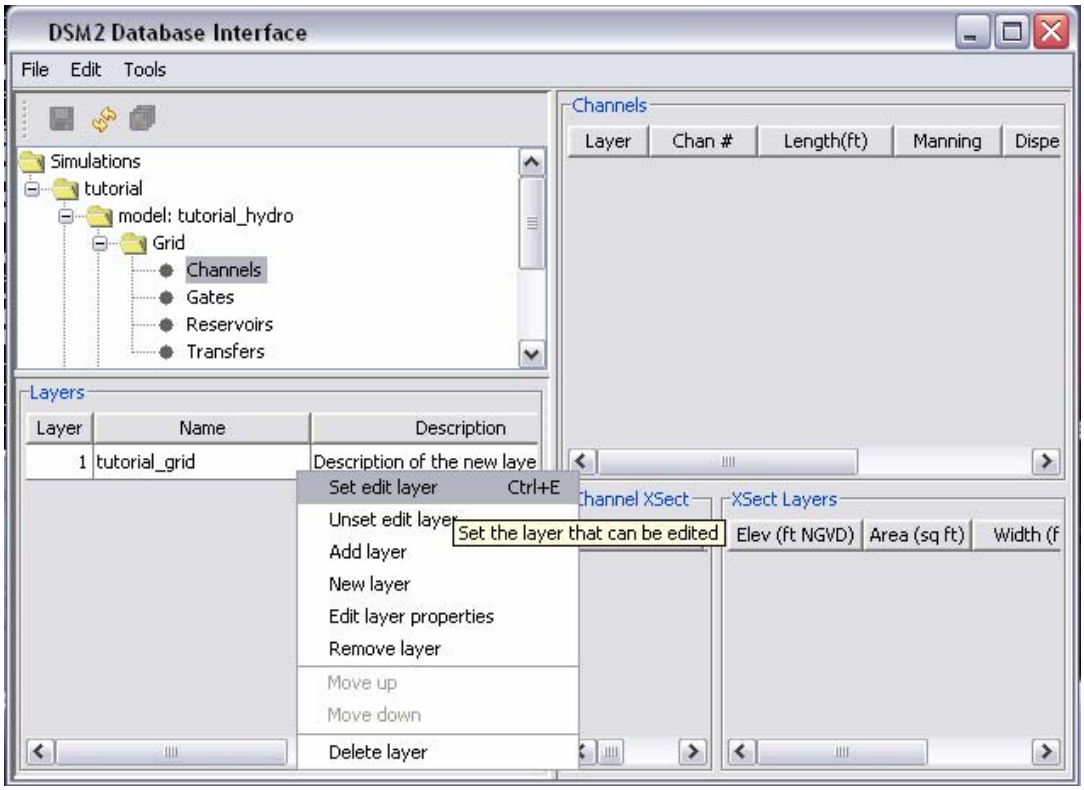
3) In the *Layer window*, name the new layer, *tutorial_grid*, and add a description.



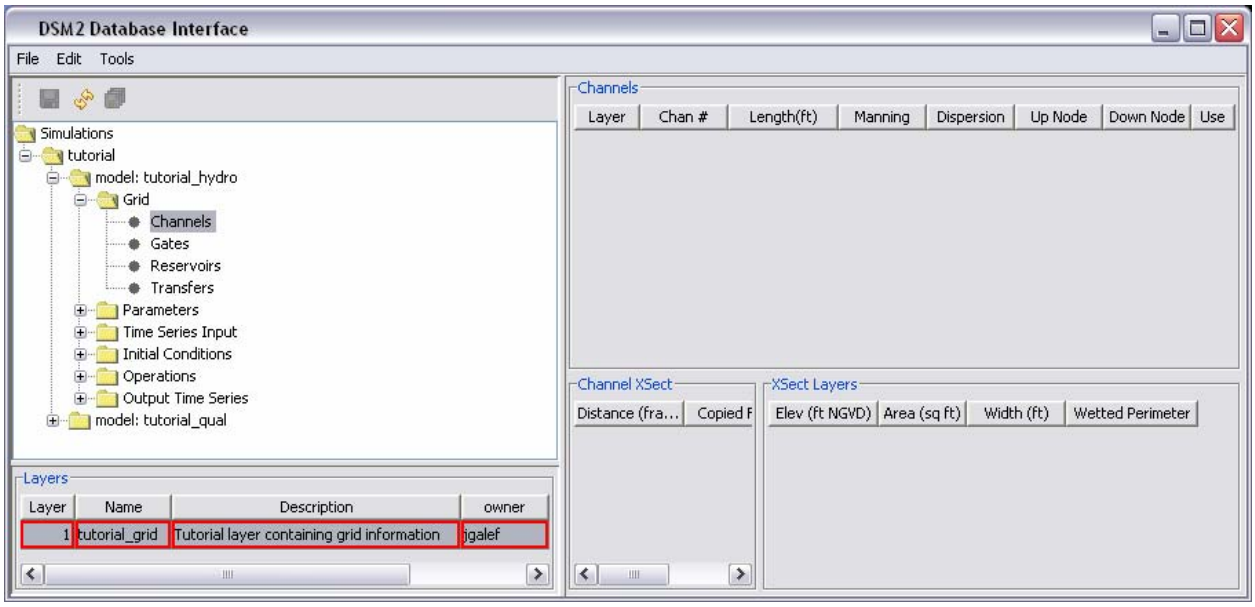
4) In the *Input window*, add as layer *1*.

5) The *tutorial_grid* layer then appears in the *Layers panel*.

c. In the *Layers panel*, right-click and select *Set edit layer*.



d. In the *Select Layers* window, double-click the *tutorial_grid* layer. The layer then appears with a red border in the *Layers panel*.



e. In the *Channels View*, note that there are three tables which require information: *Channels*, *Channel XSect*, and *XSec Layers*. The headers for each of these tables are displayed with blue letters.

f. The following directions involve entering information into these three tables for each of the channels shown in Figure 1.

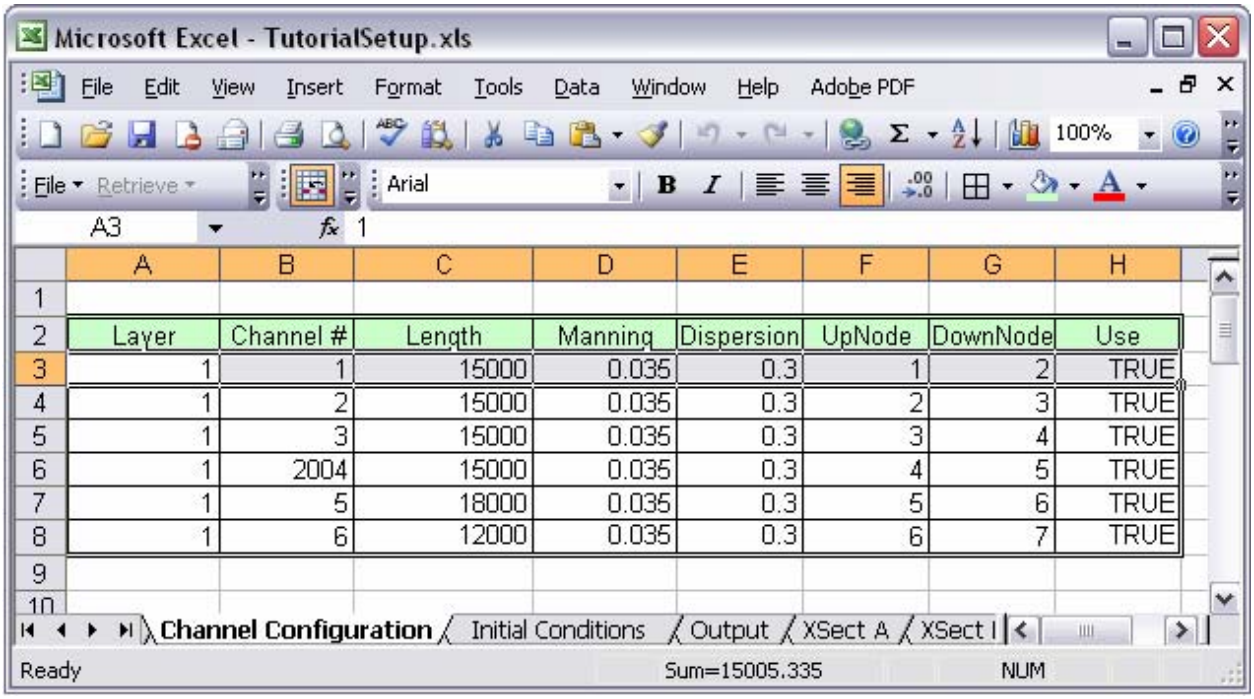
1) In *Windows Explorer*, navigate to the following directory: *dsm2_training\tutorial\simulations\simple*

2) Open the Excel file, *TutorialSetup.xls*, and do the following:

i) Select the *Channel Configuration* tab.

ii) View the channel information and note that Channel 2004 looks mislabeled. This is deliberate, demonstrating that there is no upper limit on channel numbers.

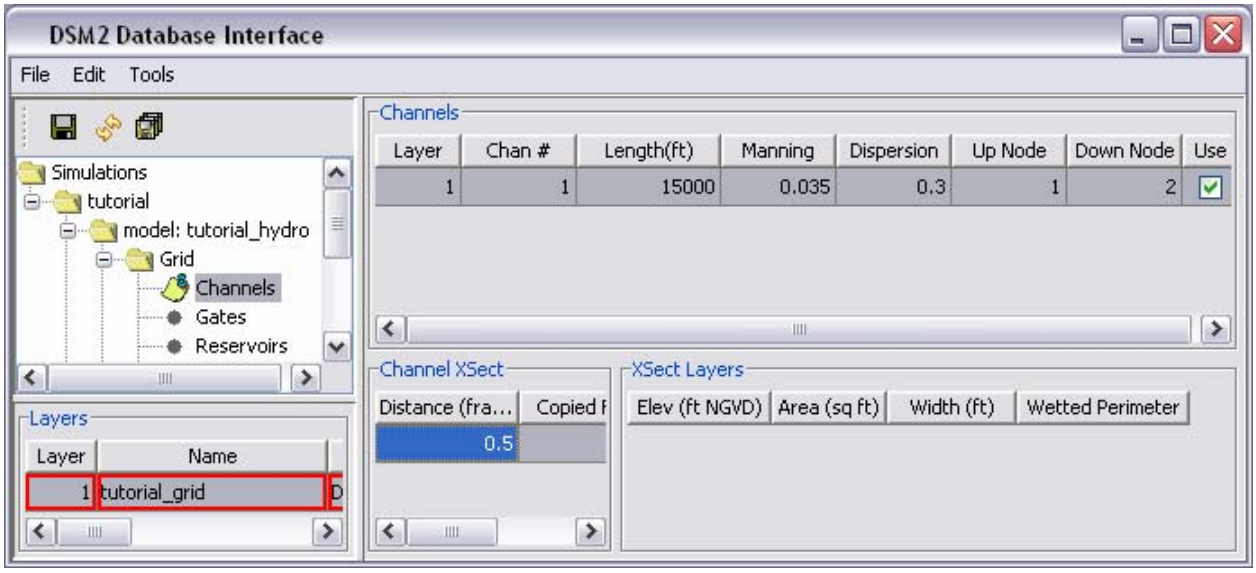
iii) Copy the information from the first row of the table (for *Channel 1*) to the clipboard. Do **not** include headers when copying and pasting.



Layer	Channel #	Length	Manning	Dispersion	UpNode	DownNode	Use
1	1	15000	0.035	0.3	1	2	TRUE
1	2	15000	0.035	0.3	2	3	TRUE
1	3	15000	0.035	0.3	3	4	TRUE
1	2004	15000	0.035	0.3	4	5	TRUE
1	5	18000	0.035	0.3	5	6	TRUE
1	6	12000	0.035	0.3	6	7	TRUE

3) Return to the GUI.
4) In the *Channels table*, left-click and then hit control-v to paste the information.
5) In the *Channel XSect table*:
   i) Right-click and select *Insert row*.

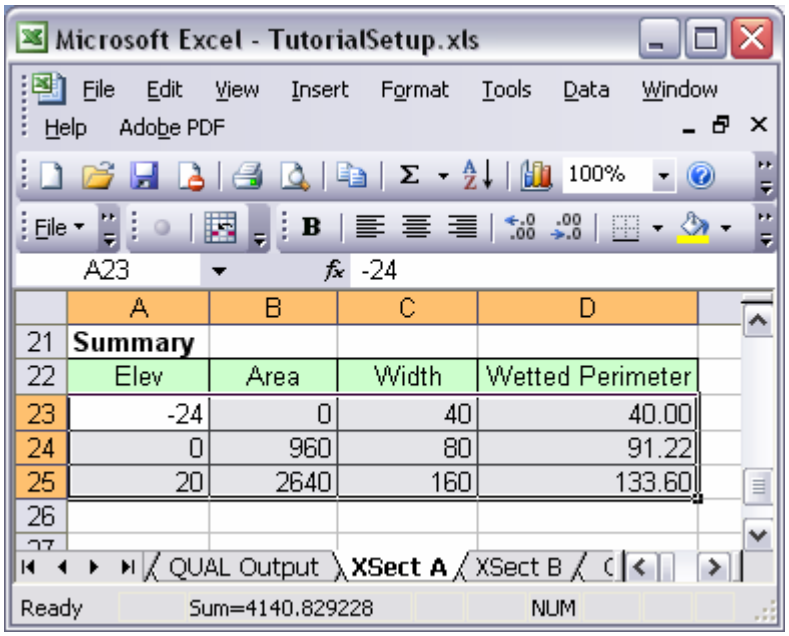
   ii) In the *Distance (fraction)* field, type *0.5* and press *enter*. As seen from the channel configuration diagram, the cross-section is at 0.5 times the length of the channel.



6) Navigate back to the Excel file, *TutorialSetup.xls*.

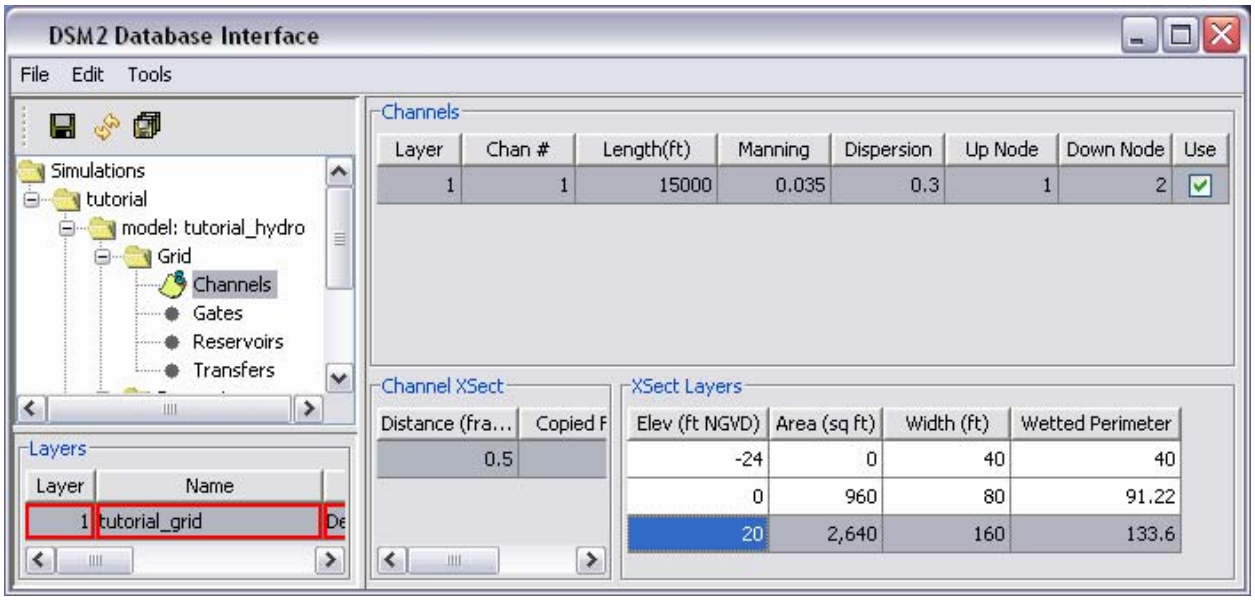
   i) Select the *XSect A* tab.

   ii) Locate the Summary table and copy all of the information to the clipboard. Do **not** include the headers.



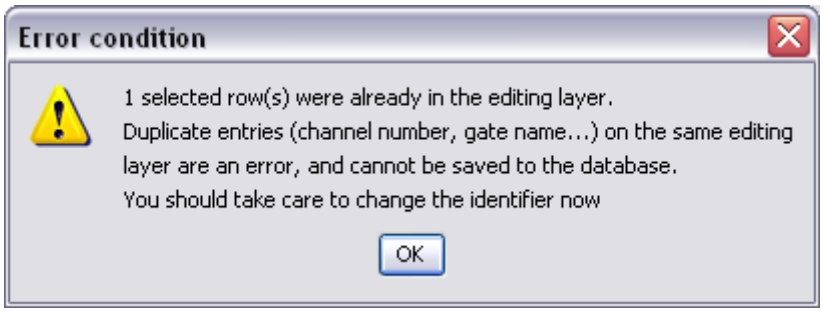
7) Return to the GUI.

8) In the *XSect Layers table*, left-click and then hit control-v to paste the table.



9) In the *Channels table*, right-click the row and select *Copy row to edit layer (with subtables)*.

The following message will appear:



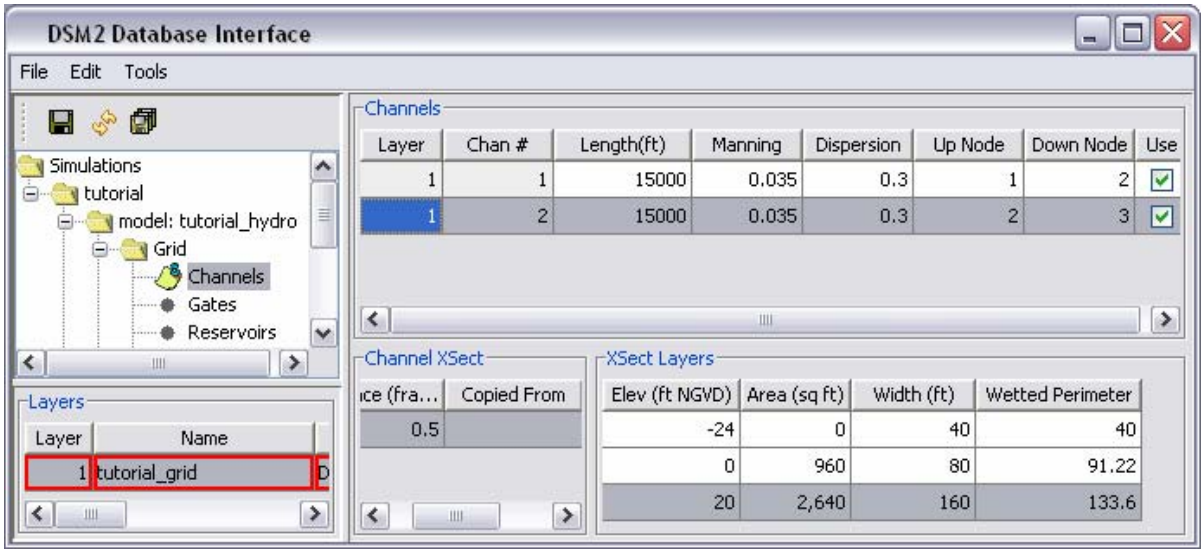
This message warns against duplicate rows. Since the new row will be modified before being saved, there will be no problem, so proceed by clicking *OK*.

10) Navigate back to the Excel file, *TutorialSetup.xls*, and select the *Channel Configuration* tab.

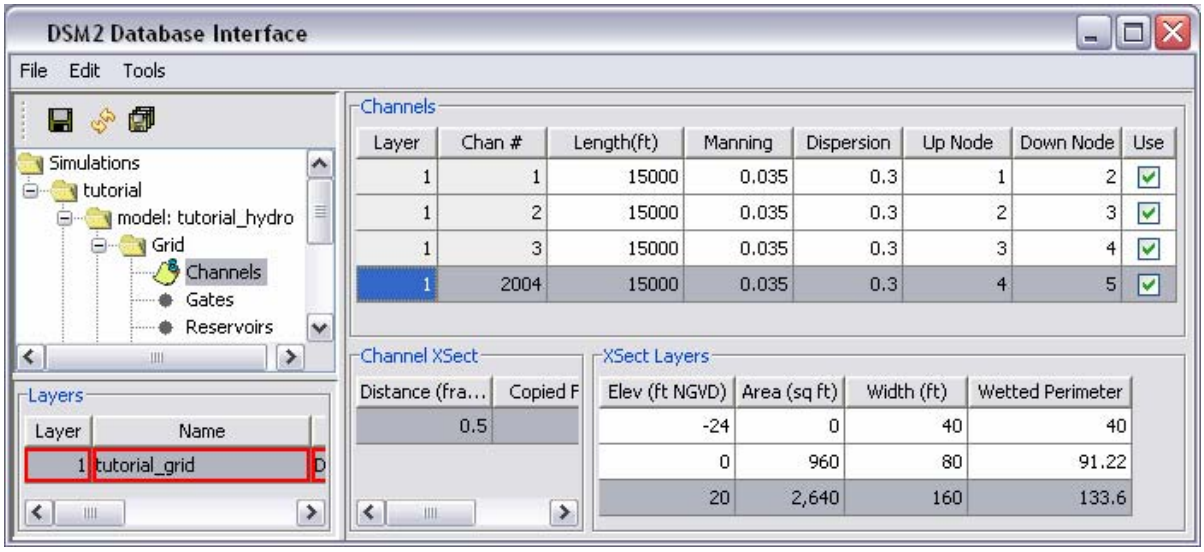
11) View the information in the second row of the table, and note that the information for *Channel 2* is identical to that of *Channel 1*, with the exception of the *Channel, Up Node,* and *Down Node* fields. These numbers are incremented by one.

12) Navigate back to the GUI, and select the second row in the *Channels* table.

13) Make the changes to the *Channel, Up Node,* and *Down Node* fields.



14) Note that the situation is identical for *Channels 3* and *2004*. Repeat steps 18 through 22 with the appropriate information for these two channels. The GUI should then look as follows:



15) Enter information for *Channel 5*:

i) In the *Channels* table, right-click and select *Insert row*.

ii) Fill-in the fields of the new row for *Channel 5*, so that they match those in the *Channel Configuration* table of the Excel file.

iii) Note from Figure 1 that *Channel 5* has two cross-sections; one at 0.2 times the length of the channel, and one at 0.8 times the length of the channel. In the *Channel XSect table*, right-click and select *Insert row*.

iv) In the *Distance (fraction)* column, enter *0.2*.

v) Navigate to the Excel file and select the *XSect A* tab.

vi) Copy the information in the summary table to the clipboard.

vii) Navigate back to the GUI.

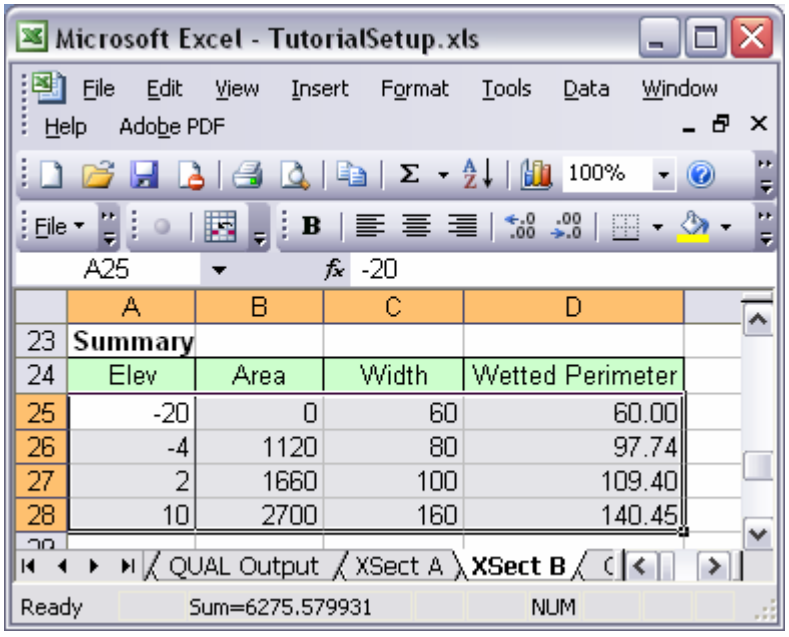
viii) In the *XSect Layers* table, left-click and then hit control-v.

ix) Right-click in the *Channel XSect* table, and select *Insert Row*.

x) In the *Distance (fraction)* column, type *0.8* and press *enter*.

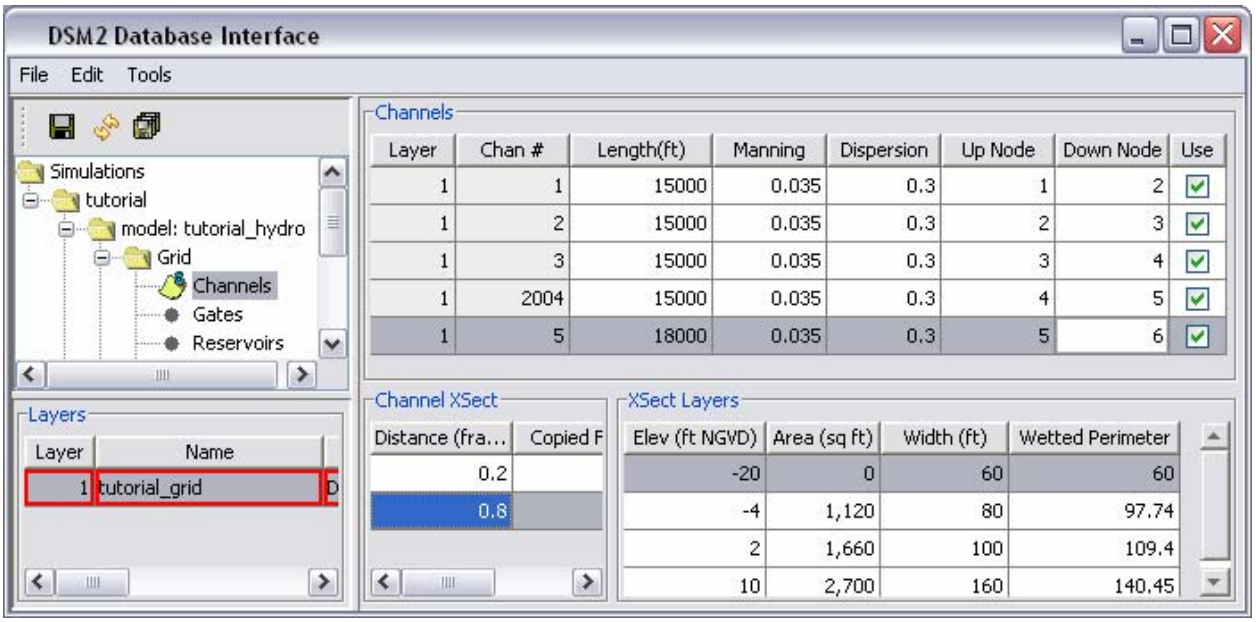
xi) Navigate to the Excel file and select the *XSect B* tab.

xii) Copy the information in the summary table to the clipboard. Do **not** include the headers.



xiii) Navigate back to the GUI.

xiv) In the *XSect Layers* table, left-click and hit control-v.



16) Enter information for *Channel 6*:

i) In the *Channels* table, right-click and select *Insert row*.

ii) Enter the information for *Channel 6* from the *Channel Configuration* tab of the Excel file into this row.

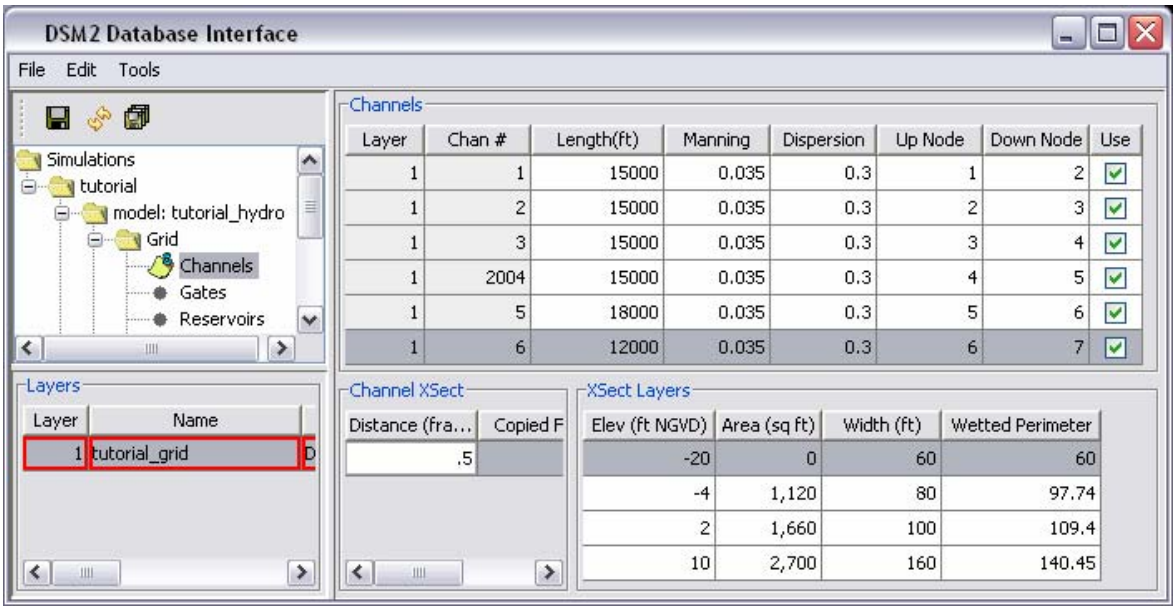
iii) In the *Channel XSect* table, right-click and select *Insert Row*.

iv) In the *Distance (fraction)* column, type *0.5* and press enter.

v) Navigate to the Excel file and select the *XSect B* tab.

vi) Copy the information in the summary table to the clipboard.

vii) In the *XSect Layers* table, left-click and then hit control-v.



17) Save the current settings.

18) In the *Layers panel*, right-click and select *Unset edit layer* [optional].

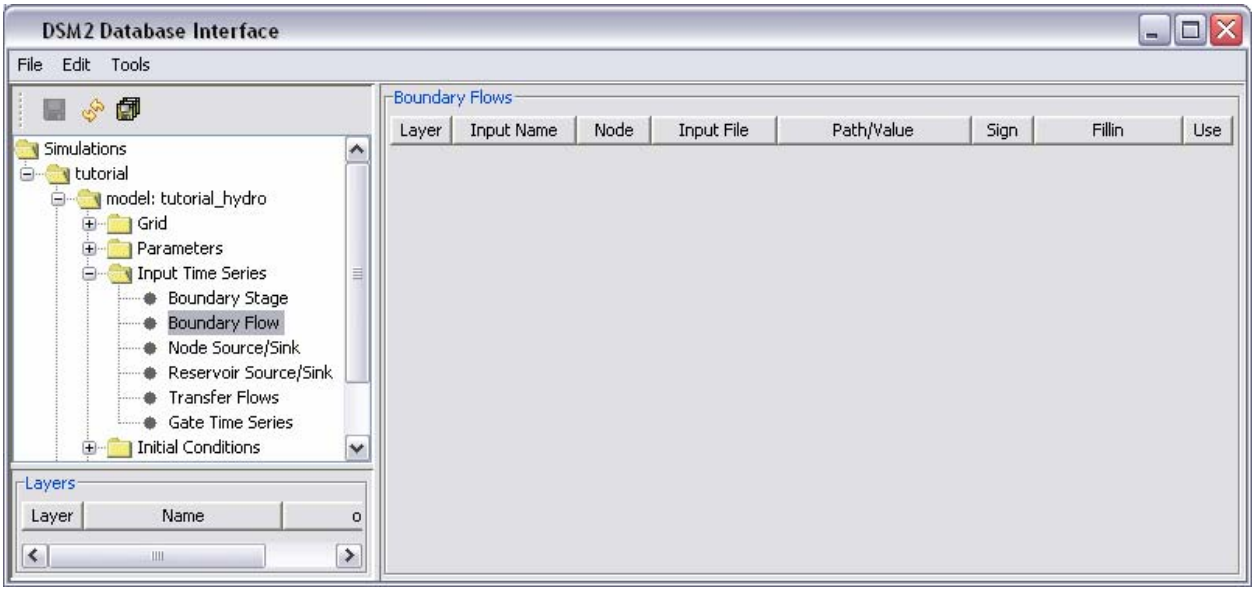
6. In HYDRO, set the Boundary Flow information:

a. In the *Simulations Navigator*:

1) Collapse the *Grid* folder [optional].

2) Expand the *Input Time Series* folder.

3) Double-click on *Boundary Flow*.

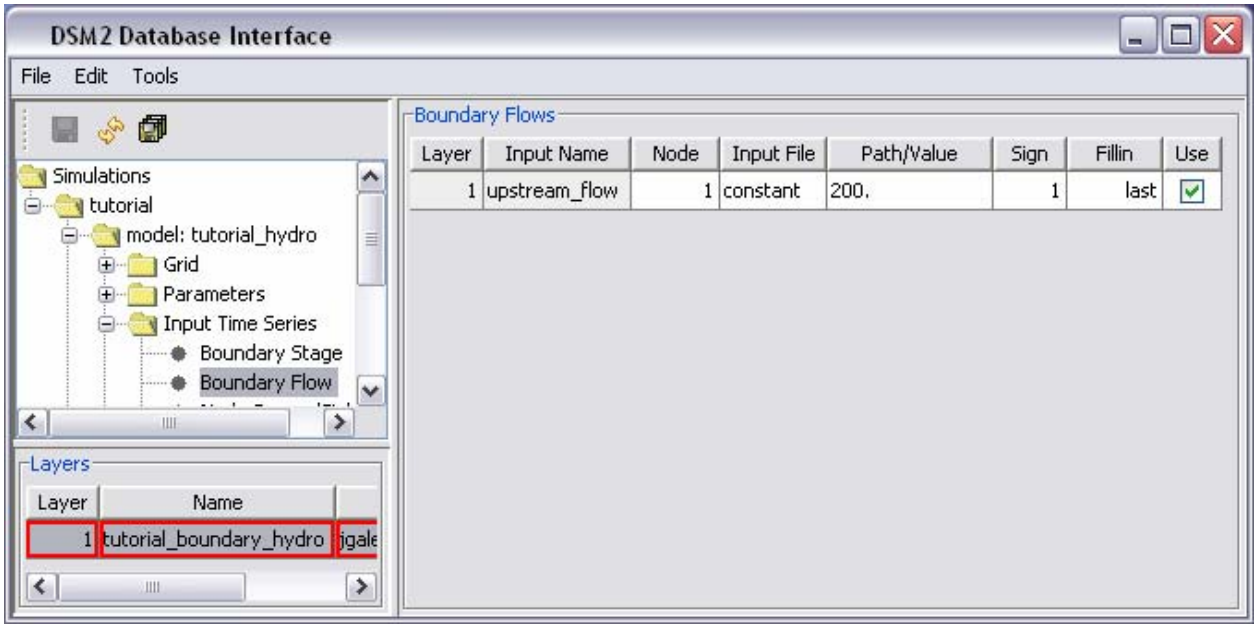


b. Add a HYDRO Boundary Layer:
   1) In the *Layers panel*, right-click and select *New Layer*.
   2) Select *Yes* to confirm the refresh.
   3) Name the new layer, *tutorial_boundary_hydro*, and add a description.
   4) In the *Input window*, add as layer *1*.

c. In the *Layers panel*, right-click and select *Set edit layer*.

d. In the *Select Layers* window, double-click the *tutorial_boundary_hydro* layer.

e. In the *Boundary Flows* table:
   1) Right-click and click *Insert row*.
   2) In the new row, enter the following values into the correct fields:
      i) Input Name: *upstream_flow*
      ii) Node: *1*
      iii) Input File: *constant*
      iv) Path/Value: *200.*
      iv) Sign: *1*
      v) Fillin: *Last*
      vi) Use: Make sure that the row contains a check-mark.



b. Save the current settings.

c. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

2. In HYDRO, set the Boundary Stage information:

a. In the *Simulations Navigator*:

1) Remain in the *Time Series Input* folder.

2) Double-click on *Boundary Stage*.

b. In the *Layers panel*, right-click and select *Set edit layer*.

c. In the *Select Layers* window, double-click the *tutorial_boundary_hydro* layer.

d. In the *Boundary Stage table*:

1) Right-click and select *Insert Row*.

2) In the new row, enter the following values into the appropriate fields:

i) Input Name: *downstream_stage*

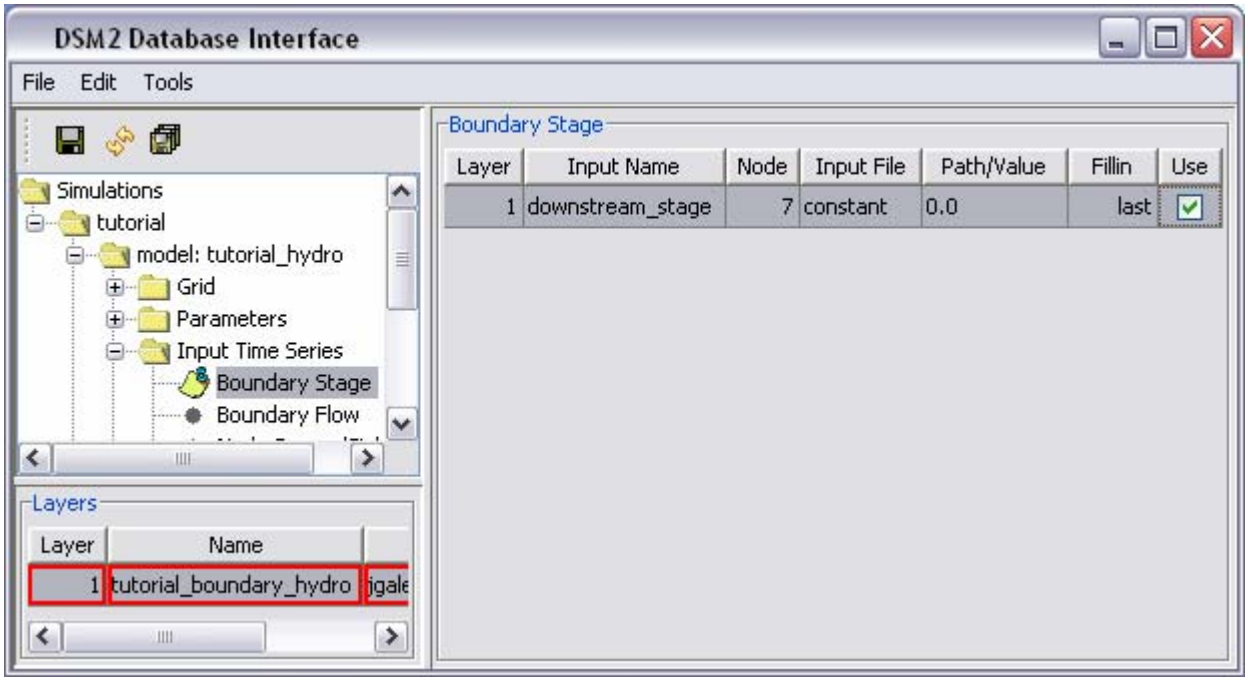
ii) Node: *7*

iii) Input File: *constant*

iv) Path/Value: *0.0*

v) Fillin: *Last*

vi) Use: Make sure that the entry contains a checkmark.



b. Save the current settings.

c. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

2. In HYDRO, set the Initial Conditions for stage and flow:

For each of the channels, the stage and flow will be set *0*. These *0*-values will be applied at both the *0* and *length* distances along the channel. With seven channels, two variables, and two locations to set the values, there will be a total of 28 rows. To save time, this information has been provided in the accompanying Excel spreadsheet.

a. In the *Simulations Navigator*:

1) Collapse the *Input Time Series* folder [optional].

2) Expand the *Initial Conditions* folder.

3) Double-click on *Channel IC*.

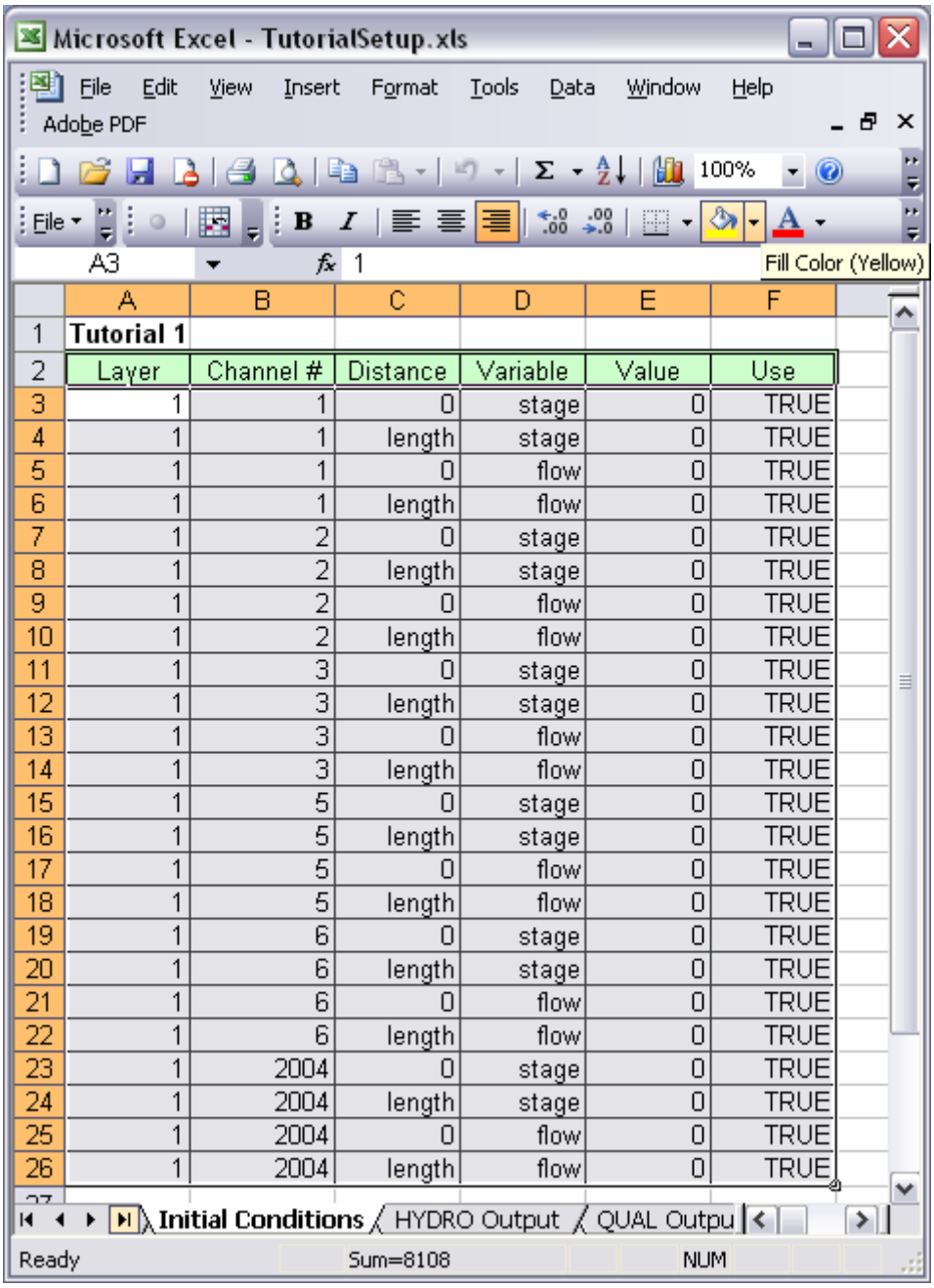
b. In the *Layers panel*, right-click and select *Set edit layer*.

c. In the *Select Layer* window, double-click the layer, *tutorial_grid*.

d. Navigate back to the Excel spreadsheet, *TutorialSetup.xls*.

1) Select the *Initial Conditions* tab.

2) Copy the information from the table to the clipboard. Do **not** include the headers.



Microsoft Excel - TutorialSetup.xls

File Edit View Insert Format Tools Data Window Help

Adobe PDF

100%

File

A3 1

Fill Color (Yellow)

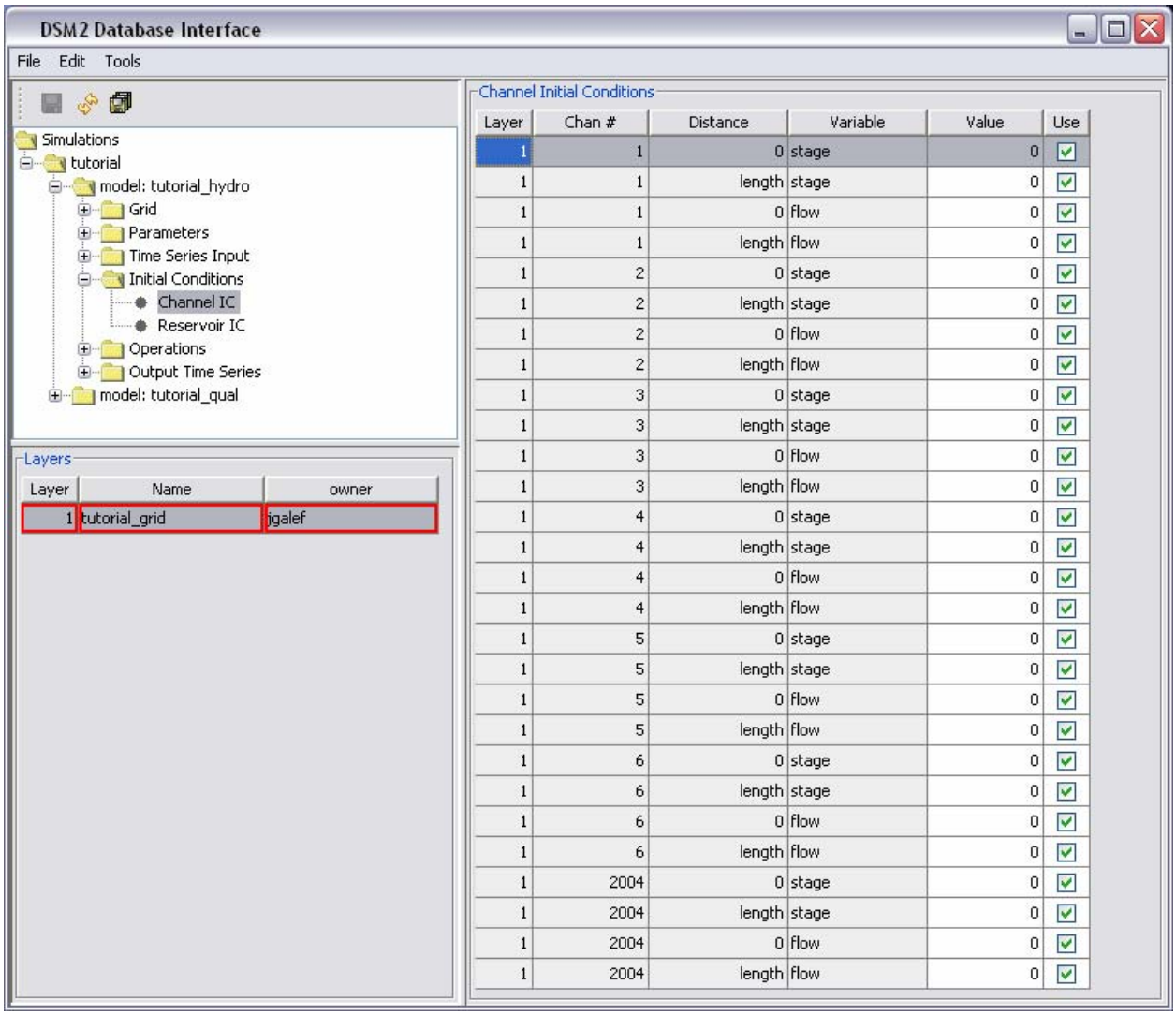
	A	B	C	D	E	F
1	**Tutorial 1**					
2	Layer	Channel #	Distance	Variable	Value	Use
3	1	1	0	stage	0	TRUE
4	1	1	length	stage	0	TRUE
5	1	1	0	flow	0	TRUE
6	1	1	length	flow	0	TRUE
7	1	2	0	stage	0	TRUE
8	1	2	length	stage	0	TRUE
9	1	2	0	flow	0	TRUE
10	1	2	length	flow	0	TRUE
11	1	3	0	stage	0	TRUE
12	1	3	length	stage	0	TRUE
13	1	3	0	flow	0	TRUE
14	1	3	length	flow	0	TRUE
15	1	5	0	stage	0	TRUE
16	1	5	length	stage	0	TRUE
17	1	5	0	flow	0	TRUE
18	1	5	length	flow	0	TRUE
19	1	6	0	stage	0	TRUE
20	1	6	length	stage	0	TRUE
21	1	6	0	flow	0	TRUE
22	1	6	length	flow	0	TRUE
23	1	2004	0	stage	0	TRUE
24	1	2004	length	stage	0	TRUE
25	1	2004	0	flow	0	TRUE
26	1	2004	length	flow	0	TRUE

Initial Conditions HYDRO Output QUAL Outpu

Ready Sum=8108 NUM

e. Return to the GUI.

f. In the *Channels table*, left-click and hit control-v to paste the initial conditions information from Excel.



g. Save the current settings.

h. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

3. In HYDRO, Specify the Output Locations:

A new layer will be created for the output locations. These locations will include the two boundaries, two locations along Channel 2, and the beginning of Channel 2004. The output variables will include both stage and flow.

a. In the *Simulations Navigator*:

1) Collapse the *Initial Conditions* folder [optional].

2) Expand the *Output Time Series* folder.

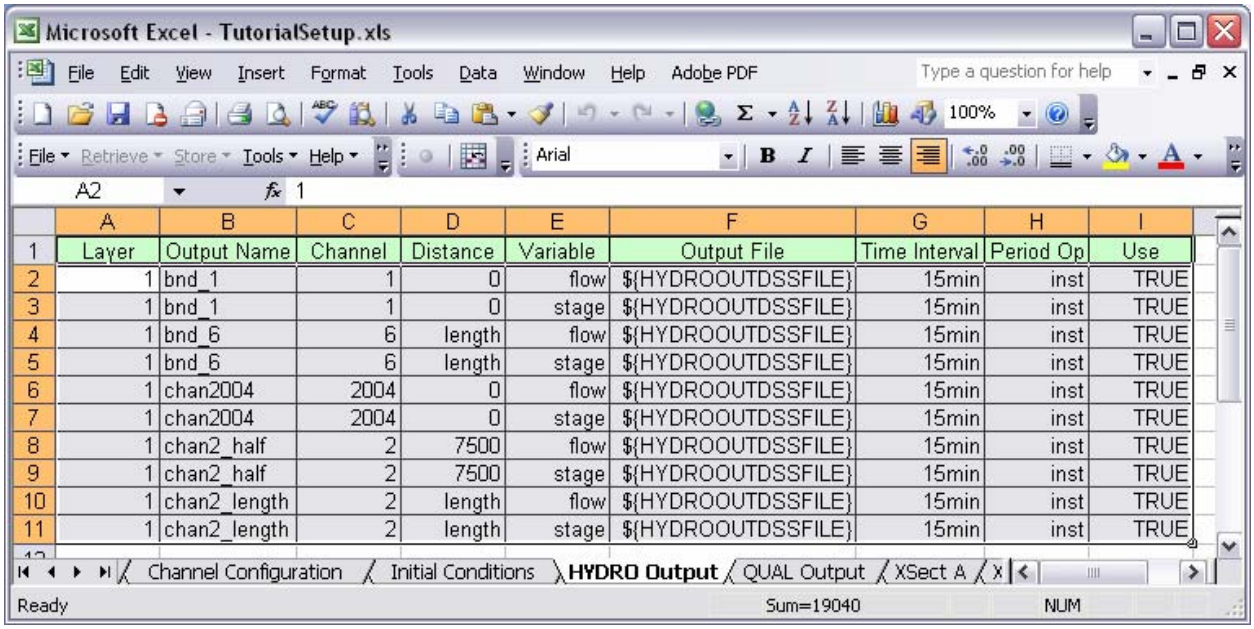
3) Double-click on *Channel Output*.

b. Create a new Output Layer:
   1) In the *Layers panel*, right-click and select *New layer* from the menu.
   2) Select *Yes* to confirm the refresh.
   3) Name the new layer, *tutorial_output*, and add a description.
   4) In the *Input window*, add as layer *1*.

c. In the *Layers panel*, right-click and select *Set edit layer*.

d. In the *Select Layers* window, double-click the *tutorial_output* layer.

e. Navigate back to the Excel spreadsheet, *TutorialSetup.xls*.
   1) Select the *HYDRO Output* tab.
   2) Copy the information from the table to the clipboard. Do **not** include the headers.

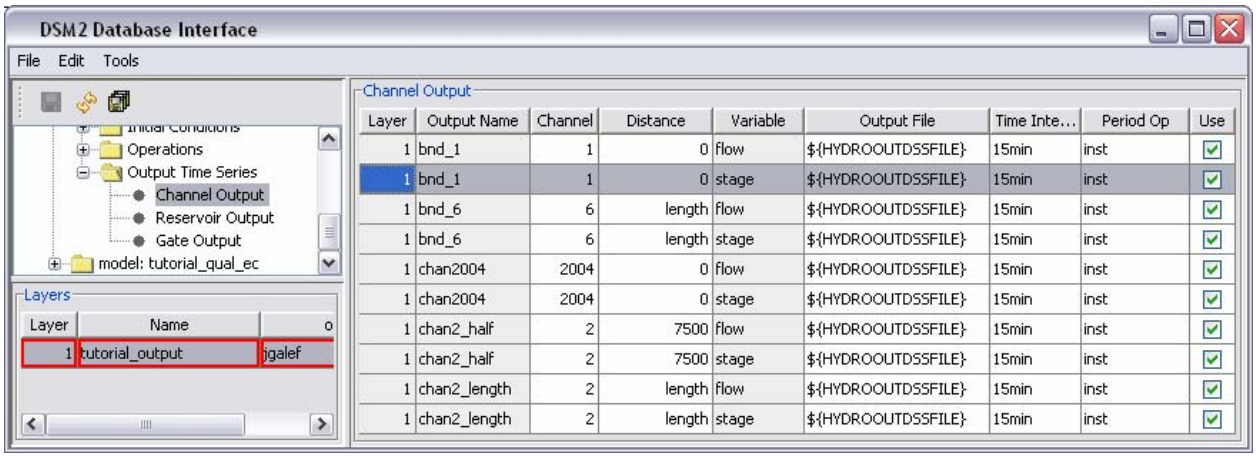


	A	B	C	D	E	F	G	H	I
1	Layer	Output Name	Channel	Distance	Variable	Output File	Time Interval	Period Op	Use
2	1	bnd_1	1	0	flow	${HYDROOUTDSSFILE}	15min	inst	TRUE
3	1	bnd_1	1	0	stage	${HYDROOUTDSSFILE}	15min	inst	TRUE
4	1	bnd_6	6	length	flow	${HYDROOUTDSSFILE}	15min	inst	TRUE
5	1	bnd_6	6	length	stage	${HYDROOUTDSSFILE}	15min	inst	TRUE
6	1	chan2004	2004	0	flow	${HYDROOUTDSSFILE}	15min	inst	TRUE
7	1	chan2004	2004	0	stage	${HYDROOUTDSSFILE}	15min	inst	TRUE
8	1	chan2_half	2	7500	flow	${HYDROOUTDSSFILE}	15min	inst	TRUE
9	1	chan2_half	2	7500	stage	${HYDROOUTDSSFILE}	15min	inst	TRUE
10	1	chan2_length	2	length	flow	${HYDROOUTDSSFILE}	15min	inst	TRUE
11	1	chan2_length	2	length	stage	${HYDROOUTDSSFILE}	15min	inst	TRUE

f. Return to the GUI.

g. In the *Channel Output* table, left-click and hit control-v to paste the *Channel Output Locations* information from Excel.

h. The GUI should now look as follows:



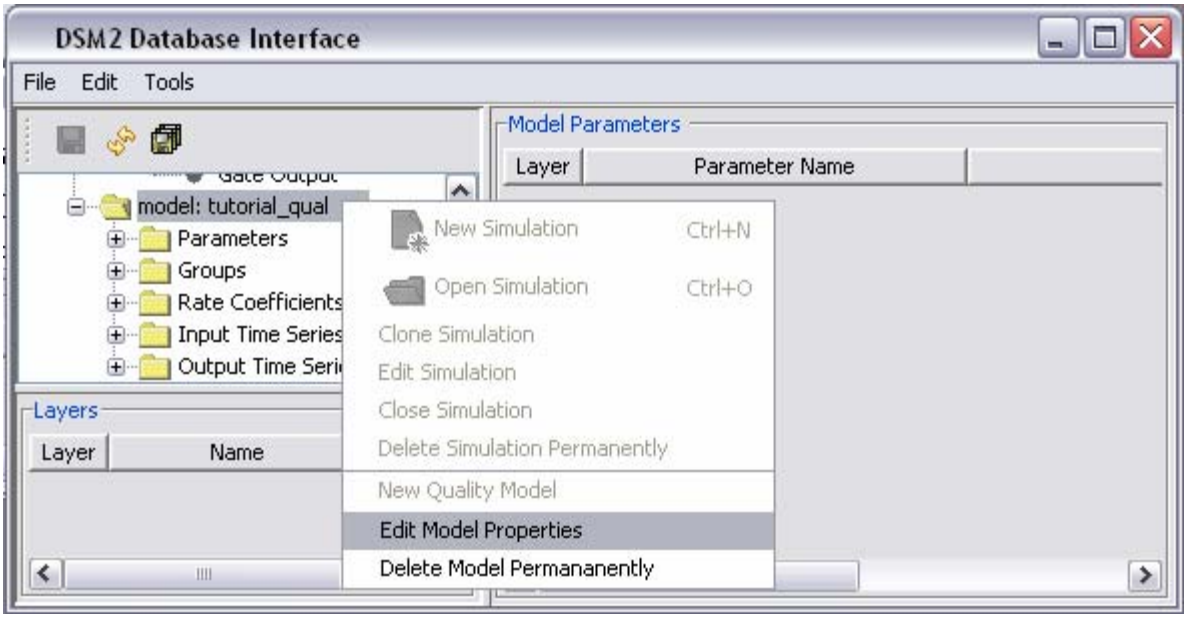
i. Save the current settings.

j. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

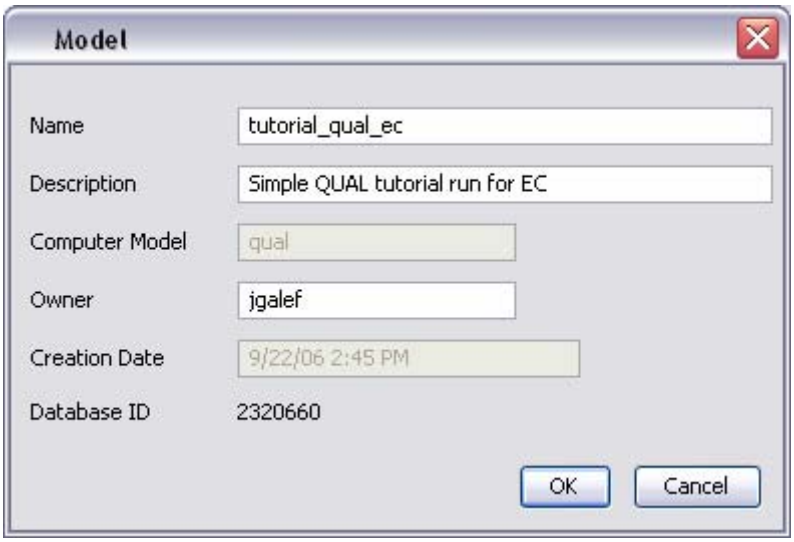
4. Rename QUAL Model

By default, the QUAL model is named *tutorial_qual*.

a. In the *Sessions Navigator*, right-click on *model:tutorial_qual* and select *Edit model properties*.



b. In the *Model Window, c*hange the name of the model to *tutorial_qual_ec*, and add a description.



2. In QUAL, add the Parameter information:

a. In the *Simulations Navigator*:

1) Collapse the *model: tutorial_hydro* folder [optional].
2) Expand the *model: tutorial_qual-ec* folder.
3) Expand the *Parameters* folder.
4) Double-click on *Model Parameters*.

b. In the *Layers panel*, right-click and select *Add layer* from the menu.

c. In the *Select Layer* window, double-click the *qual_standard_parameters* layer.

d. In the *Input window*, add as layer *1*.

3. In QUAL, set the Boundary Concentration information:

a. In the *Simulations Navigator*:

1) Collapse the *Parameters* folder [optional].
2) Expand the *Input Time Series* folder.
3) Double-click on *Boundary Concentration*.

b. Add a QUAL Boundary Layer:

1) In the *Layers panel*, right-click and select *New layer*.
2) Click *yes* to confirm the refresh.

3) Name the layer, *tutorial_boundary_qual_ec*, and provide a description.

4) In the *Input window*, add as layer *1*.

c. In the *Layers panel*, right-click and select *Set edit layer*.

d. In the *Select Layer* window, double-click the *tutorial_boundary_qual_ec* layer.

e. In the *Boundary Concentration* table, add an upstream concentration row:

1) Right-click and select *Insert row*.

2) In the new row, enter the following information into the appropriate fields:

i) Input Name: *upstream_flow*.

ii) Node: *1*.

iii) Variable: *ec*.

iv) Input File: *constant*.

v) Path/Value: *200.*

vi) Sign: *1*.

vii) Fillin: *last*.

viii) Use: Make sure that the entry contains a checkmark.

f. In the *Boundary Concentration* table, add a downstream concentration row:

1) Right-click and select *Insert row*.

2) In the newest row, enter the following information into the appropriate fields:

i) Input Name: *downstream_stage*.

ii) Node: *7*.

iii) Variable: *ec*.

iv) Input File: *constant*.

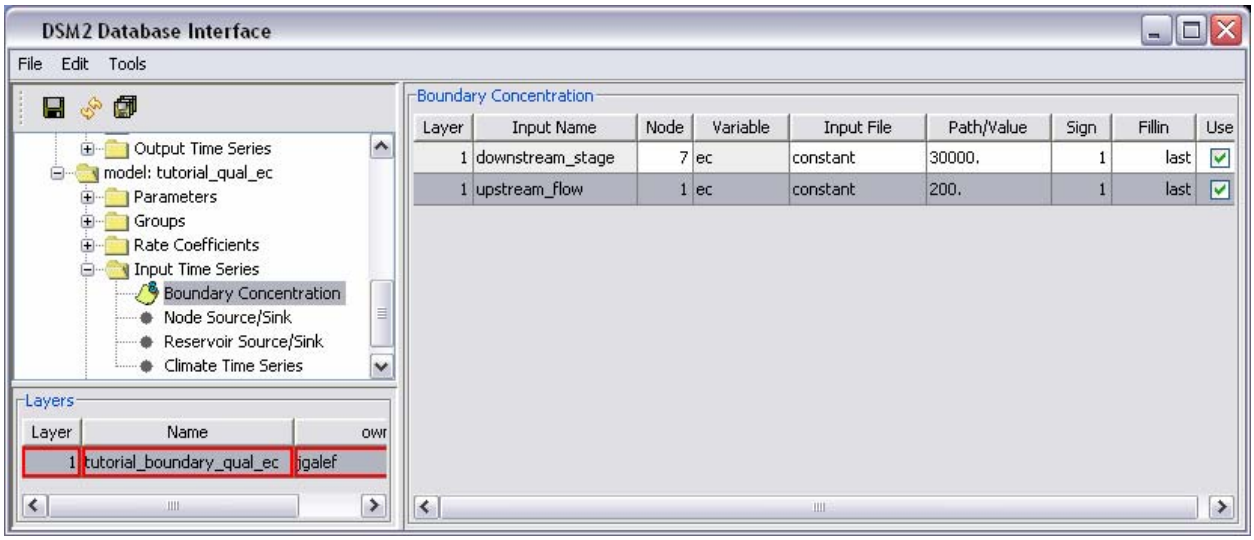
v) Path: *30000.*

vi) Sign: *1*.

vii) Fillin: *last*.

viii) Use: Make sure that the entry contains a checkmark.

g. When complete, the interface should look as follows:



h. Save the current settings.

i. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

4. In QUAL, Specify the Output Locations:

A new layer will be created for the output locations. These locations will include the two boundaries, two locations along Channel 2, and the beginning of Channel 2004. The output variable will be *ec*.

a. In the *Simulations Navigator*:
   1) Collapse the *Input Time Series* folder [optional].
   2) Expand the *Output Time Series* folder.
   3) Double-click on *Channel Output*.

b. Create a QUAL Output Layer:
   1) In the *Layers panel*, right-click and select *New layer*.
   2) Select *Yes* to confirm the refresh.
   3) Name the new layer, *tutorial_output_ec*, and add a description of the new layer.
   4) Enter *1* for the layer number.

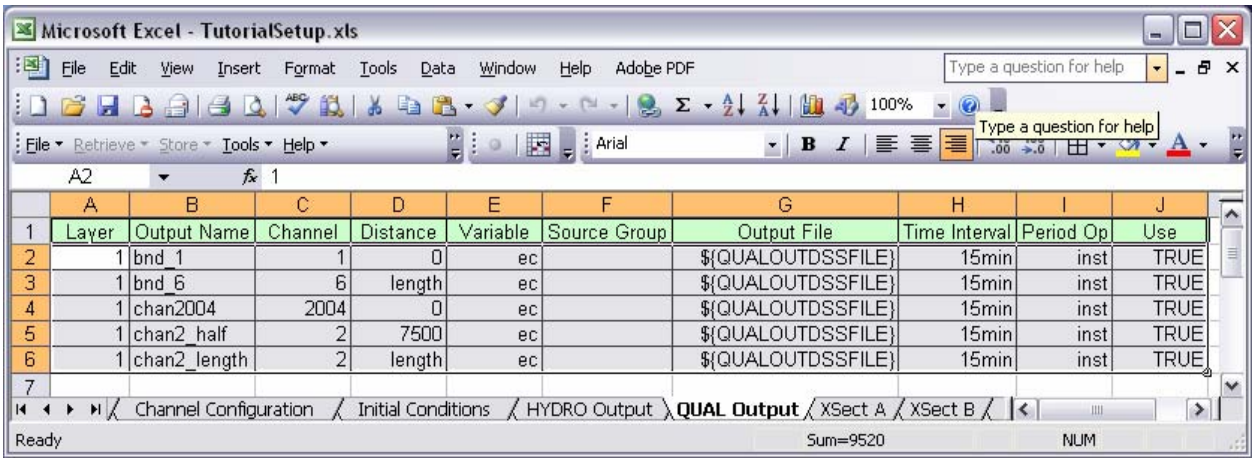
c. In the *Layers panel*, right-click and select *Set edit layer*.

d. In the *Select Layer* window, double-click the *tutorial_output_ec* layer.

e. Navigate back to the Excel spreadsheet, *TutorialSetup.xls*.

1) Select the *QUAL Output* tab.

2) Copy the information from the table to the clipboard. Do **not** include the headers.

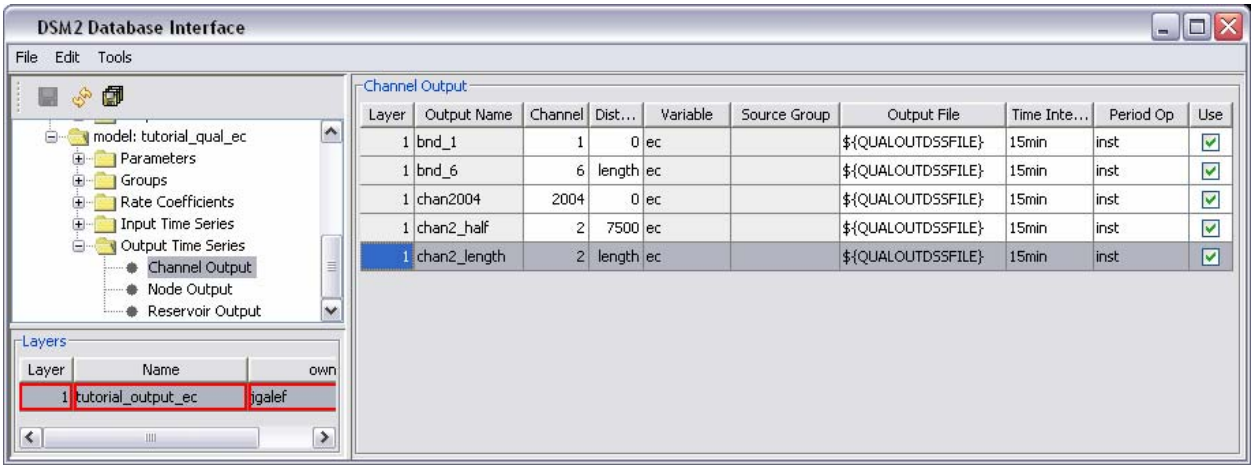


	A	B	C	D	E	F	G	H	I	J
1	Layer	Output Name	Channel	Distance	Variable	Source Group	Output File	Time Interval	Period Op	Use
2	1	bnd_1	1	0	ec		${QUALOUTDSSFILE}	15min	inst	TRUE
3	1	bnd_6	6	length	ec		${QUALOUTDSSFILE}	15min	inst	TRUE
4	1	chan2004	2004	0	ec		${QUALOUTDSSFILE}	15min	inst	TRUE
5	1	chan2_half	2	7500	ec		${QUALOUTDSSFILE}	15min	inst	TRUE
6	1	chan2_length	2	length	ec		${QUALOUTDSSFILE}	15min	inst	TRUE
7										

f. Return to the GUI.

g. In the *Channel Output* table, left-click and hit control-v to paste the output locations information from Excel.

h. The GUI should now look as follows:



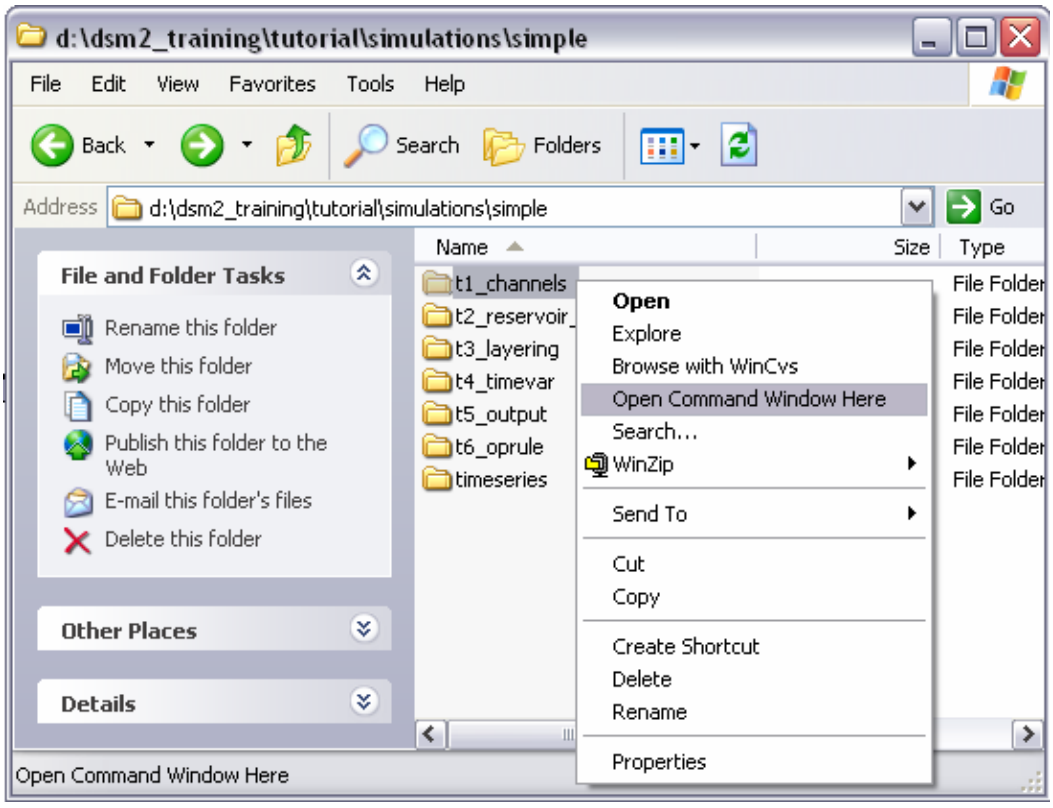
i. Save the current settings.

j. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

5. Running HYDRO and QUAL

a. In Windows Explorer, navigate to the directory: *{DSM2_home}**tutorial**simulations**simple*\.

b. Right-click on the directory, *t1_channels*, and select *Open Command Window Here*.



c. In the command window, type: *hydro hydro.inp*.

d. HYDRO will then run and create an *output.dss* file in the same directory.

e. In the command window, type: *qual qual.inp*.

f. QUAL will then run and add output to the *output.dss* file.

g. Open the *output.dss* file and examine the results.

III. Tutorial 2: Reservoir_Gate_Transfer

The purpose of this tutorial is to add a reservoir, gate, and transfer to the simple channel-only grid created in Tutorial 1. As shown in the PowerPoint presentation, the channels have the following configuration and specifications:

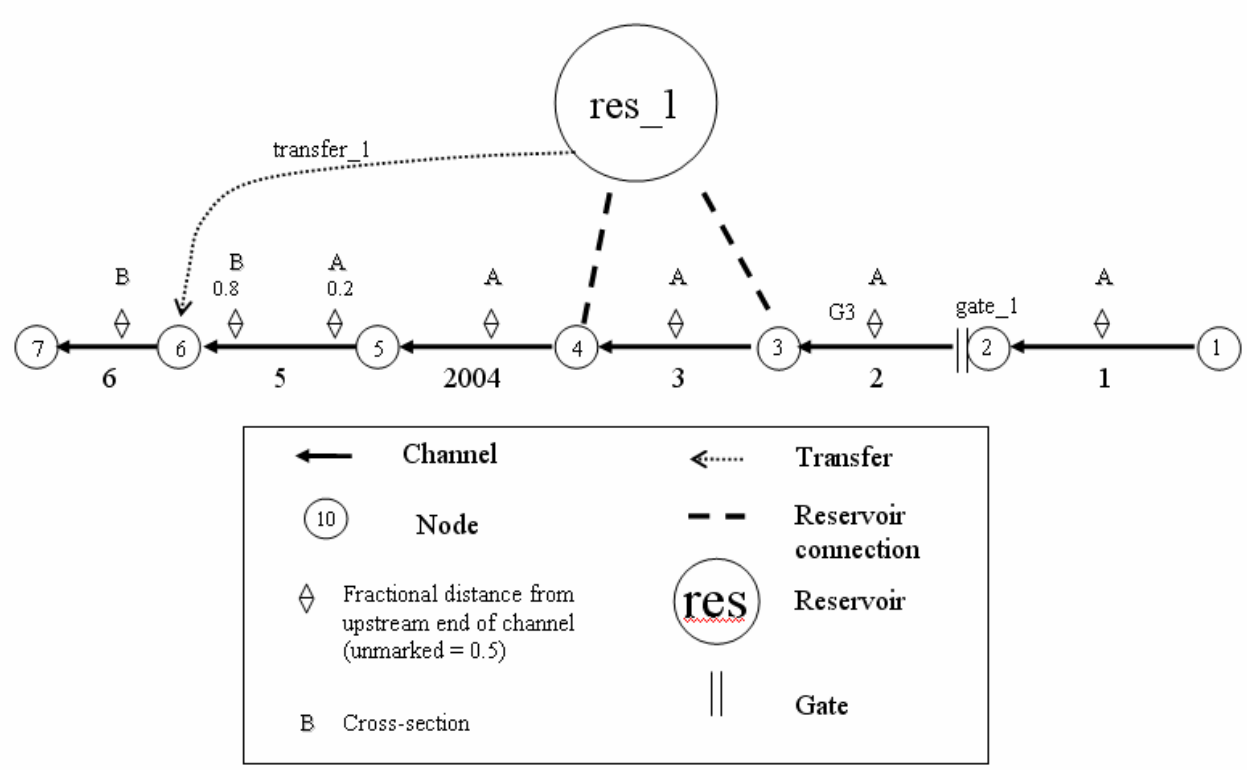


Figure 2 - Simple channel with a new reservoir, gate, and transfer.

The following steps will instruct you on how to create these new features and add them to the simple channel system.

1. Create the reservoir:

a. In the *Simulations Navigator*:
   1) Expand the *model: tutorial_hydro* folder.
   2) Expand the *Grid* folder.
   3) Double-click on *Reservoirs*.

b. In the *Layers panel*, right-click and select *Set edit layer*.

c. In the *Select Layers* window, double-click the *tutorial_grid* layer.

d. In the *Reservoirs table*:
   1) Right-click and select *Insert row*.

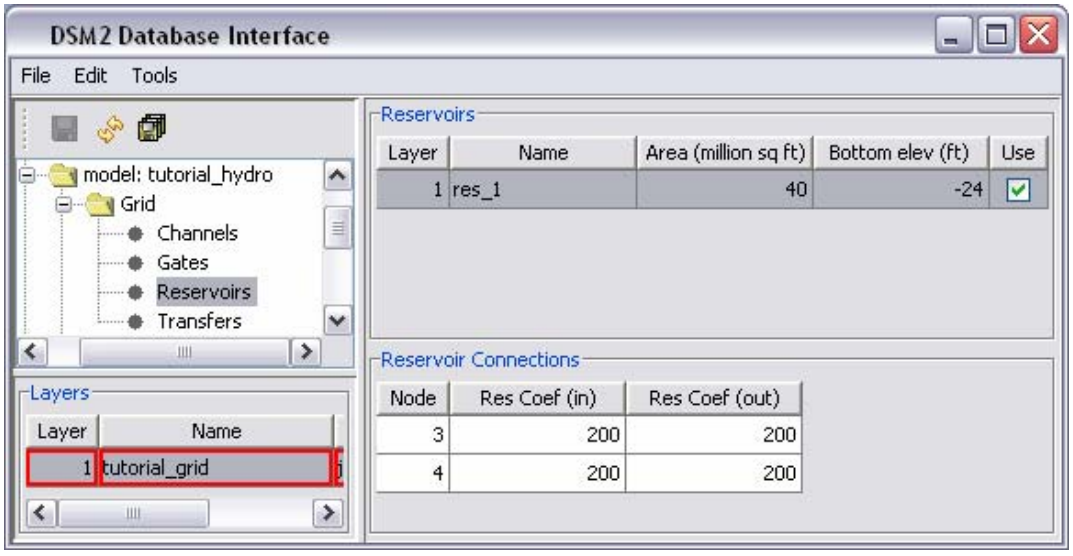
2) Enter the following values into the appropriate fields:
   i) Name: *res_1*
   ii) Area (million sq ft): *40*
   iii) Bottom elev (ft): *-24*
   iv) Use: Make sure that the entry contains a checkmark.

e. Note from Figure 2 that the reservoir has two connections; one at Node 3, and one at Node 4. Therefore, two rows of information will be needed for the *Reservoir Connections* table.

f. In the *Reservoir Connections table*:
   1) Right-click and select *Insert row*.
   2) Enter the following values into the appropriate fields:
      i) Node: *3*
      ii) Res Coef (in): *200*
      iii) Res Coef (out): *200*
   3) Again, right-click and select *Insert row*.
   4) Enter the following values into the appropriate fields:
      i) Node: *4*
      ii) Res Coef (in): *200*
      iii) Res Coef (out): *200*

g. Save the current settings.

h. At this point, the GUI should look as follows:



i. In the *Layers Panel*, right-click and select *Unset edit layer* [optional].

2. Create the Gate.

a. Note from Figure 2 that the gate is located at Node 2 of Channel 2. This gate consists of both a weir and a pipe. Therefore, two rows of information will be needed for the *Gate Devices* table.

b. In the *Simulations Navigator*:
   1) Remain in the *Grid* folder.
   2) Double-click on *Gates*.

c. In the *Layers panel*, right-click and select *Set edit layer*.

d. In the *Select Layers* window, double-click the *tutorial_grid* layer.

e. In the *Gates table*:
   1) Right-click and select *Insert row*.
   2) Enter the following values into the appropriate fields:
      i) Name: *gate_1*
      ii) Connected object: *Channel*
      iii) Name/No: *2*
      iv) to Node: *2*
      v) Use: Make sure that the entry contains a checkmark.

f. In the *Gate Devices table*:
   1) Right-click and select *Insert row*.
   2) Enter the following values into the appropriate fields:
      i) Name: *weir*
      ii) Structure: *weir*
      iii) Gate Control: *gated_top*
      iv) # Dupl: *2*
      v) Radius/width: *20*
      vi) Elev: *2*
      vii) Height: *9,999*
      viii) CF from Node: *0.8*

ix) CF to Node: *0.8*

x) Default Op: *gate_open*

g. Again, in the *Gate Devices* table:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Name: *pipe*

ii) Structure: *pipe*

iii) Gate Control: *no_gate*

iv) # Dupl: *2*

v) Radius/width: *20*

vi) Elev: *2*

vii) Height: *9,999*

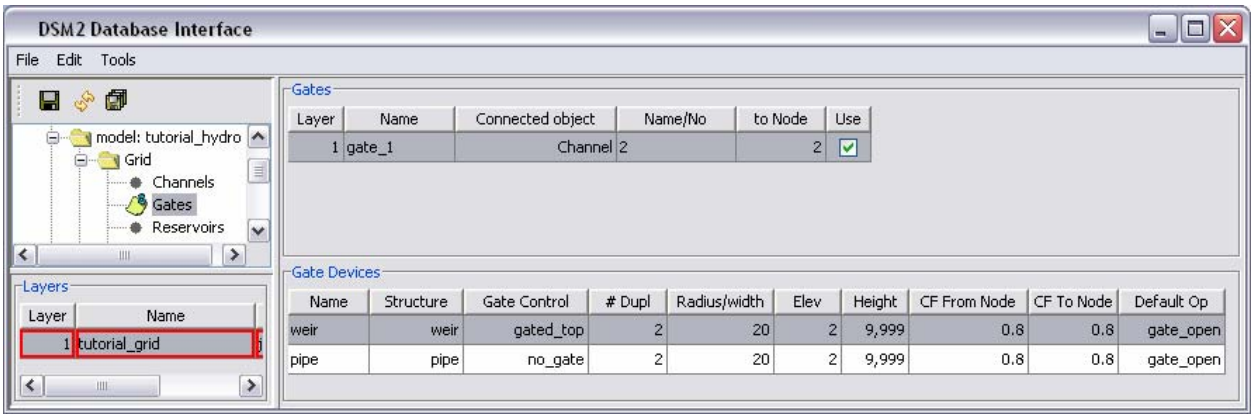
viii) CF from Node: *0.8*

ix) CF to Node: *0.8*

x) Default Op: *gate_open*

h. Save the current settings.

i. At this point, the GUI should look as follows:



j. In the *Layers Panel*, right-click and select *Unset edit layer* [optional].

2. Create the Transfer:

a. In the *Simulations Navigator*:

1) Remain in the *Grid* folder.

2) Double-click on *Transfers*.

b. In the *Layers panel*, right-click and select *Set edit layer*.

c. In the *Select Layers* window, double-click the *tutorial_grid* layer.

d. In the *Transfers table*:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Name: *transfer_1*

ii) From Object: *Reservoir*

iii) Name/Number: *res_1*

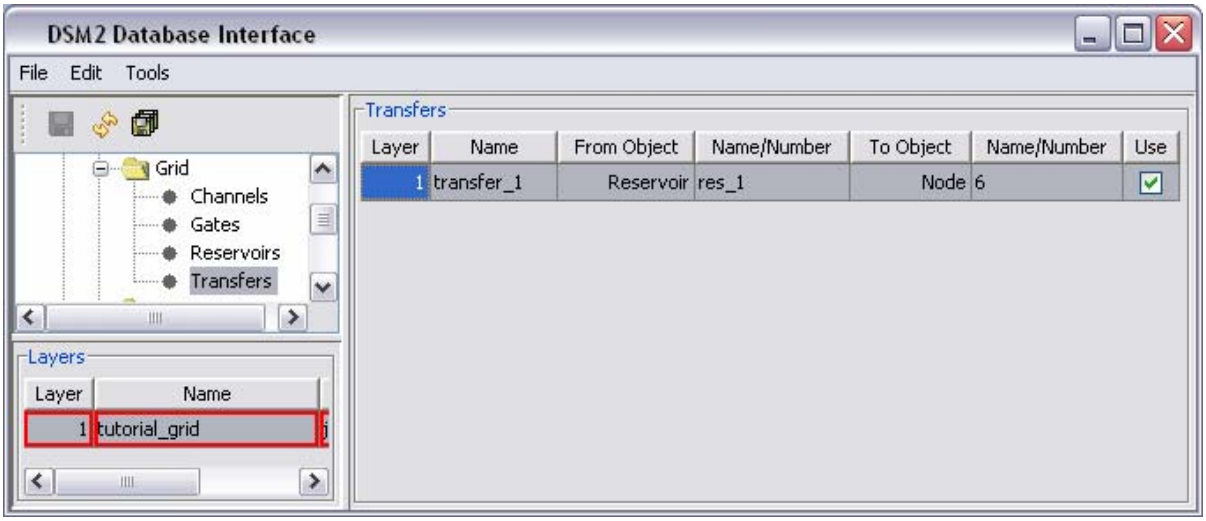
iv) To Object: *Node*

v) Name/Number: *6*

vi) Use: Make sure that the entry contains a checkmark.

e. Save the current settings.

f. At this point, the GUI should look as follows:



g. In the *Layers Panel*, right-click and select *Unset edit layer* [optional].

3. Add Initial Conditions for the Reservoir:

a. In the *Simulations Navigator*:

1) Collapse the *Grid* folder [optional].

2) Expand the *Initial Conditions* folder.

3) Double-click on *Reservoir IC*.

b. In the *Layers panel*, right-click and select *Set edit layer*.

c. In the *Select Layers* window, double-click the *tutorial_grid* layer.

d. In the *Reservoir Initial Conditions table*:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Reservoir Name: *res_1*

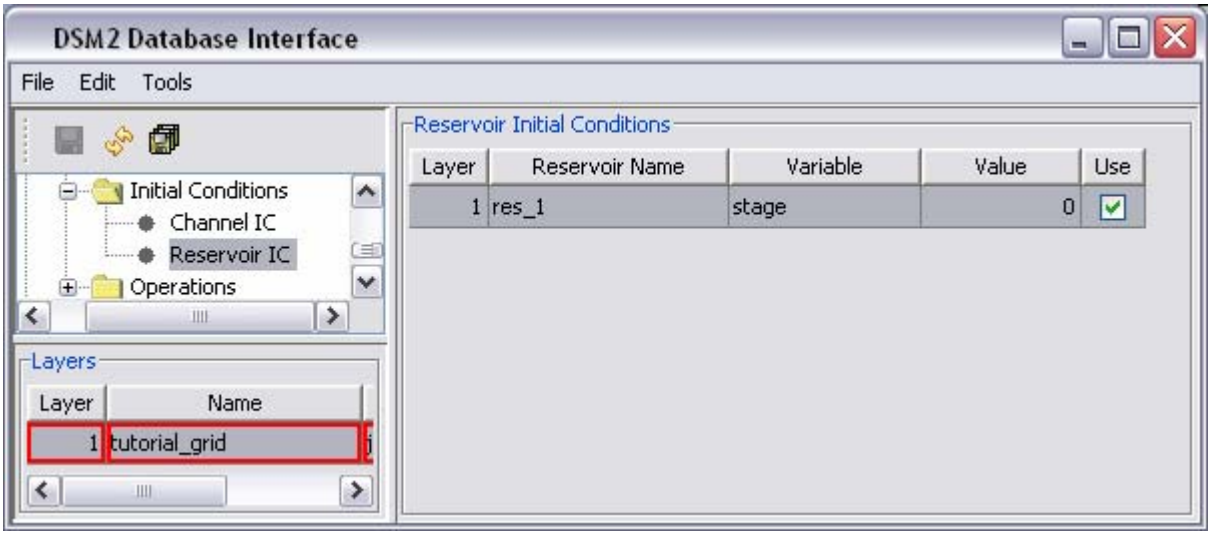
ii) Variable: *stage*

iii) Value: *0*

iv) Use: Make sure that the entry contains a checkmark.

e. Save the current settings.

f. At this point, the GUI should look as follows:



g. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

4. Add the Transfer Flow Time Series:

a. In the *Simulations Navigator*:

1) Collapse the *Initial Conditins* folder [optional].

2) Expand the *Input Time Series* folder.

3) Double-click on *Transfer Flows*.

b. In the *Layers panel*, right-click and select *Set edit layer*.

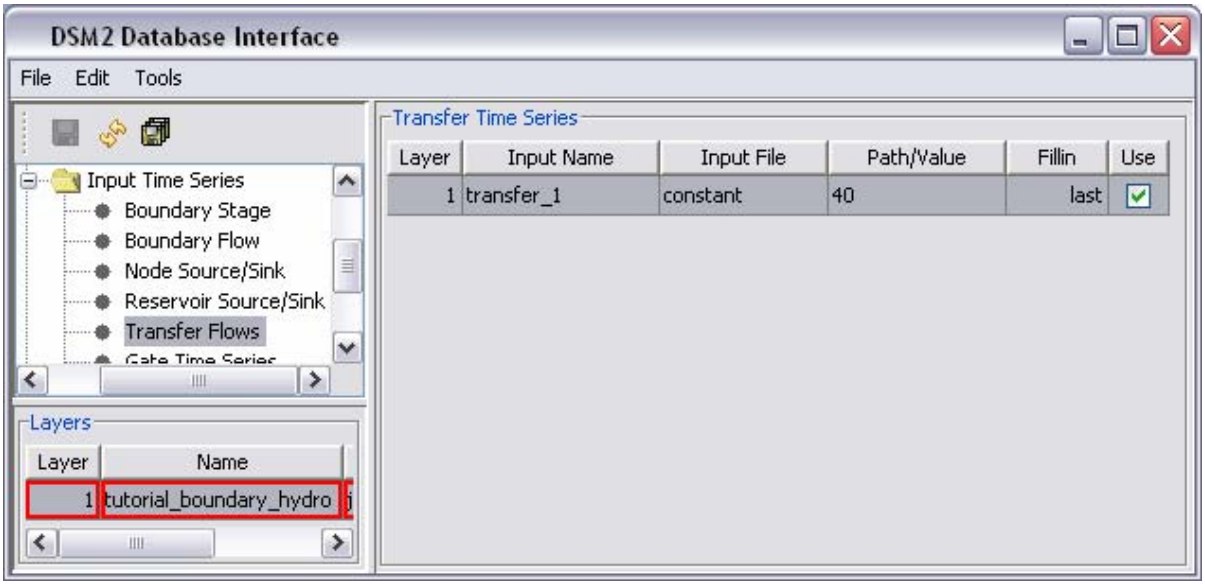
c. In the *Select Layers* window, double-click the *tutorial_boundary_hydro* layer.

d. In the *Transfer Time Series* table:

1) Right-click and select *Insert row*.
2) Enter the following values into the appropriate fields:
   i) Input Name: *transfer_1*
   ii) Input File: *constant*
   iii) Path/Value: *40*
   iv) Fillin: *last*
   v) Use: Make sure that the entry contains a checkmark.

e. Save the current settings.

f. At this point, the GUI should look as follows:



g. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

5. Running HYDRO and QUAL

a. In Windows Explorer, navigate to the directory: *\{DSM2_home}\tutorial\simulations\simple*.

b. Right-click on the directory, *t2_reservoir_gate_transfer*, and select *Open Command Window Here*.

c. In the command window, type: *hydro hydro.inp*.

d. In the command window, type: *qual qual.inp*.

e. Open the *output.dss* file in the *t2_reservoir_gate_transfer* directory, and examine the results.

IV. Tutorial 3: Layering

The purpose of this tutorial is to demonstrate the use and benefits of layering. Instructions are given for modifying existing model information in the database by adding new data layers. Layers are key to the DSM2 management system. They allow input items to be grouped in logical bundles, and allow changes to be brought into an old simulation without erasing or altering archived items. The following steps will instruct you on how to create and use layers.

1. Creating and Disabling a new reservoir:

In this section, a new reservoir will be created by adding another layer. This new layer will then be deep copied to a second new layer. The second new layer will then be edited so that its *Use* column is no longer checked. This process renders the new reservoir invisible to the model, demonstrating that the new reservoir does not have to be erased from the database, it can simply be masked.

a. In the *Simulations Navigator*:
    1) Expand the *model: tutorial_hydro* folder.
    2) Expand the *Grid* folder.
    3) Double-click on *Reservoirs*.
b. Create a new Reservoir Layer:
    1) In the *Layers panel*, right-click and select *New layer*.
    2) Select *Yes* to confirm the refresh.
    3) Name the new layer, *tutorial_grid_add_reservoir*, and add a description.
    4) Enter *2* for the layer number.
c. In the *Layers panel*, right-click and select *Set edit layer*.
d. In the *Select Layers* window, double-click the *tutorial_grid_add_reservoir* layer.
e. In the *Reservoirs* table:
    1) Right-click and select *Insert row*.
    2) Enter the following values into the appropriate fields:
        i) Name: *dummy_res*
        ii) Area (million sq ft): *60*
        iii) Bottom elev (ft): -30

iv) Use: Make sure that the entry contains a checkmark.

f. In the *Reservoir Connections* table:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Node: *5*

ii) Res Coef (in): *220*

iii) Res Coef (out): *220*

g. Again, in the *Reservoir Connections* table:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

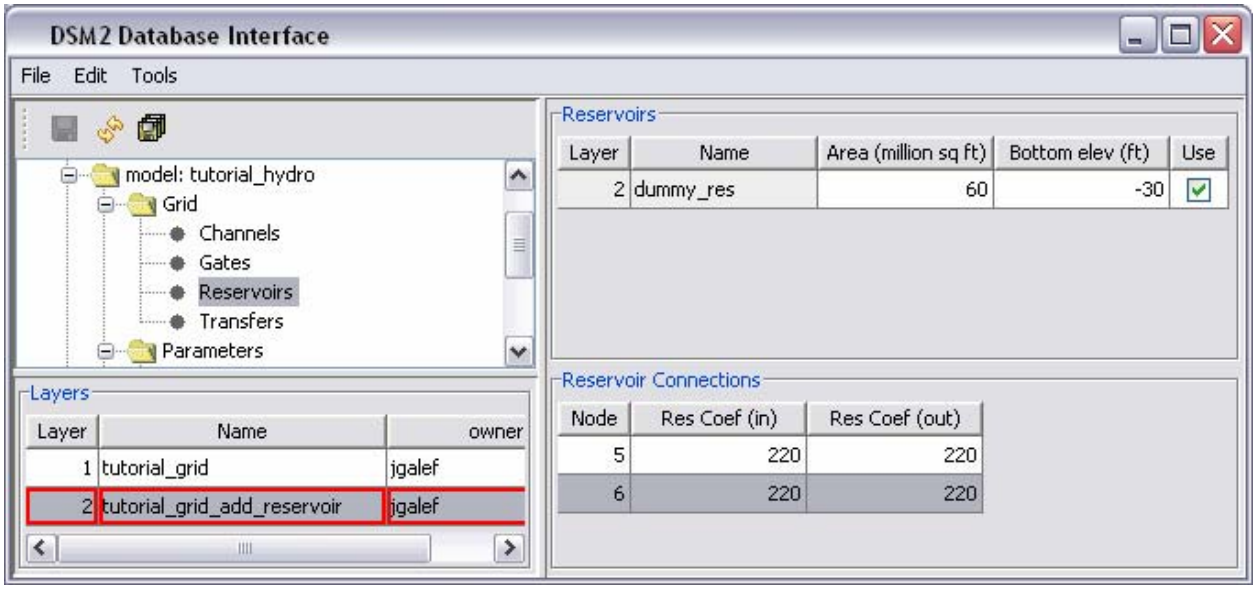
i) Node: *6*

ii) Res Coef (in): *220*

iii) Res Coef (out): *220*

h. Save the current settings.

i. At this point, the GUI should look as follows:



j. Create a Reservoir Revision Layer:

1) In the *Layers panel*, right-click and select *New layer*.

2) Select *Yes* to confirm the refresh.

3) Name the new layer, *tutorial_grid_revision*, and add a description.

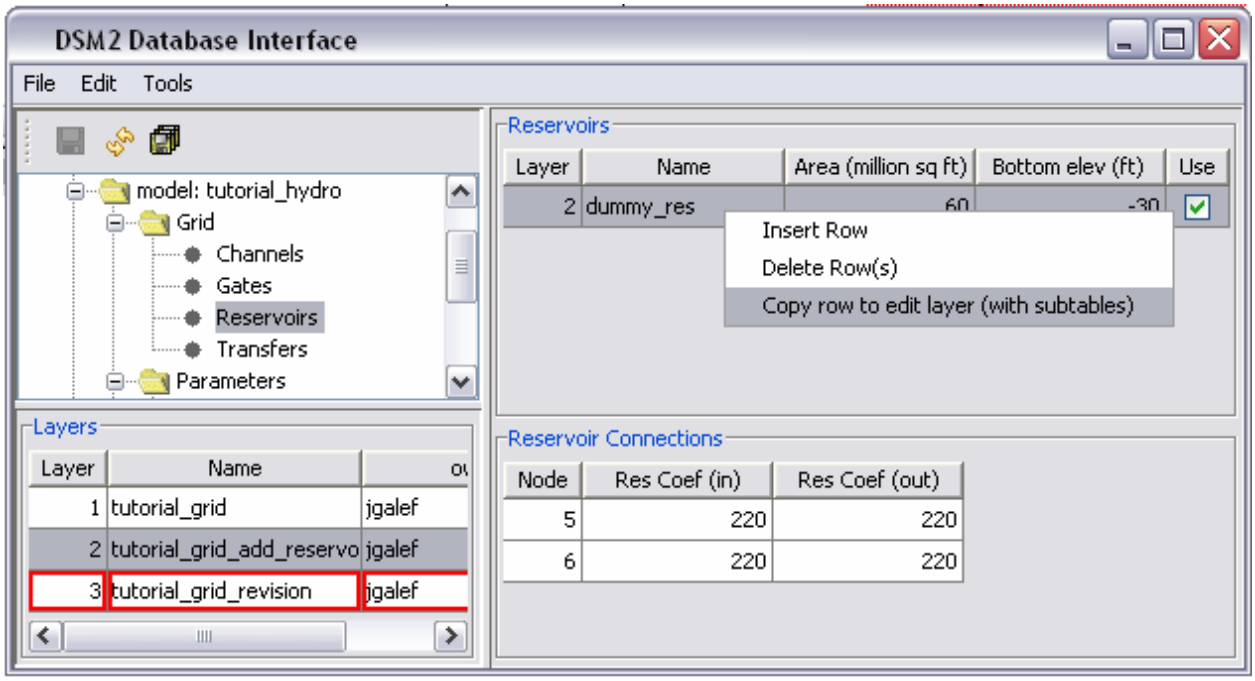
4) Enter *3* for the layer number.

k. In the *Layers panel*, right-click and select *Set edit layer*.

l. In the *Select Layers* window, double-click the *tutorial_grid_revision* layer.

m. In the *Layers panel*, click on the *tutorial_grid_add_reservoir* layer.

n. In the *Reservoirs* table, right-click the layer and select *Copy row to edit layer with subtables*.



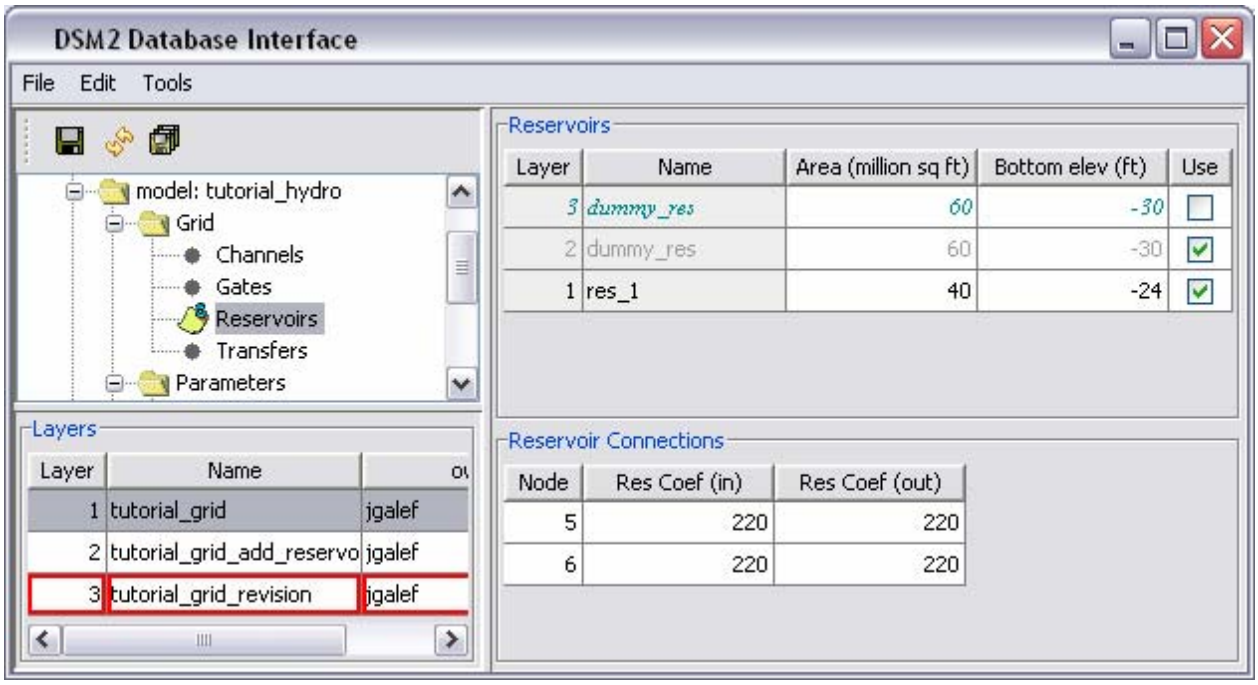
o. In the *Layers panel*, click on the *tutorial_grid_revison* layer.

p. In the *Reservoirs* table, double-click the *Use* field to get rid of the checkmark.

q. Save the current settings.

r. In the *Layers panel*, highlight all three layers.

s. At this point, the GUI should look as follows:

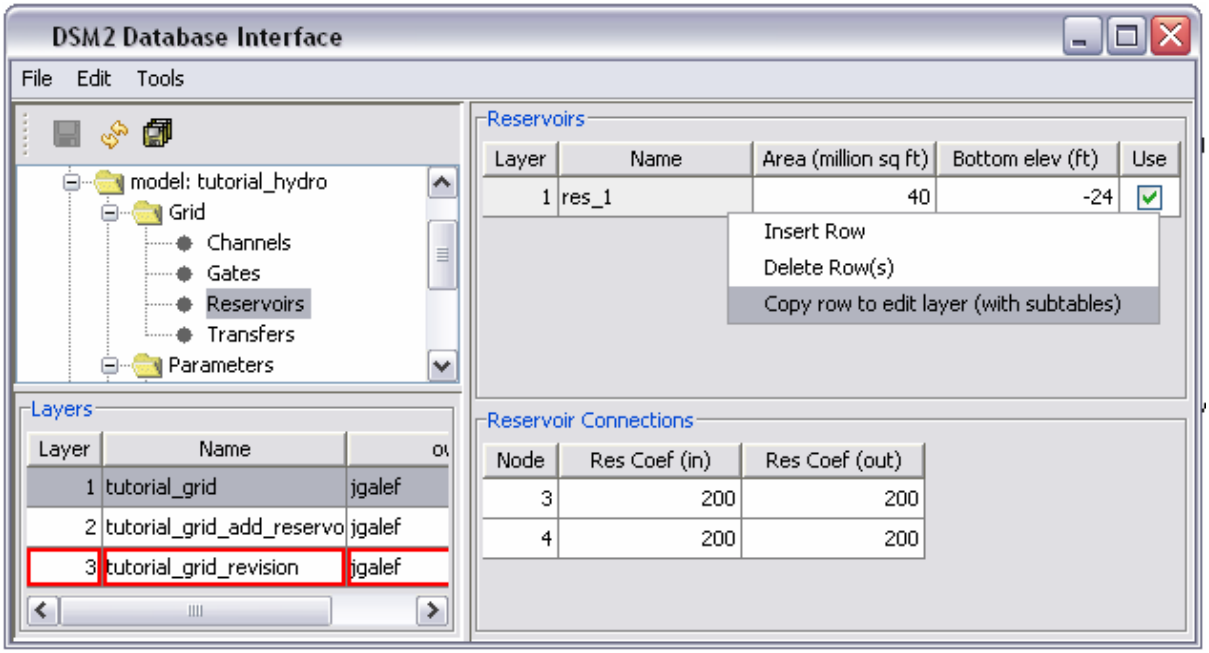


Note that *dummy_res* of Layer 2 has been grayed-out. This indicates that the new reservoir will no longer be used by the model.

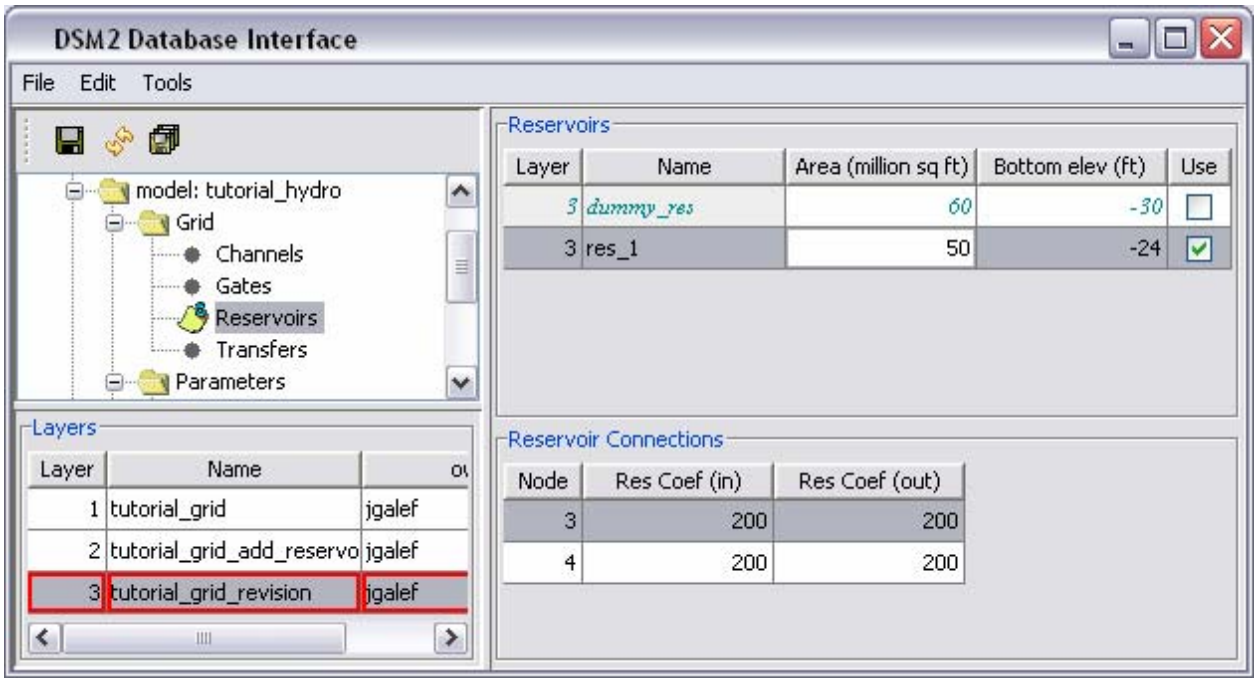
2. Altering the Properties of the Original Reservoir:

a. In the *Layers panel*, click on the *tutorial_grid* layer.

b. In the *Reservoirs table*, right-click the layer and select *Copy row to edit layer (with subtables)*.



c. In the *Reservoirs* table, for the new row in Layer 3, change the *Area (million sq ft)* field from *40* to *50*.



d. Save the current settings.

e. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

3. Changing the name of Channel 2004:

a. In the *Simulations Navigator*:

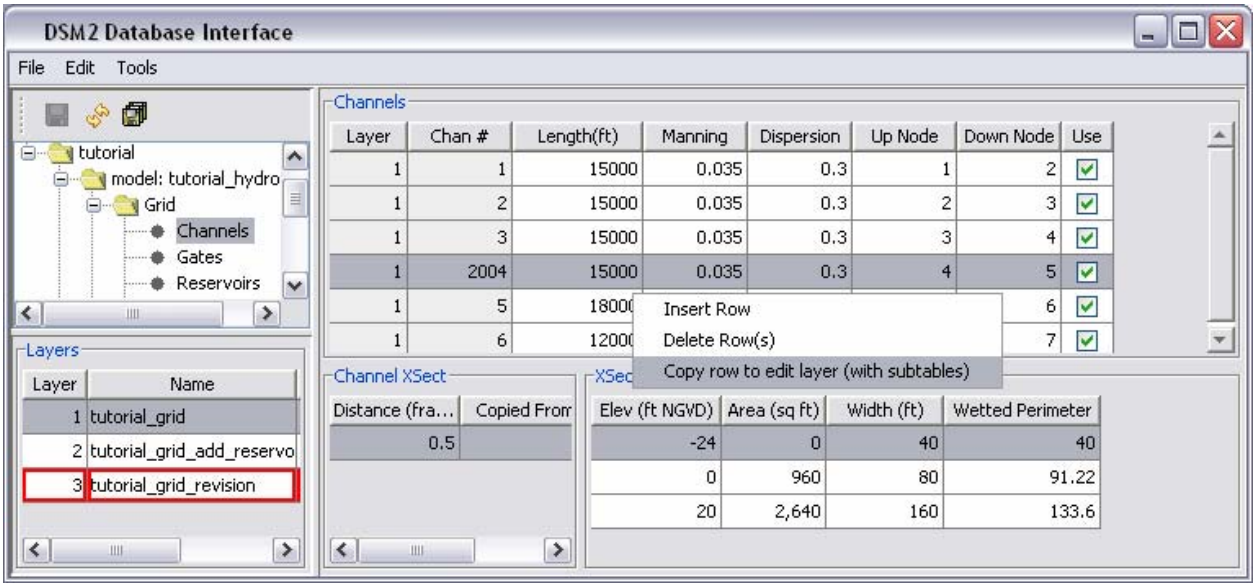
1) Remain in the *Grid* folder.
2) Double-click on *Channels*.

b. In the *Layers panel*, right-click and select *Set edit layer*.

c. In the *Select Layers* window, double-click the *tutorial_grid_revision* layer.

d. In the *Layers panel*, click on the *tutorial_grid* layer.

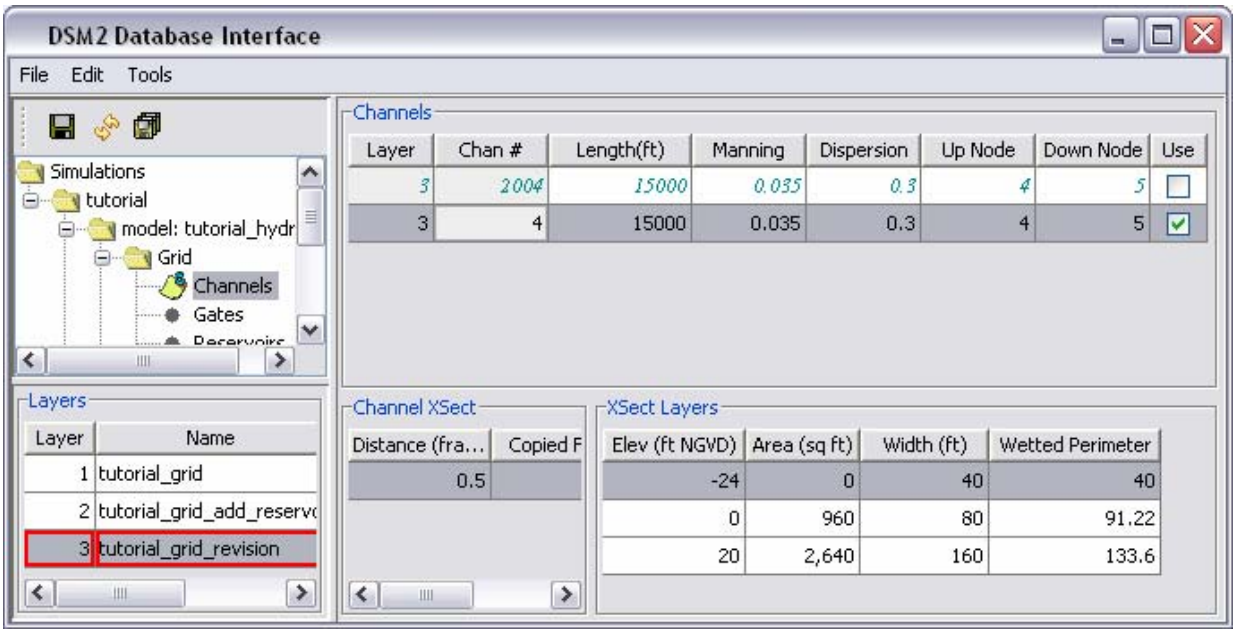
e. In the *Channels* table, right-click the layer with Channel 2004 and select *Copy row to edit layer (with subtables)*. Repeat this procedure.



f. In the *Layers panel*, click on the *tutorial_grid_revison* layer.

g. In the *Channels* table:

1) For the first row, double-click the *Use* field to get rid of the checkmark.
2) For the second row, change the *Chan #* field from *2004* to *4*.



h. Save the current settings.

i. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

4. Add the New *Channel 4* to the Output Layer:

a. In the *Simulations Navigator*:

1) Collapse the *Grid* folder [optional].

2) Expand the *Output Time Series* folder.

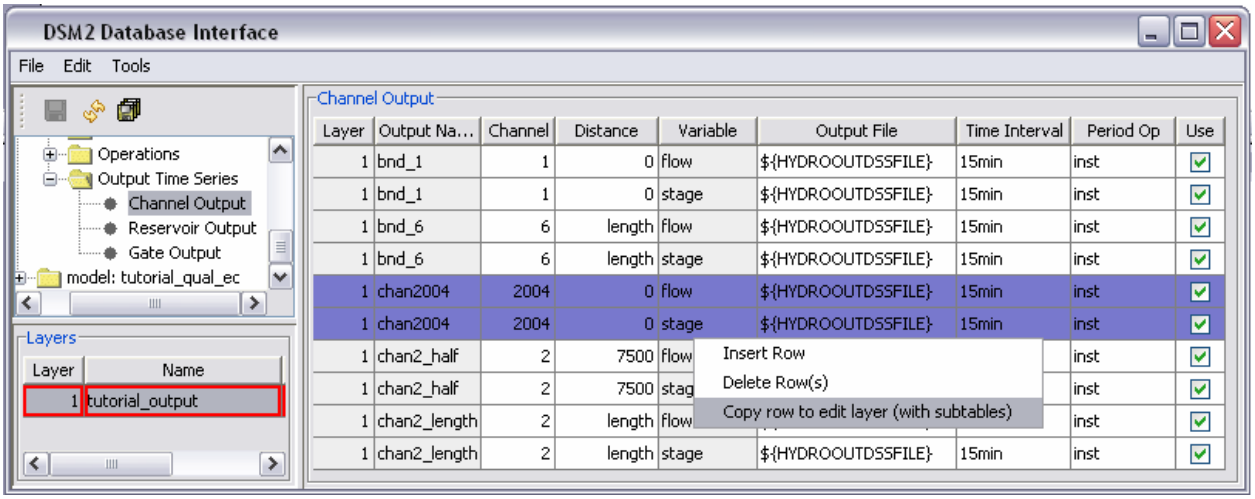
3) Double-click on *Channel Output*.

b. In the *Layers panel*, right-click and select *Set edit layer*.

c. In the *Select Layers* window, double-click the *tutorial_grid*.

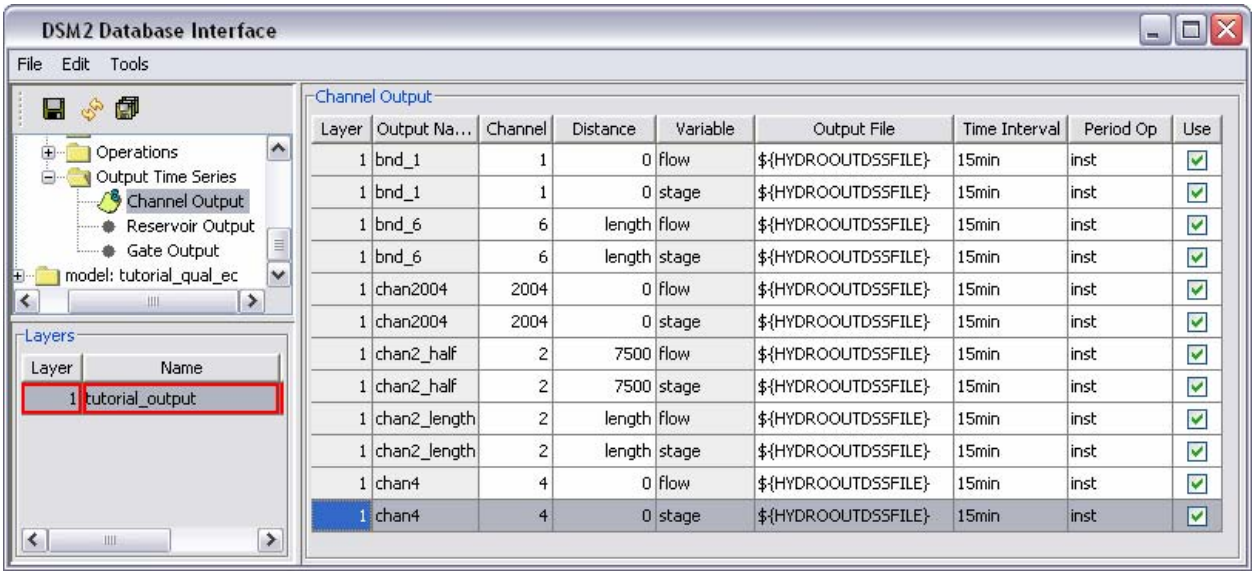
d. In the *Channel Output* table, hold down the *shift* key while clicking on the two rows associated with *Channel 2004*. Holding down *shift* allows for the selection of the entire row.

e. Right-click and select *Copy row to edit layer (with subtables)*.



f. Select *OK* to accept the Error Condition.

g. For the two new layers, change the Output Name field to *chan4*, and the *Channel field* to *4*.



The model will function properly despite the fact that the layers with *chan2004* are still present. The model will simply ignore these layers.

h. Save the current settings.

i. In the *Layers panel*, right-click and select *Unset edit layer*.

5. Add Initial Conditions for the New *Channel 4*:

a. In the *Simulations Navigator*:

1) Collapse the *Output Time Series* folder [optional].

2) Expand the *Initial Conditions* folder.

3) Double-click on *Channel IC*.

b. In the *Layers panel*, right-click and select *Set edit layer*.

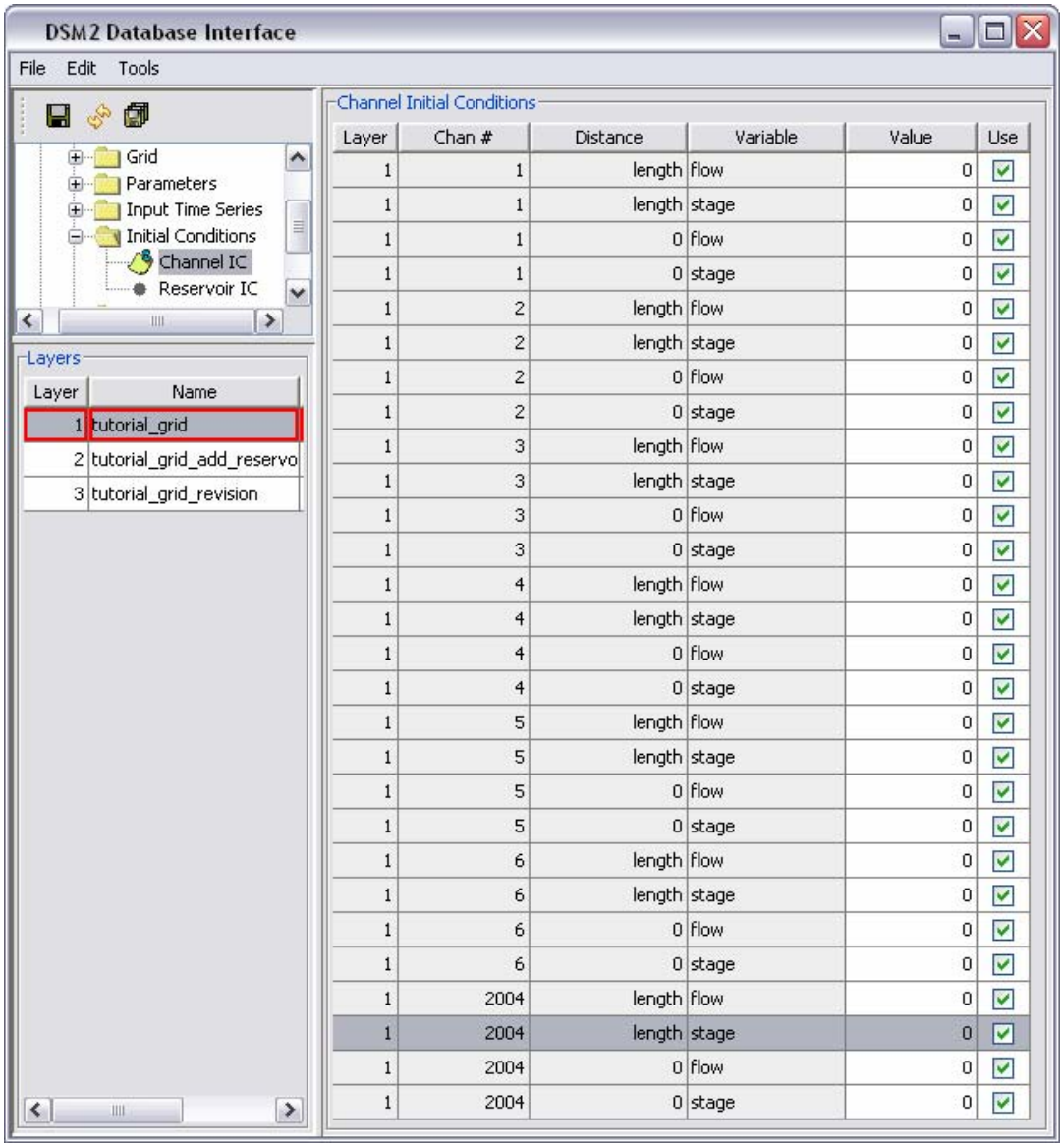
c. In the *Select Layers* window, double-click the *tutorial_grid*.

d. In the *Channel Initial Conditions* table, hold down the *shift* key while clicking on the four rows associated with *Channel 2004*.

e. Right-click and select *Copy row to edit layer (with subtables)*.

f. Select *OK* to accept the Error Condition.

g. For the four new layers, change the *Chan #* field from *2004* to *4*.



h. Save the current settings.

i. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

6. Running HYDRO and QUAL

a. In Windows Explorer, navigate to the directory: *{DSM2_home}**tutorial**simulations**simple*\.

b. Right-click on the directory, *t3_layering*, and select *Open Command Window Here*.

c. In the command window, type: *hydro hydro.inp*.

d. In the command window, type: *qual qual.inp*.

e. Open the *output.dss* file in the *t3_layering* directory, and examine the results.

V. Tutorial 4: Timevar

The purpose of this tutorial is to incorporate time-varying information to the model. In the previous sections, all boundary conditions and gate timings were set as constant, and no input files were needed. In this section, the model is set to read time-varying information stored in DSS files. The following steps will instruct you on how to add the time-varying information.

1. Change the Transfer Flows:

a. In the *Simulations Navigator*:
   1) Remain in the *model: tutorial_hydro* folder.
   2) Expand the *Input Time Series* folder.
   3) Double-click on *Transfer Flows.*

b. Add a new Transfer Flow Layer:
   1) In the *Layers panel*, right-click and select *New layer*.
   2) Select *Yes* to confirm the refresh.
   3) Name the new layer, *tutorial_hydro_ts*, and add a description.
   4) Enter *2* for the layer number.

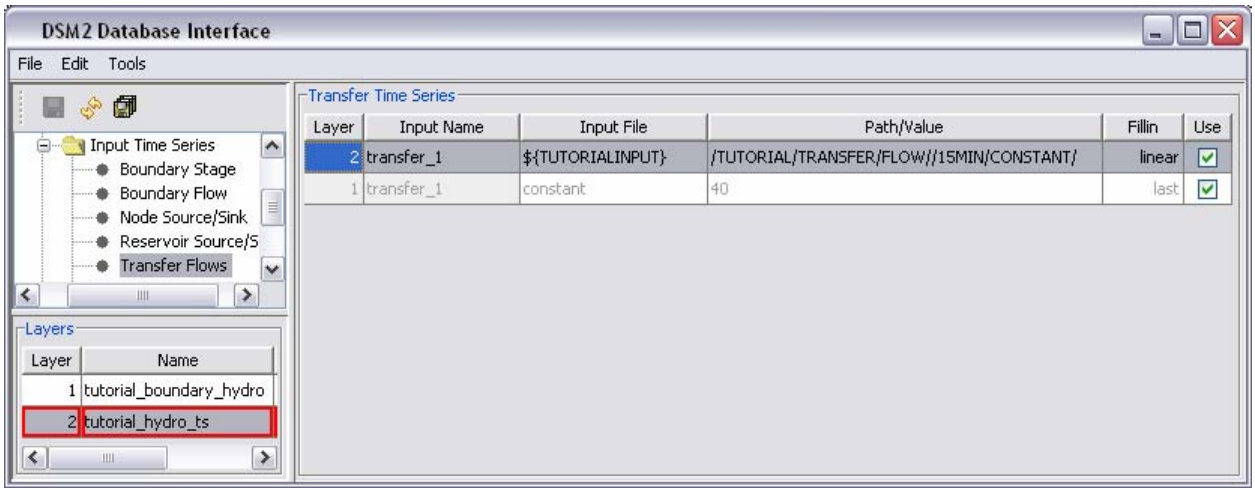
c. In the *Layers panel*, right-click and select *Set edit layer*.

d. In the *Select Layers* window, double-click the *tutorial_hydro_ts* layer.

e. In the *Transfer Time Series table*:
   1) Right-click and select *Insert row*.
   2) Enter the following values into the appropriate fields:
      i) Input Name: *transfer_1*
      ii) Input File: *${TUTORIALINPUT}*
      iii) Path/Value: */TUTORIAL/TRANSFER/FLOW//15MIN/CONSTANT/*
      iv) Fillin: *linear*
      v) Use: Make sure that the entry contains a checkmark.

f. Save the current settings.

g. At this point, the GUI should look as follows:



h. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

2. Running HYDRO and QUAL

a. In Windows Explorer, navigate to the directory: *{DSM2_home}**tutorial**simulations**simple*\.

b. Right-click on the directory, *t4_timevar*, and select *Open Command Window Here*.

c. In the command window, type: *hydro hydro.inp*.

d. In the command window, type: *qual qual.inp*.

e. Open the *output.dss* file in the *t4_timevar* directory, and verify that the results are identical to the results from the previous tutorial (located in the *t3_layering* directory).

3. Adjust the Text Input Files:

a. In Windows Explorer, navigate to the directory: *{DSM2_home}**tutorial**simulations**simple**t4_timevar*

b. Open *hydro.inp* for editing.

c. In the *ENVVARS* section, change the *DSM2MODIFIER* environment variable from *timevar_1* to *timevar_2*.

d. Open *qual.inp* for editing.

e. In the *ENVVARS* section, change the *DSM2MODIFIER* environment variable from *timevar_1* to *timevar_2*.

4. Add Source information into HYDRO:

a. In the *Simulations Navigator*:

1) Remain in the *Input Time Series* folder.

2) Double-click on *Node Source/Sink.*

b. In the *Layers panel*, right-click and select *Set edit layer*.

c. In the *Select Layers* window, double-click the *tutorial_hydro_ts* layer.

d. In the *Node Source Time Series Input table*:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Input Name: *source1*

ii) Node: *5*

iii) Input File: *${TUTORIALINPUT}*

iv) Path/Value: */TUTORIAL/SOURCE/FLOW//15MIN/CONSTANT/*

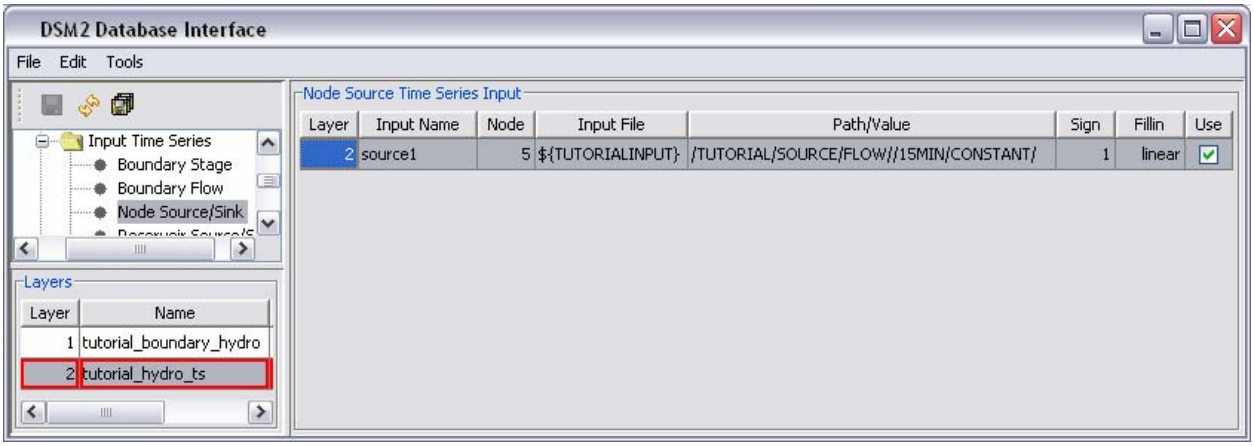
v) Sign: *1*

vi) Fillin: *linear*

vii) Use: Make sure that the entry contains a checkmark.

e. Save the current settings.

f. At this point, the GUI should look as follows:



g. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

5. Add Source information into QUAL:

a. In the *Simulations Navigator*:
   1) Collapse the *model: tutorial_hydro* folder [optional].
   2) Expand the *model:tutorial_qual_ec* folder.
   3) Expand the *Input Time Series* folder.
   4) Double-click on *Node Source/Sink.*

b. Create a new QUAL Source Layer:
   1) In the *Layers panel*, right-click and select *New layer*.
   2) Select *Yes* to confirm the refresh.
   3) Name the new layer, *tutorial_qual_ts*, and add a description.
   4) Enter *2* for the layer number.

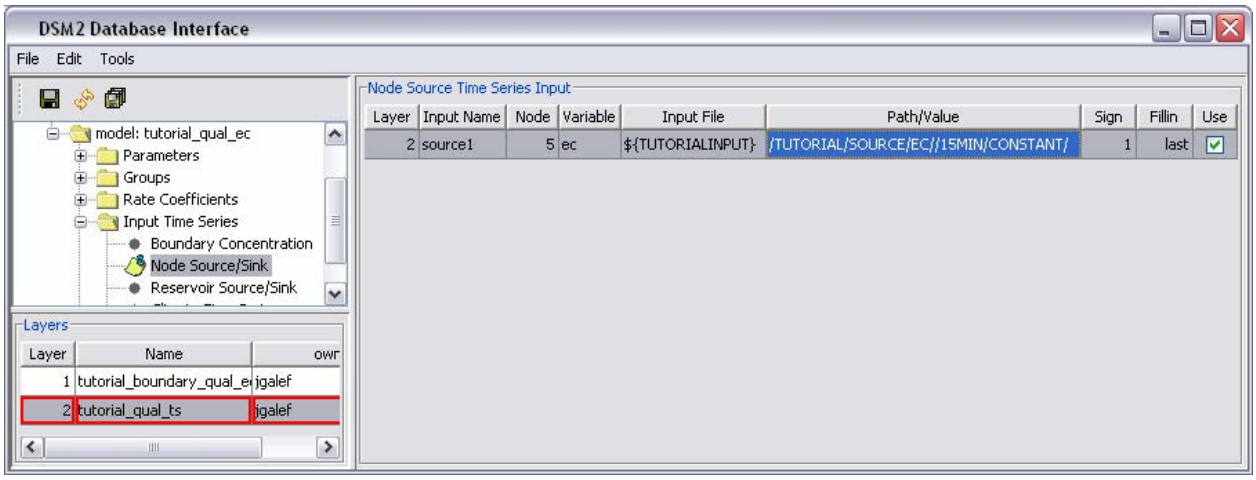
c. In the *Layers panel*, right-click and select *Set edit layer*.

d. In the *Select Layers* window, double-click the *tutorial_qual_ts* layer.

e. In the *Node Source Time Series table*:
   1) Right-click and select *Insert row*.
   2) Enter the following values into the appropriate fields:
      i) Input Name: *source1*
      ii) Node: *5*
      iii) Variable: *ec*
      iv) Input File: *${TUTORIALINPUT}*
      v) Path/Value: */TUTORIAL/SOURCE/EC//15MIN/CONSTANT/*
      vi) Sign: *1*
      vii) Fillin: *last*
      viii) Use: Make sure that the entry contains a checkmark.

f. Save the current settings.

g. At this point, the GUI should look as follows:



h. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

6. Add Tide Information for Downstream Boundary in HYDRO:

a. In the *Simulations Navigator*:
   1) Collapse the *model: tutorial_qual_ec* folder [optional].
   2) Expand the *model: tutorial_hydro* folder.
   3) Expand the *Input Time Series* folder.
   4) Double-click on *Boundary Stage.*

b. Create a new HYDRO Time Series Boundary:
   1) In the *Layers panel*, right-click and select *New layer*.
   2) Select *Yes* to confirm the refresh.
   3) Name the new layer, *tutorial_ hydro_tide*, and add a description.
   4) Enter *3* for the layer number.

c. In the *Layers panel*, right-click and select *Set edit layer*.

d. In the *Select Layers* window, double-click the *tutorial_hydro_tide* layer.

e. In the *Boundary Stage table*:
   1) Right-click and select *Insert row*.
   2) Enter the following values into the appropriate fields:
      i) Input Name: *downstream_stage*
      ii) Node: *7*

iii) Input File: *${TUTORIALINPUT}*

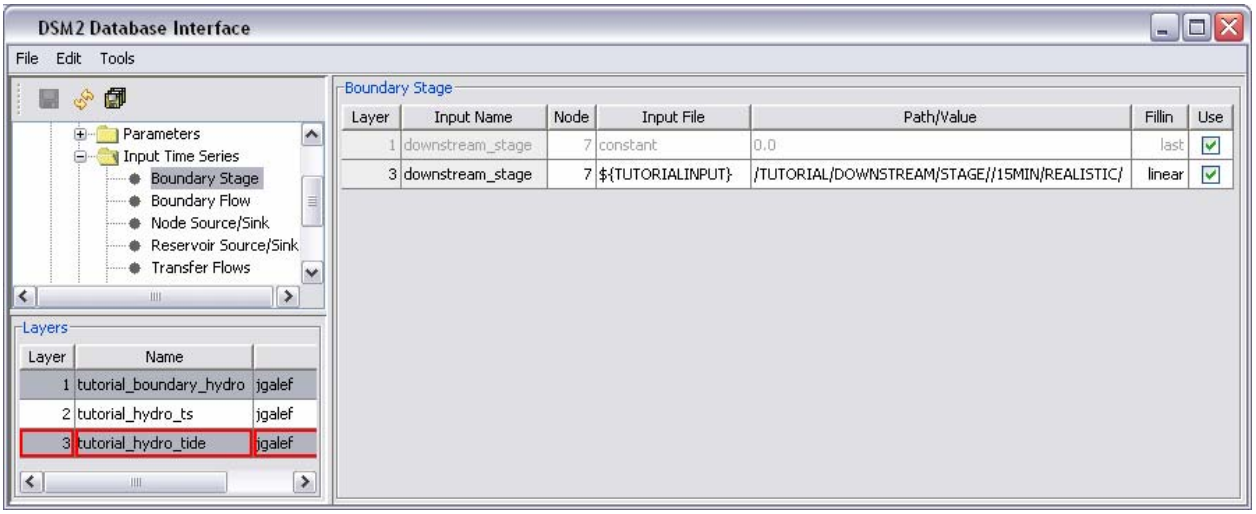
iv) Path/Value: */TUTORIAL/DOWNSTREAM/STAGE//15MIN/REALISTIC/*

v) Fillin: *linear*

vi) Use: Make sure that the entry contains a checkmark.

f. Save the current settings.

g. At this point, the GUI should look as follows:



h. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

7. Add Tide Information for Downstream Boundary in QUAL:

a. In the *Simulations Navigator*:

1) Collapse the *model: tutorial_hydro* folder [optional].

2) Expand the *model: tutorial_qual_ec* folder.

3) Expand the *Input Time Series* folder.

4) Double-click on *Boundary Concentration.*

b. Add a new QUAL Time Series Boundary:

1) In the *Layers panel*, right-click and select *New layer*.

2) Select *Yes* to confirm the refresh.

3) Name the new layer, *tutorial_ qual_tide*, and add a description.

4) Enter *3* for the layer number.

c. In the *Layers panel*, right-click and select *Set edit layer*.

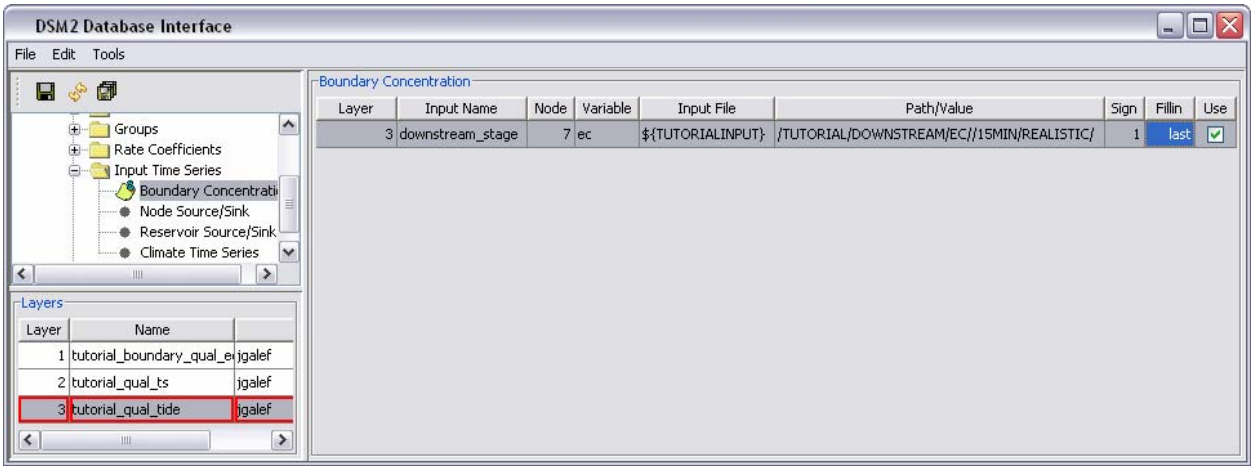
d. In the *Select Layers* window, double-click the *tutorial_qual_tide* layer.

e. In the *Boundary Concentration table*:

1) Right-click and select *Insert row*.
2) Enter the following values into the appropriate fields:
   i) Input Name: *downstream_stage*
   ii) Node: *7*
   iii) Input File: *${TUTORIALINPUT}*
   iv) Path/Value: */TUTORIAL/DOWNSTREAM/EC//15MIN/REALISTIC/*
   v) Sign: *1*
   vi) Fillin: *last*
   vii) Use: Make sure that the entry contains a checkmark.

f. Save the current settings.

g. At this point, the GUI should look as follows:



h. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

8. Add a Gate Time Series to HYDRO:

This gate time series will control the weir. The pipe is to be left open.

a. In the *Simulations Navigator*:
   1) Collapse the *model: tutorial_ qual_ec* folder [optional].
   2) Expand the *model: tutorial_ hydro* folder.
   3) Expand the *Input Time Series* folder.
   4) Double-click on *Gate Time Series.*

b. Add a Gate Time Series Layer:
   1) In the *Layers panel*, right-click and select *New layer*.

2) Select *Yes* to confirm the refresh.
3) Name the new layer, *tutorial_ gate_input*, and add a description.
4) Enter *4* for the layer number.

c. In the *Layers panel*, right-click and select *Set edit layer*.

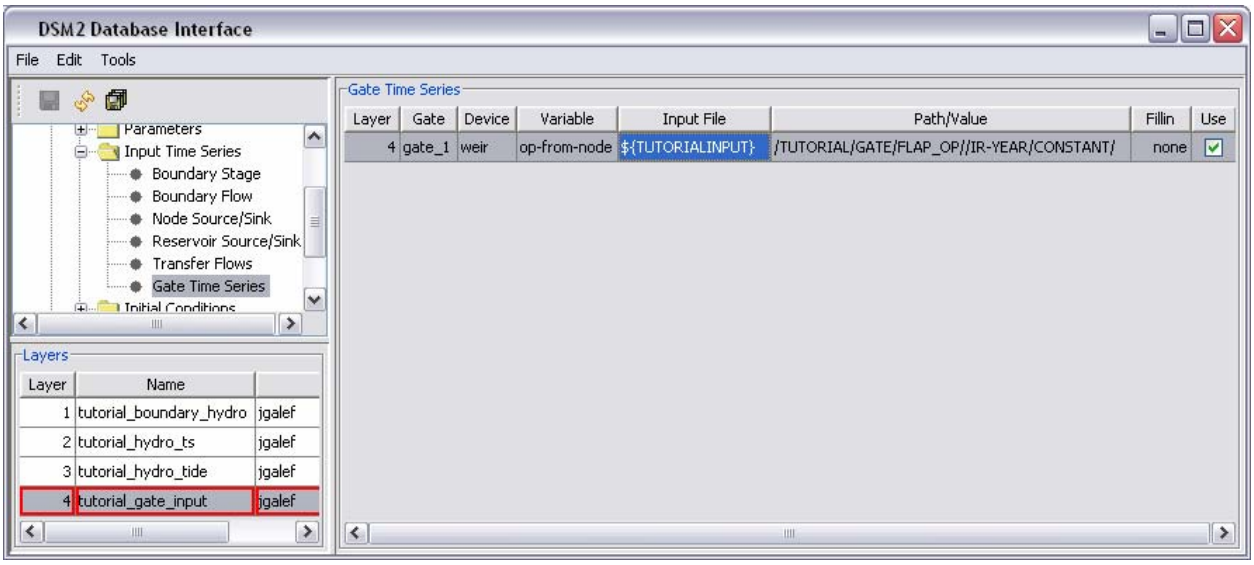
d. In the *Select Layers* window, double-click the *tutorial_gate_input* layer.

e. In the *Gate Time Series table:*

1) Right-click and select *Insert row*.
2) Enter the following values into the appropriate fields:
   i) Gate: *gate_1*
   ii) Device: *weir*
   iii) Variable: *op_from_node*
   iv) Input File: *${TUTORIALINPUT}*
   v) Path/Value: */TUTORIAL/GATE/FLAP_OP//IR-YEAR/CONSTANT/*
   vi) Fillin: *none*
   vii) Use: Make sure that the entry contains a checkmark.

f. Save the current settings.

g. At this point, the GUI should look as follows:



h. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

9. Running HYDRO and QUAL

a. In Windows Explorer, navigate to the directory: *{DSM2_home}**tutorial**simulations**simple*\.

b. Right-click on the directory, *t4_Timevar*, and select *Open Command Window Here*.

c. In the command window, type: *hydro hydro.inp*.

d. In the command window, type: *qual qual.inp*.

e. Open the *output.dss* file in the *t4_timevar* directory, and examine the results.

VI. Tutorial 5: Output

The purpose of this tutorial is to provide instruction on advanced output options. The first part involves modifications to the text input file, *hydro.inp*. The second part describes the use of *groups* in the GUI. With *groups*, the user can enter a small number of expressions to specify many output locations. The following steps will instruct you on how to add the *groups*.

1. Add Output Paths to *hydro.inp*:

a. In Windows Explorer, navigate to the directory, *\{DSM2_home}\tutorial\simulations\simple\t5_output*.

b. Open the file *addin.inp* and note the new output paths for the channels and reservoir. The information in this file is similar to that required for the text version of DSM2, but has an additional *Name* field plus the identification of the location being output.

c. Copy the entire file contents to the clipboard.

d. Open the file *hydro.inp*.

e. Navigate to the bottom of the file and paste the information.

2. Add *Boundary* and *Source Groups* to the Database:

a. Navigate back to the GUI.

b. In the *Simulations Navigator*:

1) Expand the *model: tutorial_qual_ec* folder.

2) Expand the *Groups* folder.

3) Double-click on *Groups.*

c. Add a Groups Layer:

1) In the *Layers panel*, right-click and select *New layer*.

2) Select *Yes* to confirm the refresh.

3) Name the new layer, *tutorial_ groups*, and add a description.

4) Enter *1* for the layer number.

d. In the *Layers panel*, right-click and select *Set edit layer*.

e. In the *Select Layers* window, double-click the *tutorial_groups* layer.

f. In the *Groups table*:

1) right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Name: *boundary*

ii) Description of Group: *boundary*

iii) Use: Make sure that the entry contains a checkmark.

g. In the *Group Members table*:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Member Type: *Boundary Stage*

ii) Identifier/Pattern: *.*stream.**

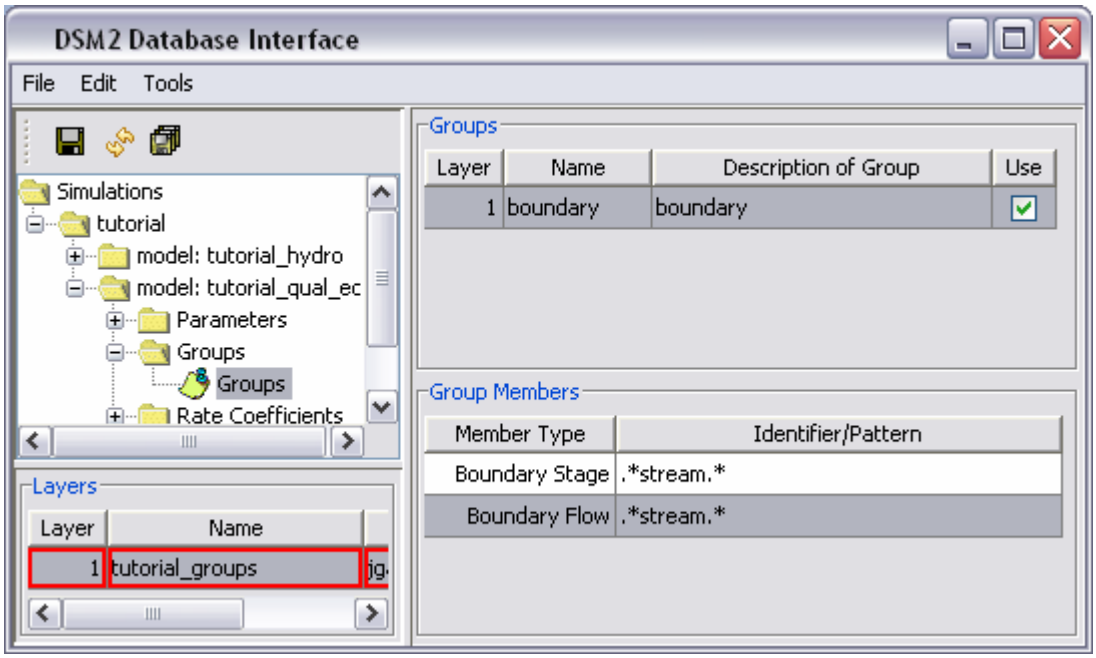
3) Again, right-click and select *Insert row*.

4) Enter the following values into the appropriate fields:

i) Member Type: *Boundary Flow*

ii) Identifier/Pattern: *.*stream.**

h. At this point, the GUI should look as follows:



i. In the *Groups table*:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Name: *sources*

ii) Description of Group: *inflow from mid-stream sources*

iii) Use: Make sure that the table contains a checkmark.

j. In the *Group Members table*:

1) Right-click and select *Insert row*.

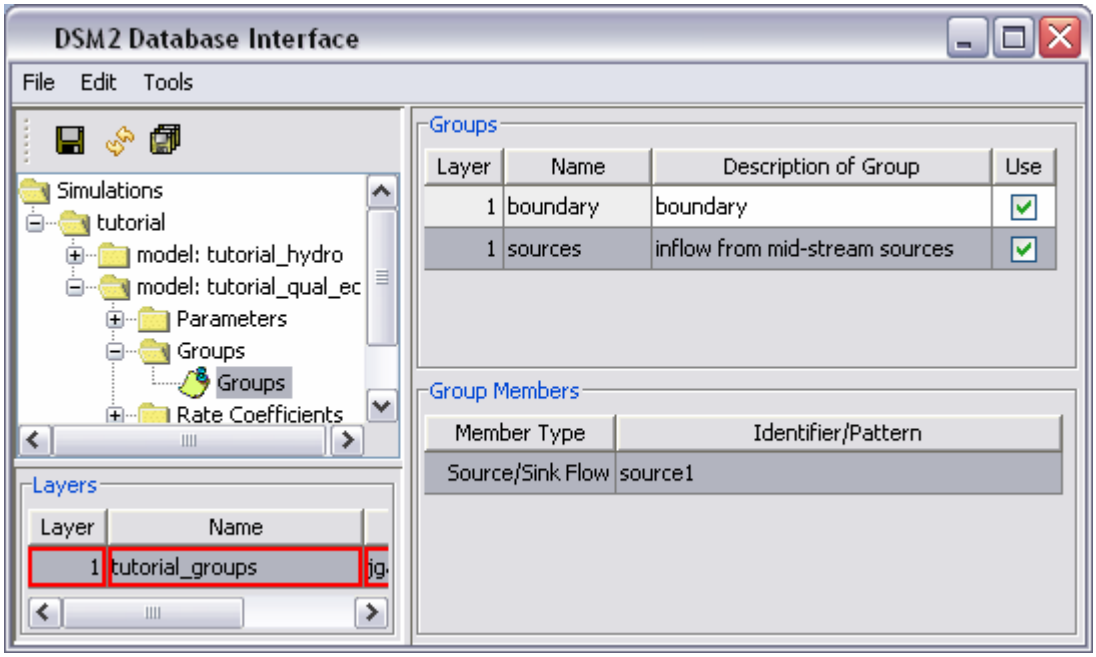
2) Enter the following values into the appropriate fields:

i) Member Type: *Source/Sink Flow*

ii) Identifier/Pattern: *source1*

k. Save the current settings.

l. At this point, the GUI should look as follows:



m. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

3. Add *Group Output* for *Channel 5*:

a. In the *Simulations Navigator*:

1) Collapse the *Groups* folder [optional].

2) Expand the *model: tutorial_qual_ec* folder.

3) Expand the *Output Time Series* folder.

4) Double-click on *Channel Output.*

b. Add a new Output Layer:

1) In the *Layers panel*, right-click and select *New layer*.
2) Select *Yes* to confirm the refresh.
3) Name the new layer, *tutorial_ source_output*, and add a description.
4) Enter *2* for the layer number.

c. In the *Layers panel*, right-click and select *Set edit layer*.
d. In the *Select Layers* window, double-click the *tutorial_source_output* layer.
e. In the *Channel Output* table:
   1) Right-click and select *Insert row* a total of three times. Or, if you feel comfortable, you can click on an established row, right-click and select *Copy to edit row (with subtables)*, and make the following corrections.
   2) For the first new row, enter the following values into the appropriate fields:
      i) Name: *ch5*
      ii) Channel: *5*
      iii) Distance: *5000*
      iv) Variable: *ec*
      v) Source Group: Leave this field blank.
      vi) Output File: *${QUALOUTDSSFILE}*
      vii) Time Interval: *15min*
      viii) Period Op: *inst*
      ix) Use: Make sure that the entry contains a checkmark.
   3) For the second new row, enter the following values into the appropriate fields:
      i) Name: *ch5_bnd*
      ii) Channel: *5*
      iii) Distance: *5000*
      iv) Variable: *ec*
      v) Source Group: *boundary*
      vi) Output File: *${QUALOUTDSSFILE}*
      vii) Time Interval: *15min*
      viii) Period Op: *inst*
      ix) Use: Make sure that the entry contains a checkmark.
   4) For the third new row, enter the following values into the appropriate fields:

i) Name: *ch5_src*

ii) Channel: *5*

iii) Distance: *5000*

iv) Variable: *ec*

v) Source Group: *source*

vi) Output File: *${QUALOUTDSSFILE}*

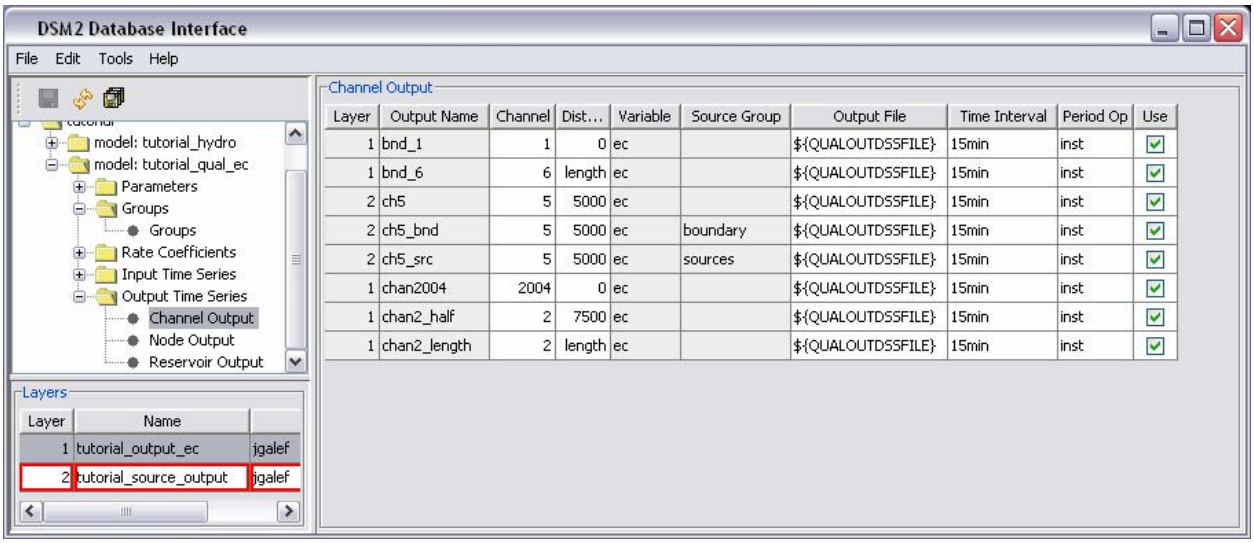
vii) Time Interval: *15min*

viii) Period Op: *inst*

ix) Use: Make sure that the entry contains a checkmark.

f. Save the current settings.

g. At this point, the GUI should look as follows:



h. In the *Layers panel*, right-click and select *Unset edit layer*.

4. Running HYDRO and QUAL

a. Open a command window for the *t5_output* directory.

b. In the command window, type: *hydro hydro.inp*.

c. In the command window, type: *qual qual.inp*.

d. Open the *output.dss* file in the *t5_output* directory, and examine the results.

VII. Tutorial 6: Oprule

The purpose of this tutorial is to provide instruction on the use of Operating Rule Language (ORL) statements to set gate operations and flows. With operating rules, expressions can be used to make the model operate gates on-the-fly; e.g., a gate can be directed to automatically close when stage conditions reach a certain threshold. The following steps will instruct you on how to add the ORL statements. We will apply operating rules to a new gate that we will create and to a source/sink inflow.

1. Adding a Second Gate Where Op Rule Will Be Applied

a. In the *Simulations Navigator*:
   1) Expand the *model: tutorial_hydro* folder.
   2) Expand the *Grid* folder.
   3) Double-click on *Gates.*

b. In the *Layer panel*, right-click and select *Set edit layer*.

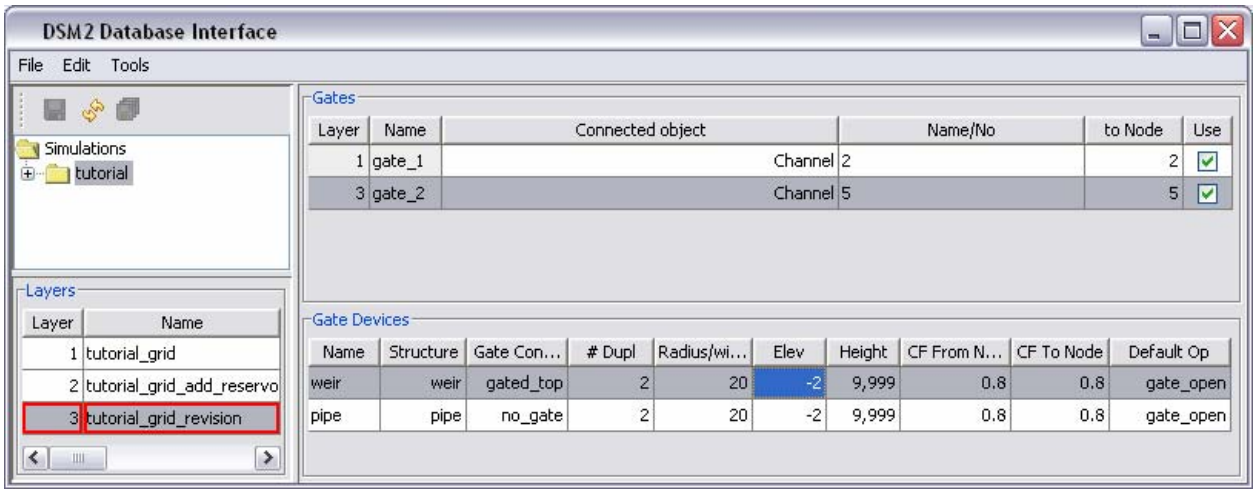
c. In the *Select Layers* window, double-click the *tutorial_grid_revision* layer.

d. In the *Gates* table:
   1) Right-click on the row with *gate_1* and select *Copy row to edit layer (with subtables)*.
   2) Click on the copied row and change the following fields:
      i) Name: *gate_2*
      ii) Name/No.: *5*
      iii) to Node: *5*

e. In the *Gate Devices table*:
   1) Click on each of the weir and pipe rows and change the following field:
      i) Elev: *-2*

f. Save the current settings.

g. At this point, the GUI should look as follows:



2. Adding Output for the Second Gate:

a. In the *Simulations Navigator*:
   1) Collapse the *Grid* folder [optional].
   2) Expand the *Output Time Series* folder.
   3) Double-click on *Gate Output.*

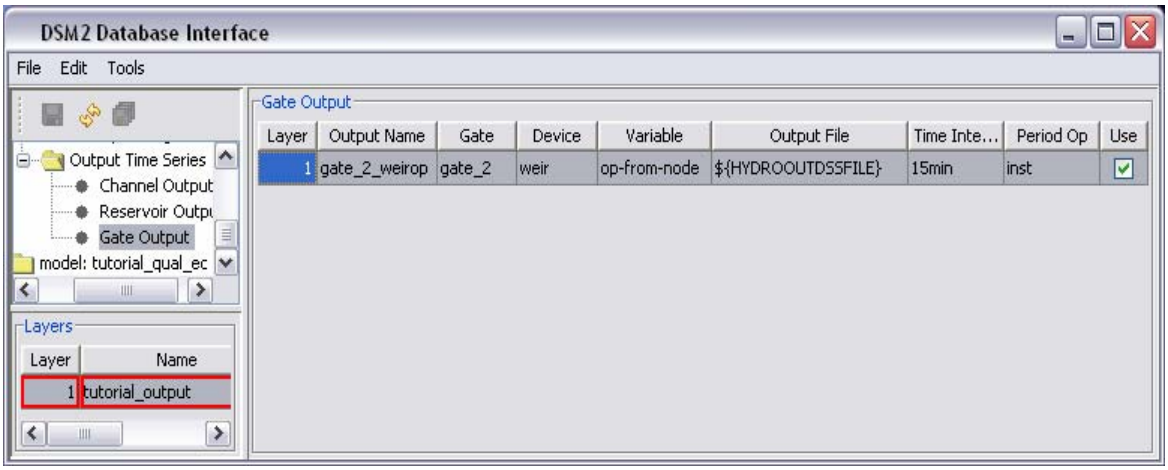
b. In the *Layer panel*, right-click and select *Set edit layer*.

c. In the *Select Layers* window, double-click the *tutorial_output* layer.

d. In the *Operating Rules* table:
   1) Right-click and select *Insert row*.
   2) Enter the following values into the appropriate fields:
      i) Output Name: *gate_2_weirop*
      ii) Gate: *gate_2*
      iii) Device: *weir*
      iv) Variable: *op-from-node*
      v) Output File: *${HYDROOUTDSSFILE}*
      vi) Time Interval: *15min*
      vii) Period Op: *inst*
      viii) Use: Make sure that the entry contains a checkmark.

e. Save the current settings.

f. At this point, the GUI should look as follows:



g. In Windows Explorer, navigate to the directory, *\{DSM2_home}\tutorial\simulations\simple\t6_oprule*.

h. Open the file, *hydro.inp*.

i. Add the following statements to the output paths section in order to view gate trigger locations:

```
trigger_loc  4    7500   stage  15min   inst  ${HYDROOUTDSSFILE}
ds_gate2     5    0      flow   15min   inst  ${HYDROOUTDSSFILE}
```

j. In the *Layers panel*, right-click and select *Unset edit layer* [optional].

3. Create an Operating Rule to Close the Weir when Stage is Low:

This operating rule closes the new gate we created during times where stage at a monitoring point is low. First we will define the rule in terms of an expression called *stage_critical* (the condition where stage violates a minimum) and *op_applies* (a seasonal condition that is True when we are controlling the gate for stage. In a later step we will define these variables.

a. In the *Simulations Navigator*:

1) Collapse the *Output Time Series* folder [optional].

2) Expand the *Operations* folder.

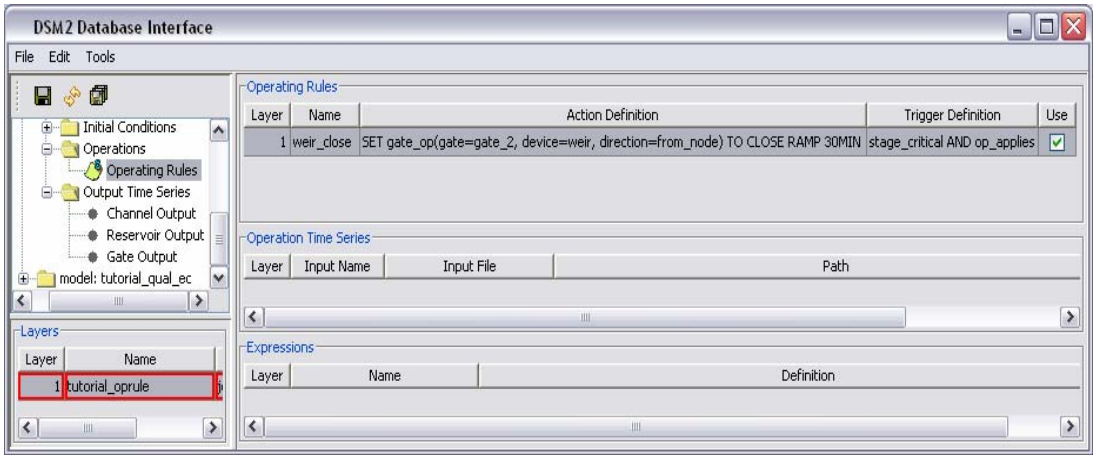
3) Double-click on *Operating Rules.*

b. Add a new Operating Rules Layer:

1) In the *Layer panel*, right-click and select *New layer*.

2) Select *Yes* to confirm the refresh.
3) Name the new layer, *tutorial_ oprule*, and add a description.
4) Enter *1* for the layer number.

c. In the *Layer panel*, right-click and select *Set edit layer*.
d. In the *Select Layers* window, double-click the *tutorial_oprule* layer.
e. In the *Operating Rules* table:
1) Right-click and select *Insert row*.
2) Enter the following values into the appropriate fields:
i) Name: *weir_close*
ii) Action Definition: *SET gate_op(gate=gate_2, device=weir, direction=from_node) TO CLOSE RAMP 30MIN*
iii) Trigger Definition: *stage_critical AND op_applies*
iv) Use: Make sure that the entry contains a checkmark.
f. Save the current settings.
g. At this point, the GUI should look as follows:



Note that the expressions stage_critical and op_applies will be created in a later step.

3. Create an Operating Rule to Open the Weir when Stage is High:

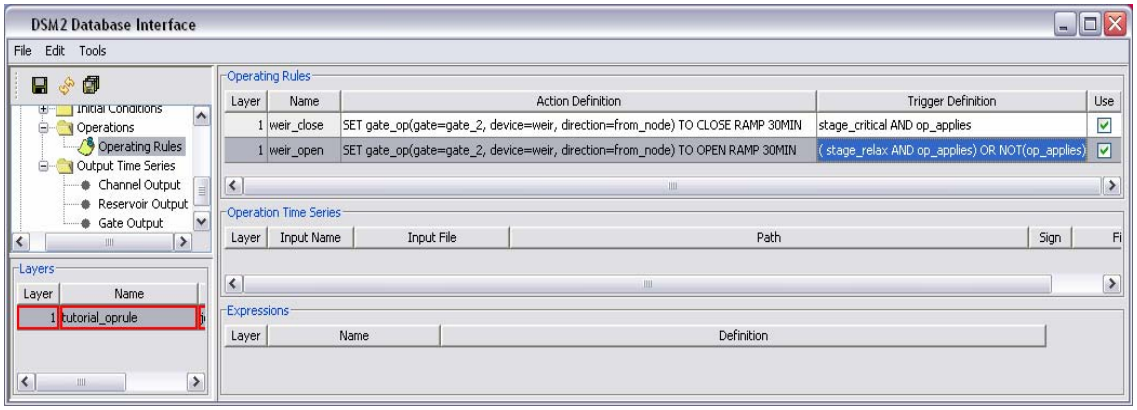
As before, we will enter the rule to open the weir first in terms of the expressions *stage_relax* (a condition where stage is safely above a threshold where we can open the gate) and *op_applies*. In the next step we will define these expressions.

a. In the *Operating Rules table*:
1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:
   i) Name: *weir_open*
   ii) Action Definition: *SET gate_op(gate=gate_2, device=weir, direction=from_node) TO OPEN RAMP 30MIN*
   iii) Trigger Definition: (stage_relax AND op_applies) OR NOT(op_applies)
   iv) Use: Make sure that the entry contains a checkmark.

b. Save the current settings.

c. At this point, the GUI should look as follows:



d. In the *hydro.inp* file, add the following environmental variables and values into the ENVVARS section:

STAGE_CRITICAL 1.4
STAGE_RELAX 1.6

4. Define Expressions used in the rule

a. In the *Expressions* table:
   1) Right-click and select *Insert row*.
   2) Enter the following values into the appropriate fields:
      i) Name: *op_applies*
      ii) Definition: *SEASON < 01FEB*
   3) Again, right-click and select *Insert row*.
   4) Enter the following values into the appropriate fields:
      i) Name: *stage_critical*
      ii) Definition: *chan_stage(channel=4, dist=7500) < ${STAGE_CRITICAL}*

5) Once again, right-click and select *Insert row*.

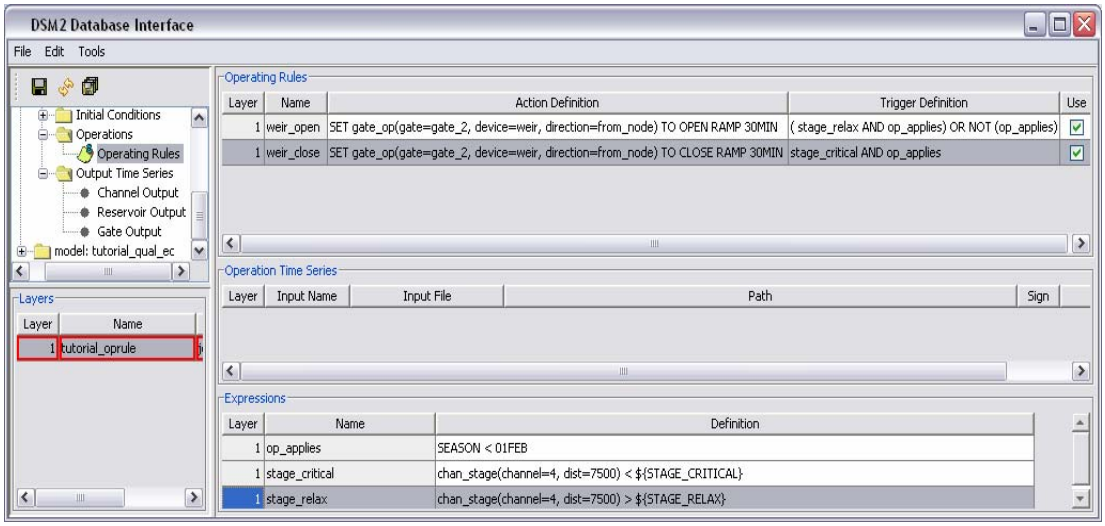
6) Enter the following values into the appropriate fields:

i) Name: *stage_relax*

ii) Definition: *chan_stage(channel=4, dist=7500) > ${STAGE_RELAX}*

b. Save the current settings.

c. At this point, the GUI should look as follows:



d. Now run HYDRO and QUAL:

1) Open a command window for the *t6_oprule* directory.

2) In the command window, type: *hydro hydro.inp*.

3) In the command window, type: *qual qual.inp*.

4) Open the *output.dss* file in the *t6_oprule* directory, and examine the results.

5. Add a Reduced Flow Operating Rule:

In our next operating rule, we will control the inflow to a node by having it toggle back and forth between a larger “full flow” and a reduced flow. First we will enter the rule and then we will define the full and reduced flows.

a. In the *Operating Rules table*:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

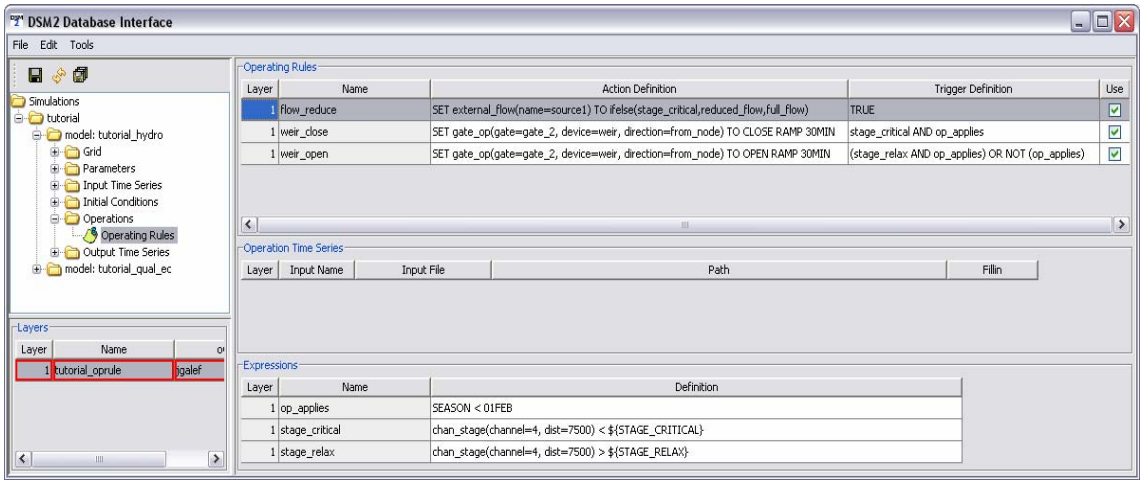
i) Name: *flow_reduce*

ii) Action Definiton: *SET external_flow(name=source1) TO ifelse(stage_critical,reduced_flow,full_flow)*

iii) Trigger Definition: *TRUE*

iv) Make sure the *Use* box is checked.

b. At this point, the GUI should look as follows:



c. Save the current settings.

d. Now create the expressions that define *full_flow* and *reduced_flow*. In the *Expressions* table:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields that define *full_flow.* This will involve the time series *source_flow* which we will enter later:

i) Input Name: *full_flow*

ii) Definition: *ts(name=source_flow) [note: this is a reference to a time series we haven't defined yet].*

3) Do the same for *reduced_flow.* Note: we are defining *reduced_*flow in terms of the time series. There is no guarantee of what order expressions will be evaluated, so you cannot safely define *reduced_flow* in terms of another expression such as *full_flow*. Enter the following values into the appropriate fields:

i) Input Name: *reduced_flow*

ii) Definition: *0.5*ts(name=source_flow).*

e. Save the current settings.

1) Now we will define the *source_flow* time series upon which the *full_flow* and *reduced_flow* expressions are based.

f. In the *Layer panel*, right-click and select *Set edit layer*.

g. In the *Select Layers* window, double-click the *tutorial_oprule* layer.

h. In the *Operation Time Series* table:

1) Right-click and select *Insert row*.

2) Enter the following values into the appropriate fields:

i) Input Name: *source_flow*

ii) Input File: *${TUTORIALINPUT}*

iii) Path: */TUTORIAL/SOURCE/FLOW//15MIN/CONSTANT/ [Note: there are two forward slashes between 15MIN and CONSTANT]*

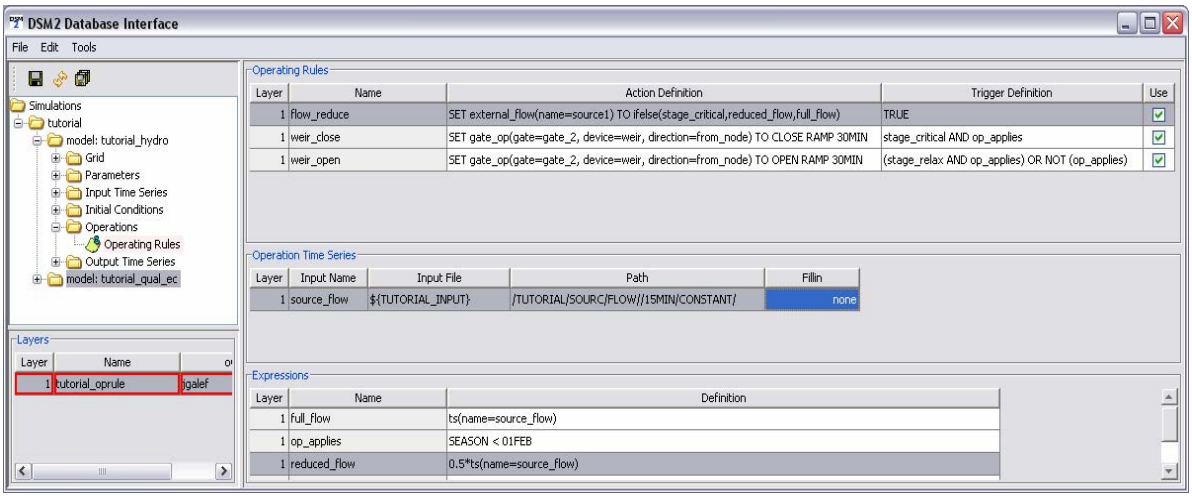
iv) Sign: *1*

v) Fillin: *none*

vi) Make sure the *Use* box is checked.

i. Save the current settings.

j. At this point, the GUI should look as follows:

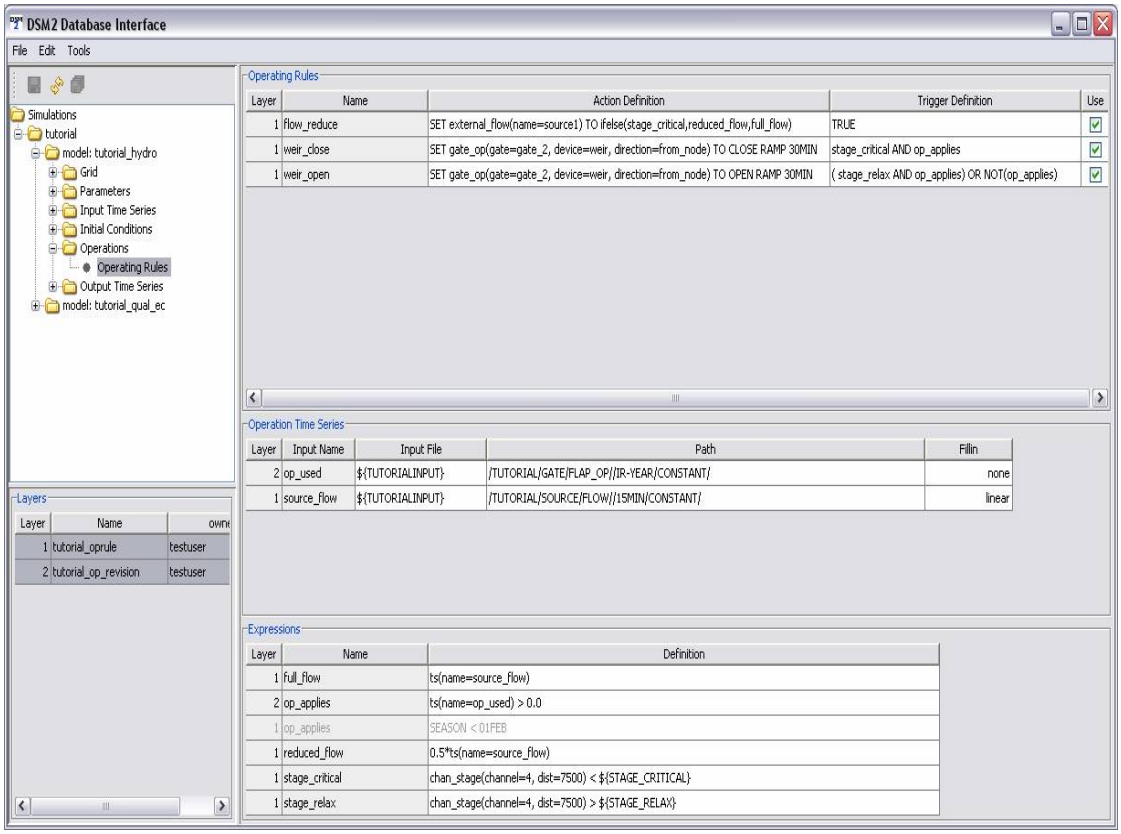


6. Override the Expression op_applies:

Recall that *op_applies* is used to determine when the weir is operated. Previously the definition of this expression was seasonal: the expression was SEASON < 01FEB.

The goal now is to make the same expression depend on a time series. Rather than change the expression, we will override it in a new layer.

a. Add a new Operating Rules Layer:
    1) In the *Layer panel*, right-click and select *New layer*.
    2) Select *Yes* to confirm the refresh.
    3) Name the new layer, *tutorial_ oprule_revision*, and add a description.
    4) Enter *2* for the layer number.
b. In the *Layer panel*, right-click and select *Set edit layer*.
c. In the *Select Layers* window, double-click the *tutorial_oprule_revision* layer.
d. Redefine the expressions that define *op_applies*. In the *Expressions* table:
    1) Right-click and select *Insert row*.
    2) Enter the following values into the appropriate fields:
        i) Input Name: *op_applies*
        ii) Definition: *ts(name=op_used) [note: this is a reference to a time series we will define in the next step].*
e. Define the time series *op_used* on which the *op_applies* expression depends. In the *Operation Time Series* table:
    1) Right-click and select *Insert row*.
    2) Enter the following values into the appropriate fields:
        i) Input Name: *op_used*
        ii) Input File: *${TUTORIALINPUT}*
        iii) Path: */TUTORIAL/GATE/FLAP_OP//IR-YEAR/CONSTANT/*
        iv) Sign: *1*
        v) Fillin: *none*
        vi) Make sure the *Use* box is checked.
f. At this point, the GUI should look as follows:



g. In the *Layers* panel, right-click and select *Unset edit layer* [optional].

h. Run HYDRO and QUAL and examine the results.

VIII. Tutorial 7: Delta SIMULATIONS with HYDRO and QUAL

1. Clone the Historical Simulation:

a. In the file menu of the GUI, select *Clone Simulation*.

b. In the *Clone Simulation* window, select *historical*.

c. Name the cloned simulation, *historical_tutorial*.

d. Rename the QUAL models:

1) In the *Sessions Navigator*:

i) Locate the directory, *model: historical_tutorial_qual1*.

ii) Right-click on the directory and select *Edit Model Properties.*

iii) Change the name to *historical_tutorial_qual_ec*.

iv) Repeat this process for *historical_tutorial_qual2*, except rename this model, *historical_tutorial_qual_do*.

2. Reset Model Name Parameters in the Text Input Files:

a. In Windows Explorer, navigate to the directory, *\{DSM2_home}\tutorial\simulations\historic*

b. Change the model name environmental variable for HYDRO and QUAL:

1) Open the file, *hydro.inp*.

i) Under the SCALAR section, locate the environmental variable, *model_name*.

ii) Note that *model_name* itself contains the environmental variable: *SIMNAME*.

iii) *SIMNAME* is not listed anywhere in the *hydro.inp* file. If fact, it is listed in the *config-hist.inp* file.

2) Open the file, *config-hist.inp*.

i) In the ENVVARS section, locate *SIMNAME*.

ii) Change its value from *historical* to *historical_tutorial*.

iii) Save *config-hist.inp*.

3) Because *qual_ec.inp* and *qual_do.inp* both utilize the environmental variable, *SIMNAME*, these files do not have to be modified.

c. Change the model name environmental variable for PTM:
   1) Open the file, *ptm.inp*.
      i) Under the SCALAR section, locate *model_name*.
      ii) Note that it is set to the different environmental variable, *MODEL_NAME*.
      iii) This environmental variable is listed in the ENVVARS section of this file.
      iv) Change the value of *MODEL_NAME* from *needs_hydro* to *historical_tutorial_hydro*, and save the file. Note that PTM requires the grid from HYDRO.
      v) Save *ptm.inp*.

d. Change the Time Interval Environmental Variable:
   1) In the GUI, the value given for all channel output is given as *${FINE_OUT}*. If you would like to verify this, you can check the *Channel Output* view of the *Output Time Series* directory for the HYDRO setup.
   2) Navigate back to the *config-hist.inp* file:
      i) Locate *FINE_OUT* under the ENVARRS section.
      ii) Change its value from *15MIN* to *1HOUR*.
      iii) Save *config-hist.inp*.

e. Select the Proper Database:
   1) In *hydro.inp*:
      i) Find the SCALAR section.
      ii) Change the *database* entry to the following:

          database dsm2input_access

      iii) Repeat this procedure for *qual_ec.inp*, *qual_do.inp*, and *ptm.inp*.

f. Change the Temporary Directory:
   1) In *hydro.inp*, *qual_ec.inp*, *qual_do.inp*, and *ptm.inp*
      i) In the SCALAR section, find the environmental variable, *temp_dir*.
      ii) Change the value to *c:\temp* or a temp directory with ample free space.
      iii) Remember to repeat this procedure for all four files.
      iv) Save the four files.

3. Run HYDRO:

a. In Windows Explorer, navigate to the directory, *\{DSM2_home}\tutorial\simulations*

b. Right-click on the *historic* directory, and select, *Open Command Window Here.*

c. In the command window, type: *dsm2 hydro config-hist.inp*.

d. Wait for HYDRO to complete its runs.

4. Replicate HYDRO's tidefiles and modify QUAL's input file to make them suitable for a Multi-tidefile Run:

a. In Windows Explorer, navigate to the directory, *\{DSM2_home}\tutorial\simulations\historic\output*

b. Locate the *hist.h5* tidefile and copy-and-paste this file three times in the same directory. These copies will emulate hydro tidefiles generated on three separate machines.

c. Rename the copies: *tf1.h5*, *tf2.h5*, and *tf3.h5*.

d. Open *qual_ec.inp*:

1) If necessary, make corrections so that the TIDEFILE section looks as follows:

```
TIDEFILE
START_DATE    END_DATE     FILENAME
#runtime       length       output/hist.h5
runtime        20JUL1996    output/tf1.h5
20JUL1996      24JUL1996    output/tf2.h5
last           length       output/tf3.h5
01SEP1996      length       ficticious.h5
END
```

2) With the exception of the commented-out (#) statement, these statements direct QUAL to the *hydro/output* location storing the tidefiles needed for the multi-tide run. The start and end date show the possibilities: using a real date, runtime (start of the simulation), length (go to end of tidefile) and last (start where the last tidefile left off).

3) Change the temporary directory from *z:\temp* to a temp location with ample free storage.

4) Save *qual_ec.inp*.

5. Run QUAL:

a. In the command window, type: *dsm2 qual_ec config-hist.inp*.

6. Running QUAL with Nonconservative Constituents and a Single Tidefile:

a. In Windows Explorer, navigate to the directory, *d:\{DSM2_home}\tutorial\simulations\historic*

b. Open *qual_do.inp*.

1) In the TIDEFILE section, ensure that only one row for tidefiles is present and uncommented. This should be the row associated with the file, *hist.h5*.

2) Change the temporary directory from *z:\temp* to a temp location with ample free storage

3) Save the *qual_do.inp* file.

c. Open *config-hist.inp*.

1) In the ENVVARS section, locate the environmental variable, DSM2MODIFIER.

2) Change the value from *hist* to *nonconserve*.

3) Save *config-hist.inp*.

d. In the command window, type: *dsm2 qual_do config-hist.inp*.

IX. Tutorial 8: Delta SIMULATIONS with PTM

1. Modify the PTM Input file to Turn On the Dispersion Parameters:

a. In Windows Explorer, navigate to the directory, *\{DSM2_home}\tutorial\simulations\historic*

b. Open the file, *ptm.inp*.

1) In the TIDEFILE section, ensure that only one row for tidefiles is present. This should be the row associated with the file, *hist.h5*.

2) Locate the GROUPS section.

3) Add two groups to the list:

sac_below_chipps	*channels*	*range:438-441*
sac_below_chipps	*channels*	*(443|452)*

4) Locate the GROUP_OUTPUT section.

5) Add this group_outputs to the list:

   sac_below_chipps 1hour ${PTMOUTPUTFILE} SAC_BELOW_CHIPPS

6) Locate the SCALARS section.

7) Change all of the dispersion parameters from *f* to *t*.

```
ptm_ivert       t       # Use Vertical velocity profile
ptm_itrans      t       # Use Transverse velocity profile
ptm_iey         t       # Use transverse mixing
ptm_iez         t       # Use vertical mixing
```

2. Run PTM and Examine the Results:

a. In the command window, type: *dsm2 ptm config-hist.inp*.

b. In Windows Explorer:

1) Navigate to the directory,

   \{DSM2_home}\tutorial\simulations\historic\output

2) Examine the output in the *ptmout.txt* file.

3) Copy the files, *anim_db.bin* and *ptmout.txt*.

4) Navigate to the directory,

   \{DSM2_home}\tutorial\simulations\historic\ptm-animate\dual\left_panel

5) Paste the files in the *left_panel* directory.

3. Now to See the Effects of having the Dispersions Parameters Turned Off:

a. In Windows Explorer, navigate to the directory,

   \{DSM2_home}\tutorial\simulations\historic

b. Open the file, *ptm.inp*.

1) Locate the SCALARS section.

2) Change all of the dispersion parameters from *t* to *f*.

```
ptm_ivert       f       # Use Vertical velocity profile
ptm_itrans      f       # Use Transverse velocity profile
ptm_iey         f       # Use transverse mixing
ptm_iez         f       # Use vertical mixing
```

c. In the command window, type: *dsm2 ptm config-hist.inp*.

d. In Windows Explorer:
    1) Navigate to the directory,
    \{*DSM2_home*}*tutorial**simulations**historic**output*
    2) Copy the files, *anim_db.bin* and *ptmout.txt*.
    3) Navigate to the directory,
    \{*DSM2_home*}*tutorial**simulations**historic**ptm-animate**dual**right_panel*
    4) Paste the files in the *right_panel* directory.
    5) Navigate to the directory,
    \{*DSM2_home*}*tutorial**simulations**historic**ptm-animate*
    6) Double-click on *dual.bat* to open the animator.
    7) Press start to start the animator and use the controls to adjust the speed.

4. Modifying the Animator Display [optional]:

a. The *left_panel* and *right_panel* directories contain files needed for operation:
    1) *fluxInfoDB.data* stores path information for the PTM output.
    2) *labelsDB.data* stores labels information.
    3) *network.dat* stores *x*- and *y*-locations for nodes and channels.
b. Examine these files.

5. Changing an Operator Rule:

a. Bring up the GUI.
b. In the *Sessions Navigator*:
    1) Expand the folder, *model: historical_hydro*.
    2) Expand the *Operations* folder.
    3) Double-click on *Operating Rules*.
c. In the *Layers* panel, click on *Layer 1* so that it is the only layer viewed.
d. Examine the rules for this gate at the Montezuma Slough.
e. Add a New Operating Rule Layer:
    1) In the *Layers* panel, right-click and select *New layer*.
    2) Select *Yes* to confirm the refresh.

3) Name the layer, *modified_montezuma_ops*.
4) The layer will be number *4*.

f. In the *Layers* panel, right-click and select *Set Edit Layer*.
g. In the *Select Layer* window, double-click *Layer 4*.
h. In the *Expressions table*:
   1) Right-click and select *Insert row*.
   2) Enter the information in the appropriate fields for the new row:
      i) Name: *mscs_calc*
      ii) Definition: *SEASON > 01AUG OR SEASON < 01FEB*
i. Save the current settings.
j. Unset the editing layer.
k. In the *Sessions Navigator*:
   1) Expand the *model: historical_tutorial_hydro* folder.
   2) Expand the *Output Time Series* directory.
   3) Double-click on *Gate Output*.
l. In the *Layers* panel, right-click and select *Add layer*.
m. In the *Select Layer* window, double-click *grid_output*.
n. In the *Layers* panel, right-click and select *Remove layer*.
o. Select *std_output_hydro_named* (note, if you omit this step there will be too many output specifications and the model will crash).
p. Rerun HYDRO:
   1) In the command window, type: *dsm2 hydro config-hist.inp*