SMS 12.3 Tutorial
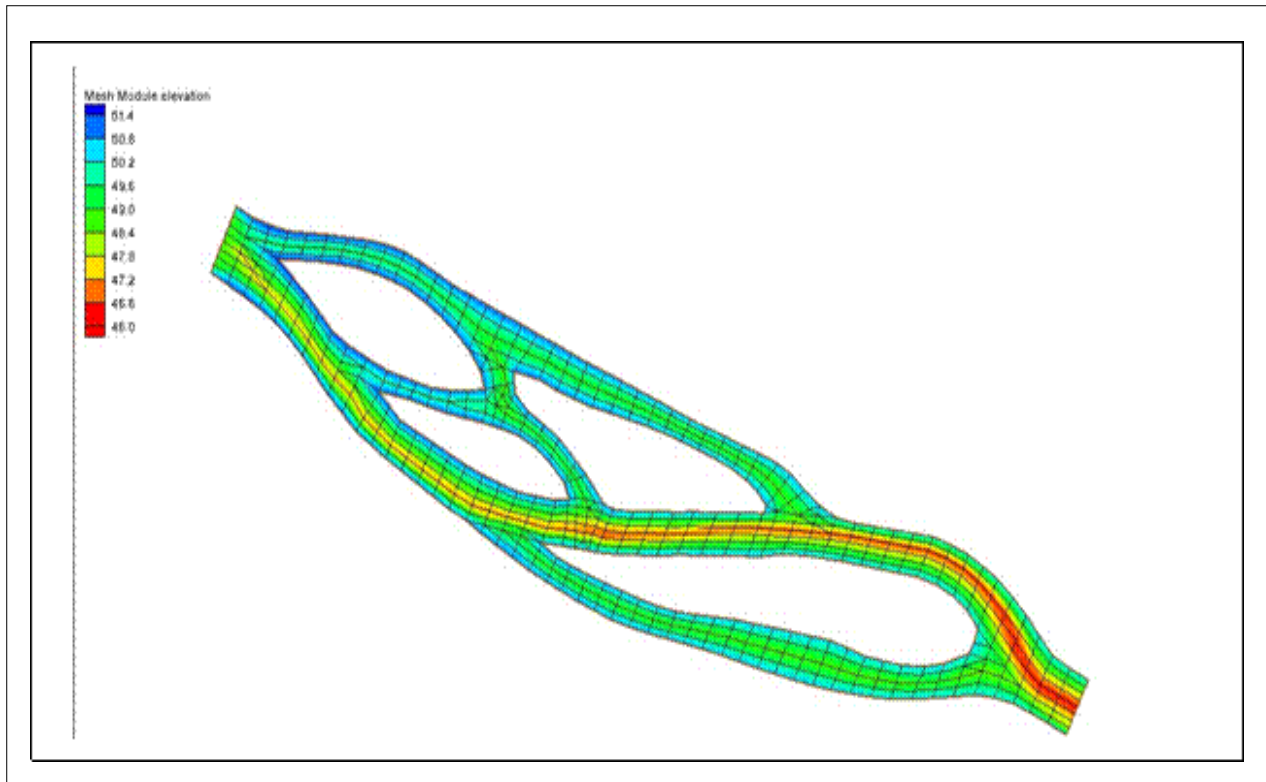
Observations

Objectives

This tutorial will give an overview of using the observation coverage in SMS. Observation points will be created to measure the numerical analysis with measured field data and calibration. Observation arcs will be set to view the results of cross sections or river profiles.

Prerequisites

- None

Requirements

- Generic Mesh Model
- Map Module
- Mesh Module

Time

- 30–45 minutes

AQUAVEO™

1   Introduction

An important part of any computer model is the verification of results. Surface water modeling is no exception. Before using a surface water model to predict results, the model must successfully simulate observed behavior. Calibration is the process of altering model input parameters (within an accepted range) until the computed solution matches observed field values (or at least as well as possible). SMS contains a suite of tools in the observation coverage to assist in the model verification and calibration processes.

The observation coverage consists of observation points and observation arcs, which help analyze the solution for a model. Observation points can be used to verify the numerical analysis with measured field data and calibration. They can also be used to see how data changes through time. Observation arcs can be used to view the results for cross sections or river profiles.

This tutorial uses meshes and solutions created with a FESWMS finite element model, but the calibration tools in SMS can be used with any model. It is not necessary to have the FESWMS model enabled to complete this tutorial as the exercises do not require running the model.

2   Opening the Data

To open the simulation and solution data:

1. Select *File* / **Open…** to bring up the *Open* dialog.

2. Select "Project Files (*.sms)" from the *Files of type* drop-down.

3. Browse to the *data files* folder for this tutorial and select "observe1.sms".

4. Click **Open** to import the project and exit the *Open* dialog.

5. If asked to delete existing data, click **Yes**.
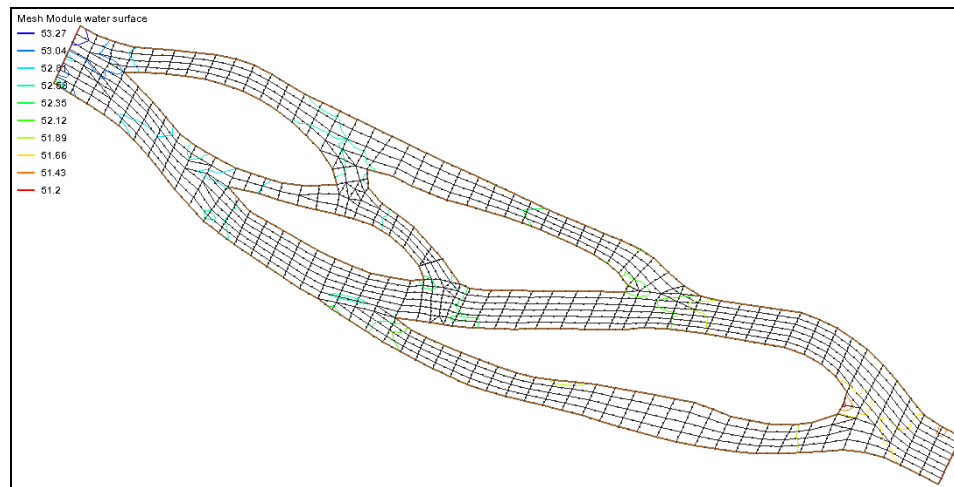
The project should appear similar to Figure 1.

Figure 1   Initial project contained in "observe1.sms"

3   Viewing Solution Data

An initial solution has already been created with this data file and was opened with the project. When the solution file is opened into SMS, various scalar and vector datasets are created.

Several display options should be changed by doing the following:

1. Select "⊞ velocity mag" and "↗ velocity" to make them active.

2. Right-click on "⬓ Mesh Data" and select **Display Options...** to open the *Display Options* dialog.

3. Select "2D Mesh" from the list on the left.

4. On the *2D Mesh* tab, click **All off** and turn on *Contours, Nodestrings*, and *Mesh boundary*.

5. Click **OK** to exit the *Display Options* dialog.
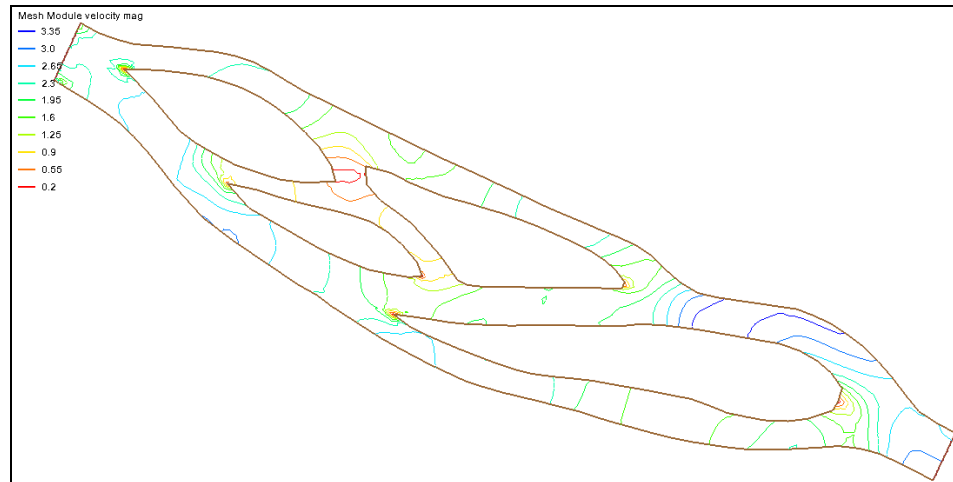
The mesh data should now appear similar to Figure 2.

Figure 2   Mesh after Display Options modified

4   Creating an Observation Coverage

The calibration tools utilize observation features in an observation coverage. To create an observation coverage:

1.   Click on " Area Property" in the Project Explorer to make it active.

2.   Right-click on " Area Property" and select **Rename**.

3.   Enter "calibration data" and press *Enter* to set the new name.

4.   Right-click on " Area Property" and select *Type | Generic |* **Observation**.

5   The Observation Coverage

The *Observation Coverage* dialog can now be used to specify what data to use in calibrating the model and to edit observation points and arcs.

1.   Select *Feature Objects |* **Attributes…** to bring up the *Observation Coverage* dialog.

In this tutorial, observation points will be used to calibrate the model; however, observation arcs or a combination of arcs and points can be used instead depending on the data collected in the field. Observation arcs work similarly to observation points. Differences will be pointed out as the tutorial proceeds.

The *Observation Coverage* dialog can display the attributes for either observation points or observation arcs, but not both at the same time. The *Feature object type* drop-down in the upper right corner determines which attributes are currently being displayed in the *Observation Coverage* dialog.

The upper spreadsheet is called the *Measurements* spreadsheet and the lower spreadsheet is called the *Observation Objects* spreadsheet. The titles of these spreadsheets change depending on what is selected as the feature object. Right now, the title of the *Measurements* spreadsheet is simply "Measurements" and the title of the *Observation Objects* spreadsheet is "Observation Points".

2.   Select "arcs" from the *Feature object type* drop-down.

The titles of the *Measurements* and *Observation Objects* spreadsheets will change to "Flux Measurements" and "Observation Arcs," respectively.

3.   Change the *Feature object type* drop-down back to "points".

Note that observation points use single values measured in the field, such as velocity and water surface elevation, to calibrate the model. On the other hand, observation arcs use fluxes that have been computed across the arc to calibrate the model. Therefore, measurements for observation arcs are called "Flux Measurements".

5.1   Creating a Measurement

By default, when the *Observation Coverage* dialog is first opened, a measurement does not exist. A measurement represents the solution data that is compared to the observed field data in the calibration process. For observation points, a measurement is tied to either a scalar or a vector dataset. This dataset is unique to the measurement and cannot be tied to another measurement. For observation arcs, a measurement is tied to both a scalar and a vector dataset. Again, this combination of datasets is unique to the measurement.

In addition to a unique *Name* and *Dataset*, two other parameters are used to define the data represented by a measurement: *Trans* and *Module*. When analyzing data that varies through time, check the box in the *Trans* column. The *Module* of a measurement refers to the SMS module where the computed data is stored.

To create a new measurement, do the following:

1.   In the *Measurements* section, select "points" from the *Feature object type* drop-down.

2.   Enter "Velocity" in the *Name* column on row *1* of the spreadsheet.

3.   Select "velocity" from the drop-down in the *Dataset* column (not "velocity mag").

Now that a measurement has been defined, observation points can be created and edited.

6   Creating an Observation Point

Observation points are created at locations in the model where the velocity or water surface elevation has been measured in the field. The measured values will be compared with the values computed by the model to determine the model's accuracy. In addition to being assigned a *Color* and a *Name*, each observation point is assigned the following data:

- The *x* and *y* coordinates, a real world location of the point, needs to be specified. Observation arcs do not have these location attributes since several points define an arc.

- *Angle*. When a measurement for observation points is tied to a vector dataset (as is the case with the "Velocity" measurement created in the previous section) an angle needs to be specified. This angle is an azimuth angle with the top of the screen representing north when in plan view.

- *Observed value*. The observed value is the value that was measured in the field corresponding to the active measurement.

- *Interval*. The interval is the allowable error (±) between the computed value and the observed value. Model verification is achieved when the error is within the interval (±) of the observed value.

Create an observation point by doing the following:

1. In the *Observation Points* section, enter the values shown in the table below for *Name*, *x* and *y* coordinates, *Observed Value*, and *Interval*. Do not enter anything in the *Angle* field at this time.

Name	x	y	Angle	Observed Value	Interval
Point 1	190.0	-369.0	0.0	3.5	0.25

2. In the *Observe* column, check the box.

When the *Observe* box for a point or arc is turned on, it is said to be observed. Note that the *Observation Points* spreadsheet will always end with a blank line for the creation of additional points. The *Observation Arcs* spreadsheet will not contain any blank lines because arcs cannot be created while in the *Observation Coverage* dialog

3. Click **OK** to close the *Observation Coverage* dialog.

An observation point has now been created at the location specified in the *Observation Coverage* dialog. In this case, the model will be verified if the computed value is ± 0.25 fps of the observed velocity, or between 3.25 and 3.75 fps.

No angle has been specified for this point. This angle can be specified in the *Observation Coverage* dialog or in the Main Graphics Window.

To make sure the observation point is displaying correctly, turn on the display of points by doing the following:

1. Click **Display Options** 🖥 to bring up the *Display Options* dialog.

2. Select "Map" in the list on the left.

3. Turn on *Point* and *Calibration target*, then click **OK** to close the *Display Options* dialog.

A point with an arrow pointing up will appear in the Graphics Window (Figure 3). A calibration target is drawn next to the point.
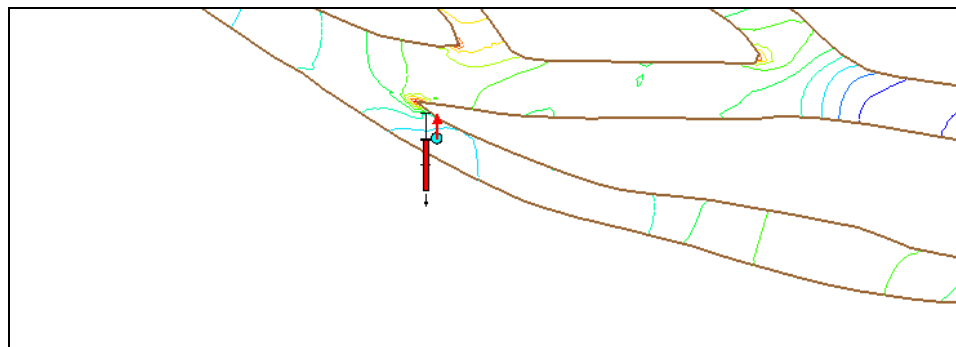

Figure 3   Observation point with calibration target

To specify the angle in the Main Graphics Window:

1. **Zoom** in on the observation point.

2. Using the **Select Feature Point** tool, select the observation point arrow and rotate the point arrow about 120˚ by dragging the end of the arrow clockwise.

The angle does not need to be exactly 120˚. The arrow only needs to be pointing in the general direction in which the velocity meter was set up in the field. This is usually in the direction of flow (as is the case here). Figure 3 shows "Point 1" with the arrow pointing up (0˚ angle), and Figure 4 shows the arrow at an angle of approximately 120˚.
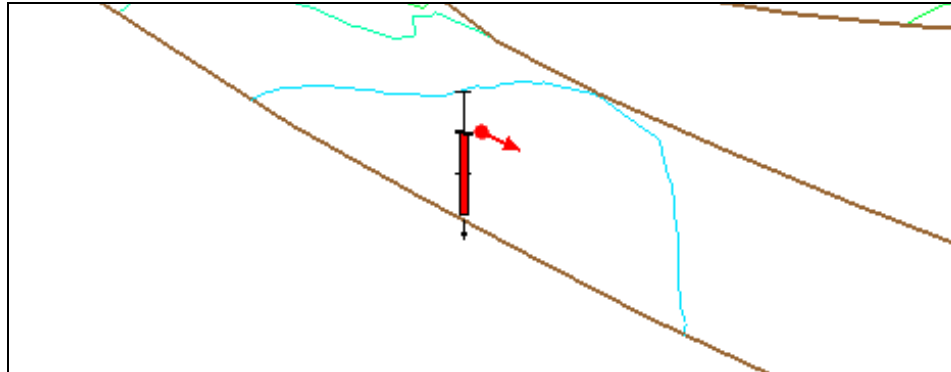

Figure 4   "Point 1" with the arrow rotated to 120º

6.1   Using the Calibration Target

A calibration target is drawn next to the observation point. The components of a calibration target are illustrated in Figure 5. These components are:

- *Target Middle*. This is the target (observed) value that was measured in the field.

- *Target Extents*. The top of the target represents the target value plus the interval, while the bottom represents the target value minus the interval.

- *Color Bar*. The color bar shows the error between the observed value and the computed value. If the bar is entirely within the target, the color bar is drawn in green. If the error is less than twice the interval, the bar is drawn in yellow. A larger error will be drawn in red.

For this example, the bar would be green if the computed value is between 3.25 and 3.75, yellow for values between 3.0–3.25 or 3.75–4.0, and red for values smaller than 3.0 or greater than 4.0.
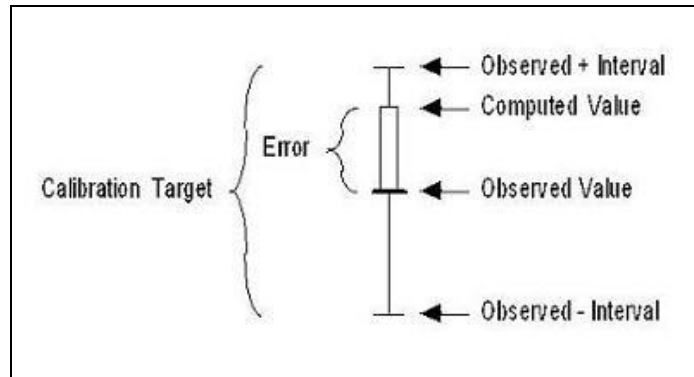
Figure 5   Calibration target

Now that the observation point has been created the target appears. The color bar in this example is red with an arrow pointing down, indicating that the computed solution has a velocity below 3.5 fps.

6.2   Multiple Measurements

Each observation point has attributes for all measurements. Similarly, each observation arc has attributes for each flux measurement. The highlighted measurement in the *Measurements* spreadsheet determines which attributes are shown in the *Observation Objects* spreadsheet.

For example, to create a new measurement:

1. Using the **Select Feature Point** ⟋⟍ tool, double-click "Point 1" to bring up the *Observation Coverage* dialog.

2. In the *Measurements* section, enter "WSE" on the second row in the *Name* column in the spreadsheet.

As with the spreadsheet in the *Observation Points* section, this spreadsheet will always end with a blank line to allow for the creation of additional measurements.

3. Press *Enter* or *Tab* to create the new measurement.

4. In the *Dataset* column, select "water surface" from the drop-down.

Note that this new measurement is now the active measurement and it is also highlighted. Several measurements can exist at a time, but calibration targets will only be displayed in the Main Graphics Window for *Observed* points in the active measurement.

In the *Observation Points* spreadsheet, the *Name*, *Color*, and *x* and *y* coordinates have remained the same for "Point 1". However, the *Observed Value* and *Interval* have reset to their default values, and there is no *Angle* column since this new measurement is tied to a scalar dataset. These attributes are for the measurement named "WSE". To view the observation point attributes previously specified for the *Velocity* measurement, select the *Velocity* row in the *Measurements* spreadsheet.

Because both the "WSE" and "Velocity" measurements will be used to calibrate the model, do not delete the "WSE" measurement at this time.

7   Reading a Set of Observation Points

Using the steps defined above, multiple observation points can be created. However, this process could become tedious for a large set of points. Normally, the data defining the points will be in spreadsheet format and can simply be copied and pasted in the *Observation Points* spreadsheet.

To copy the data from the external spreadsheet, do the following:

1. Outside of SMS, open the file "observepts.xls" in a spreadsheet program.

The data is also contained in a tab delimited file named "observepts.txt".

2. Select the cell containing "Point 2" in the *Name* column, then hold down the *Shift* key and select the cell in the second *int* column on the "Point 8" row.

The data for "Point 1" is not copied since "Point 1" has already been created.

3. Press *Ctrl + C* to copy the selected data and return to SMS.

4. In the *Observation Coverage* dialog, in the spreadsheet in the *Measurement* section, check the box in the *Active* column on row *1* to make "Velocity" the active measurement.

5. In the *Observation Points* section on row 2, select cell in the *Name* column and press *Ctrl+V* to copy data into the *Observation Points* spreadsheet.

6. Click **All On** button to make the all of the points observed.

7. Click **OK** to close the *Observation Coverage* dialog.

Seven new observation points distributed around the finite element mesh should appear in the Graphics Window (Figure 6).
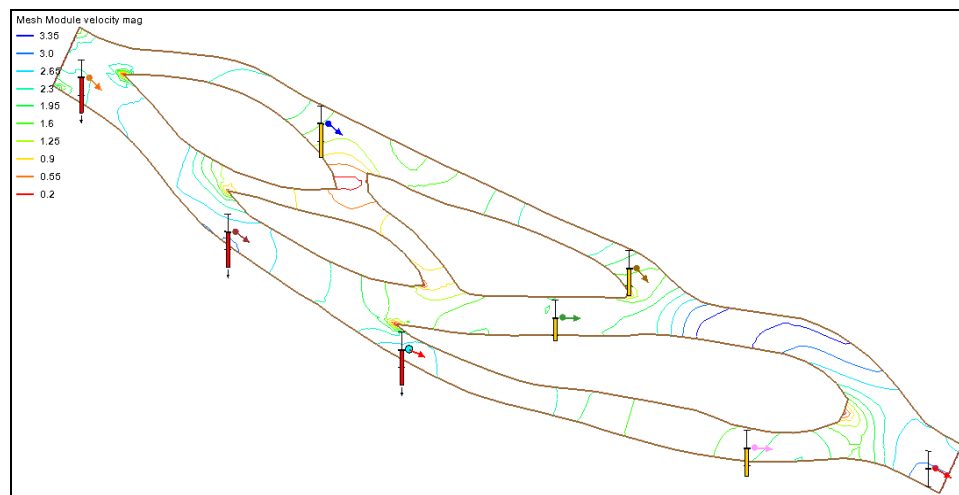

Figure 6   Observation points imported from the file "observepts.xls"

Now specify the observed values and the confidence interval for the "WSE" measurement by doing the following:

8. Using the **Select Feature Point** tool, double-click on one of the observation points to open the *Observation Coverage* dialog.

9. Outside of SMS, using the same spreadsheet file opened earlier, select the cell on the *Point 1* row in the *wse* column, then hold down the *Shift* key and select the cell on the *Point 8* row in the second *int* column.

10. Press *Ctrl + C* and return to SMS.

11. In the *Observation Coverage* dialog, in the spreadsheet in the *Measurements* section, check the box in the *Active* column on the second row so the "WSE" measurement is active and highlighted.

12. In the *Observation Points* section, select the cell on row *1* in the *Observed Value* column and press *Ctrl + V* to copy the data into the *Observed Value* and *Interval* columns.

13. Click **OK** to close the *Observation Coverage* dialog.

The observation points that appear in the Main Graphics Window do not have arrows since the active measurement is observing a scalar dataset (Figure 7).

When calibrating a model, the goal is to calibrate the model so that the computed values from the model fall within the confidence intervals of the observed field data for all measurements. At times this is difficult, and personal discretion is required to determine when the model has been sufficiently calibrated.
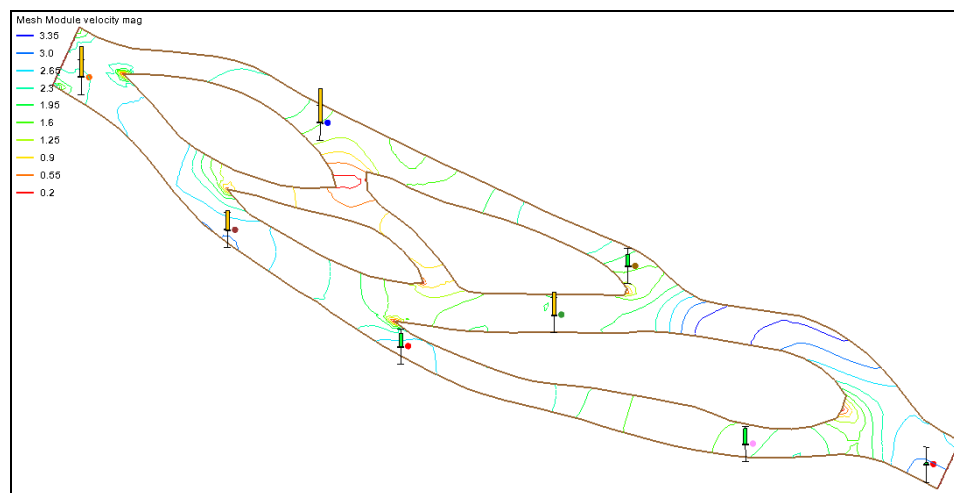
Figure 7   Observation points showing WSE scalar data

8   Generating Error Plots

Before continuing, make the "Velocity" measurement the active measurement by doing the following:

1. Using the **Select Feature Point** tool, double-click any of the points to open the *Observation Coverage* dialog.

2. In the spreadsheet in the *Measurements* section, check the box in the *Active* column on the first row so the "Velocity" measurement is active and highlighted.

3. Click **OK** to close the *Observation Coverage* dialog.

SMS can create several types of plots to analyze the error between the computed and observed values. To create a "Computed vs. Observed Data" plot and an "Error Summary" plot, do the following:

1. Select *Display* / **Plot Wizard…** to bring up the *Step 1 of 2* page of the *Plot Wizard* dialog.

2. In the *Plot Type* section, select "Computed vs. Observed Data" from the list on the left and click **Next** to go to the *Step 2 of 2* page of the *Plot Wizard* dialog.

3. In the top section, select "Velocity" from the *Measurement* drop-down and click **Finish** to close the *Plot Wizard* and bring up the *Plot 1* dialog (Figure 8).

4. Move the *Plot 1* dialog out of the way and repeat steps 1–3, selecting "Error Summary" in step 2.

Both plots have now been created (Figure 8 and Figure 9). Each plot exists in a separate window that can be resized, moved, and closed at any time.
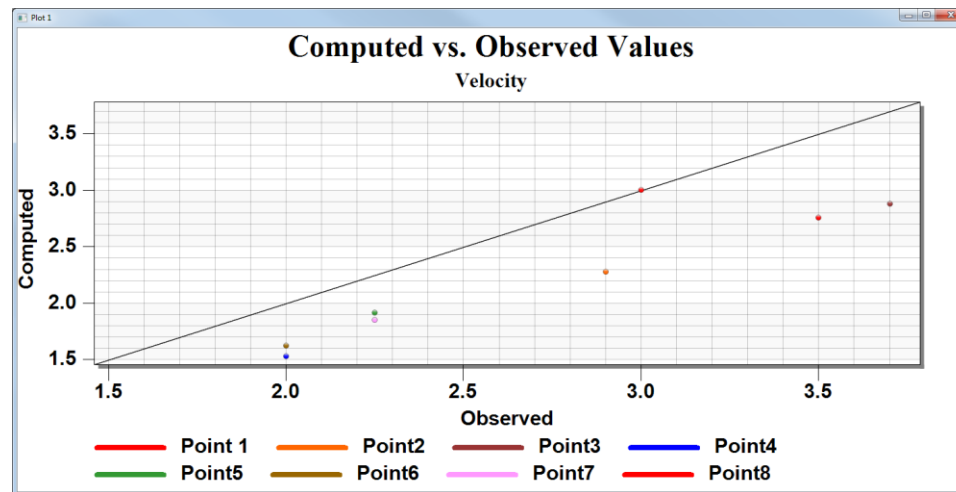
Figure 8   Computed vs. Observed Data plots using the Velocity dataset
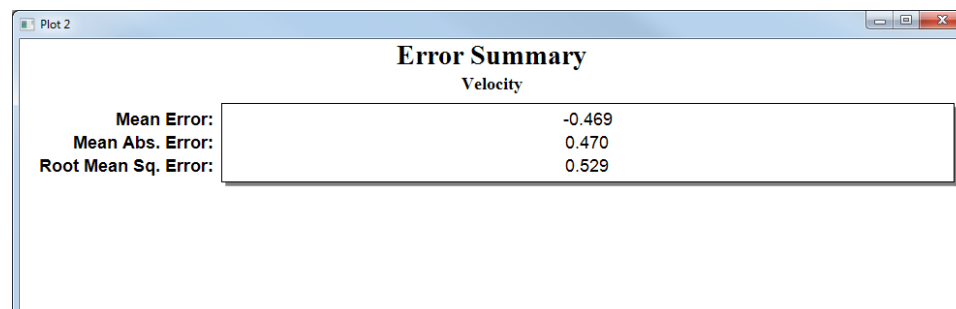
Figure 9   Error Summary using Velocity dataset

8.1   Plot Data

More plots can also be created for the "WSE" measurement or the current plots can be edited. To edit a plot:

1. Right-click the *Error Summary* plot in the Plot 2 dialog and select **Plot Data…** to bring up the *Data Options* dialog.

2. Select "WSE" from the *Measurement* drop-down and click **OK** to close the *Data Options* dialog.

The *Error Summary* plot is now updated using the data from the "WSE" measurement (Figure 10).

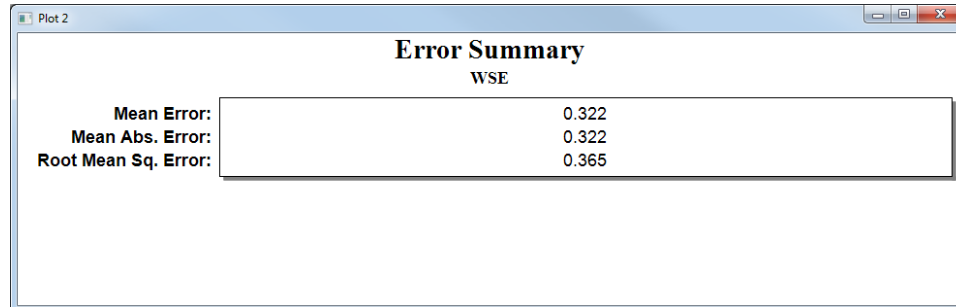<p align="center">*Figure 10   Updated error summary using WSE dataset*</p>

8.2   Using the Computed vs. Observed Data Plot

In the *Computed vs. Observed Data* plot, a symbol is drawn for each of the observation points. A point that plots on or near the diagonal line indicates a low error. Points far from the diagonal have a larger error. The position of the points relative to the line gives an indication whether the computed values are consistently higher or lower than the observed values. In this case, all points are below the line indicating that all computed velocities are lower than observed values.

To change the plotted values to use the WSE dataset, do the following:

1. Right-click the *Computed vs. Observed Values* plot in the Plot 1 dialog and select **Plot Data…** to bring up the *Data Options* dialog.

2. Select "WSE" from the *Measurement* drop-down and click **OK** to close the *Data Options* dialog.

All points should now plot above the line, indicating that all computed water surface elevations are above the observed values (Figure 11).
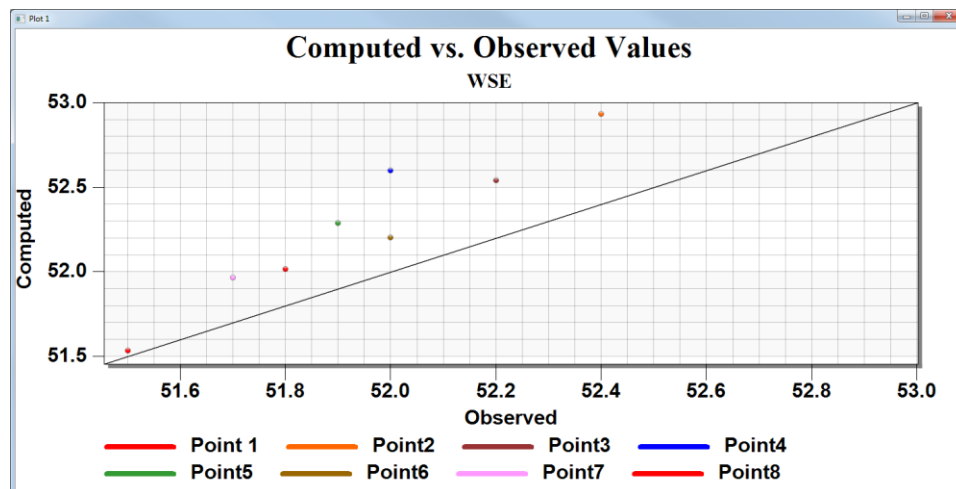
<p align="center">*Figure 11   Computed vs. Observed Data plots using the WSE dataset*</p>

8.3   Using the Error Summary Plot

In the *Error Summary* plot, the following three types of error norms are reported:

- *Mean Error*. This is the average error for the points. This value can be misleading since positive and negative errors can cancel.

- *Mean Absolute Error*. This is the mean of the absolute values of the errors. It is a true mean, not allowing positive and negative errors to cancel.

- *Root Mean Square Error*. This takes the sum of the square of the errors and then takes its square root. This norm tends to give more weight to cases where a few extreme error values exist.

Before continuing, do the following:

1. Close the two plot dialogs by clicking on the ☒ at the upper right corner of the window.

2. **Frame** 🔍 the project.

9   Calibrating the Model

The values in this solution for both measurements are not within the calibration targets. To achieve better calibration, the material properties can be changed and then the model can be re-run.

The errors through the main channel for the "Velocity" measurement are negative. This indicates that the observed velocities are larger than those computed by the model. In order to increase the velocity in these locations, it is necessary to change the parameters (eddy viscosity and/or Manning's *n*). Increasing the velocity at these locations should also decrease the water surface elevation.

To save time, solutions have been provided. In each solution the eddy viscosity parameter was changed before the model run. The eddy viscosity values used where 10.0 (observe1), 1.5 (observe2), 6.0 (observe3), and 7.0 (observe4).

9.1   Importing the New Solution

In the first attempt to calibrate the model, the eddy viscosity was set to 1.5 and then the model was run. To import the solutions for the second simulation, do the following:

1. Select *File* / **Open...** to bring up the *Open* dialog.

2. Select "observe2.h5" and click **Open** to import the solution and exit the *Open* dialog.

3. Select the "📈 velocity" and "📋 velocity mag" datasets under the "📁 observe1.flo" solution folder to make them active.

4. Switch to the ✦ᴺ **Map** module.

5. Using the **Select Feature Point** 𝄏 tool, double-click any of the points to open the *Observation Coverage* dialog.

6. In the *Measurements* section, select "velocity" from the drop-down in the *Dataset* column (not "velocity mag").

7. Click **OK** to close the *Observation Coverage* dialog.

Solutions for both the *observe1* and *observe2* models should be visible (Figure 12). The calibration targets will automatically update to show the errors for the solution that was just opened.

The verification targets now show that six points for the "Velocity" measurement are within the allowable range and two points are above the range, but still in the yellow range. There are no points more than the two times the variation above the observed value (red targets). Looking at targets for the "WSE" measurement, four points are within the allowable range and four are below the range with one point being more than two times the variation below the observed value.

Since the values for the "Velocity" measurement that are unacceptable are now higher than the observed values and the values for the "WSE" measurement that are unacceptable are now lower than the observed values, the correction made was too drastic. Specifically, the eddy viscosity was lowered too much and it needs to be raised.
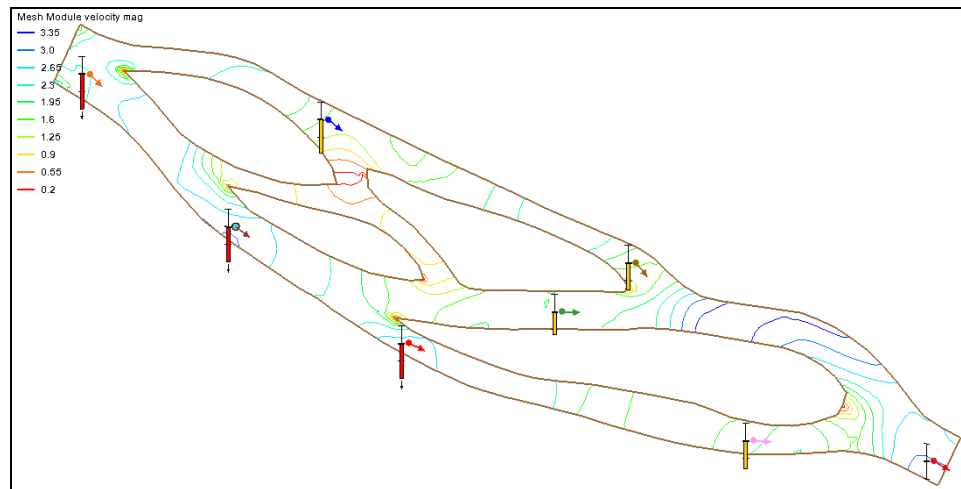
Figure 12   Calibration targets for the imported observe2.h5

9.2   Fine-tuning the Model

Since the eddy viscosity was too low on the second model run, a third model run was completed using an eddy viscosity of 6.0. To view this set of solutions:

1. Repeat steps 1–3 in section 9.1, opening "observe3.h5" in step 2.

2. Select the " velocity" and " velocity mag" datasets under the " observe3.flo" solution folder under " mesh (2)" to make them active.

Notice that all the observation point targets for the "Velocity" measurement are within the acceptable intervals (Figure 13).
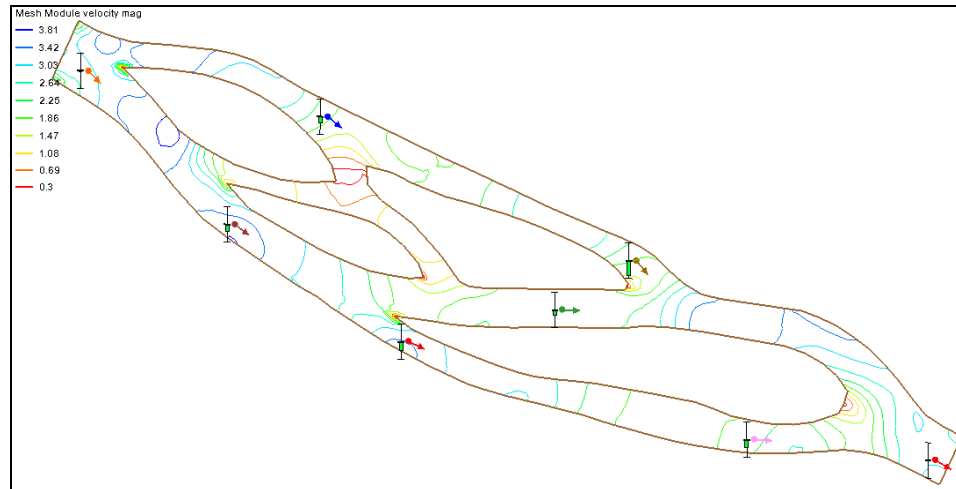
Figure 13   Calibration targets for the imported observe3.h5

Now make the "WSE" measurement the active measurement by doing the following:

3. Switch to the ✚ᴺ **Map** module.

4. Select *Feature Objects* | **Attributes…** to bring up the *Observation Coverage* dialog.

5. In the *Measurements* section, check the box in the *Active* column on the "WSE" row to make it the active measurement.

6. Click **OK** to close the *Observation Coverage* dialog.

All but two points are within the acceptable interval. To get these two points within range, a final model run was completed using an eddy viscosity of 7.0. To view this set of solutions:

7. Repeat steps 1–3 in section 9.1, opening "observe4.h5" in step 2.

8. Select the "⬀ velocity" and "▦ velocity mag" datasets under the "🗀 observe4.flo" solution folder under "⬟ mesh (3)" to make them active.

The calibration targets should show all the points are within an acceptable range (Figure 14).

The calibration process is now complete. It will not always be possible to get all observation points for each measurement to be within the acceptable confidence interval. Therefore, it will have to be decided which measurements and which points are the most important to have within the acceptable range.
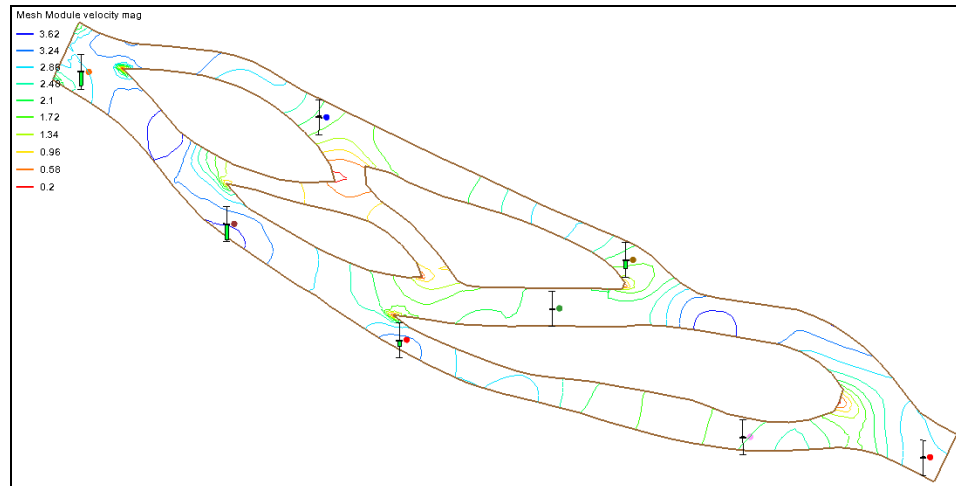
Figure 14   Calibration targets for the imported observe4.h5

10   Using the Error vs. Simulation Plot

When performing trial-and-error verification, it is often important to keep track of the error trend as new solutions are repeatedly computed. SMS provides a special verification plot to simplify this task. To create this plot:

1. Select *Display* / **Plot Wizard…** to bring up the *Step 1 of 2* page of the *Plot Wizard* dialog.

2. In the *Plot Type* section, select "Error vs. Simulation" from the list on the left.

3. Click **Next** to go to the *Step 2 of 2* page of the *Plot Wizard* dialog.

4. In the top section, select "Velocity" from the *Measurement* drop-down.

SMS will create a plot with one point for each simulation. The order of the points in the plot follows the order of the solution sets in the *Solutions* in the lower section. The solution at the top will be listed first in the plot. The default order is the order in which they were imported.

5. Use the **Move Up** and **Move Down** buttons to change the order of the solutions, if desired.

6. Click **Finish** to close the *Plot Wizard* dialog, generate the plot, and open the *Plot 3* dialog.

A new plot appears showing the *Error vs. Simulation*, as shown in Figure 15. Notice for the "Velocity" measurement that the errors decrease as each simulation was performed until the final solution where the errors slightly increase. This slight increase in error with the "Velocity" measurement was required to get that last observation point for the "WSE" measurement within the acceptable range. Generally, if the errors increase, then the model is not improving.
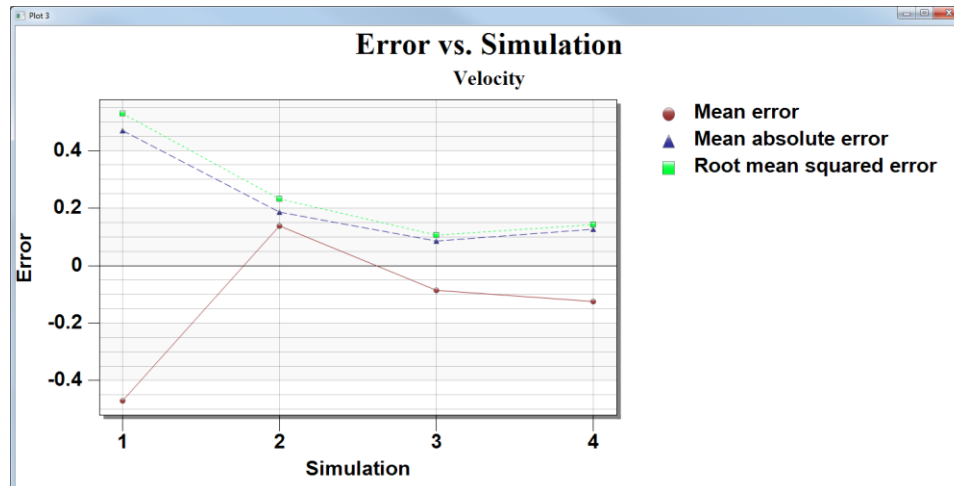
Figure 15   Error vs. Simulation plot for velocity

The plot can be changed to show changes in the "WSE" measurement by doing the following:

7. Right-click the *Error vs. Simulation* plot and select **Plot Data…** to bring up the *Data Options* dialog.

8. In the top section, select "WSE" from the *Measurement* drop-down.

9. Click **OK** to close the *Data Options* dialog.

The errors for this measurement changed dramatically from solution to solution since parameters were first changed to calibrate points for the "Velocity" measurement. However, the general trend was a decrease in error (Figure 16).

10. When done reviewing the plot, close the *Plot 3* dialog by clicking the [X] at the upper right corner of the dialog.
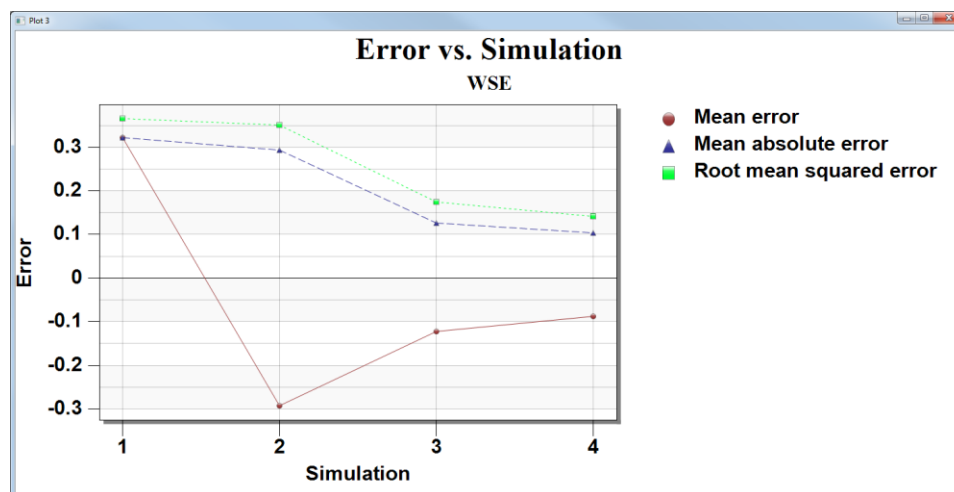

Figure 16   Error vs. Simulation plot for WSE

11   Generating Observation Profile Plots

Observation profile plots are used to view dataset values along observation arcs. The first observation arc to be created will be used to create a profile of the main channel. To create this arc:

1. Right-click on " Map Data" in the Project Explorer and select **New Coverage** to bring up the *New Coverage* dialog

2. In the *Coverage Type* section, select *Generic | **Observation*** and enter "Profiles" as the *Coverage Name*.

3. Click **OK** to close the *New Coverage* dialog.

4. Click on this new coverage to make the Map module active.

It's useful to create a separate coverage to keep the observation arcs separate from the existing observation points. When an observation arc is being created, observation points may be clicked joining them to the arc. Observation points and arcs can exist on the same coverage.

5. Using the **Create Feature Arc** tool, create an arc down the main channel, double-clicking to end the arc (Figure 17).
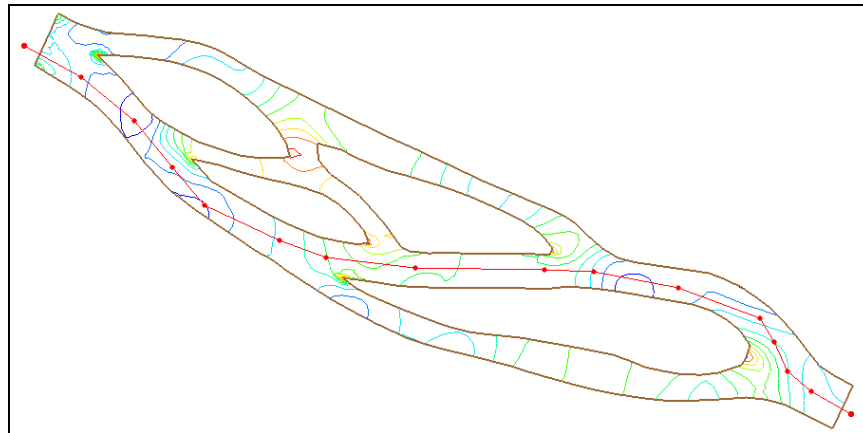

Figure 17   Observation arc, created from top left to bottom right

When the plots are drawn, they will use the name and color associated with the river profile observation arc. To change the name and color of the arc:

6. Using the **Select Feature Arc** tool, double-click on the river profile arc to bring up the *Observation Coverage* dialog.

7. In the spreadsheet in the *Observation Arcs* section, enter "river profile" in the *Name* column.

8. Leave the *Color* as the default.

9. Click **OK** to close the *Observation Coverage* dialog.

Three more arcs need to be created, each across a section of the river. These arcs will be used to create cross section plots. It is important when creating the cross section arcs to not click on the river profile arc. Doing so would split both the river profile arc and the cross section arc.

To create these arcs, do the following:

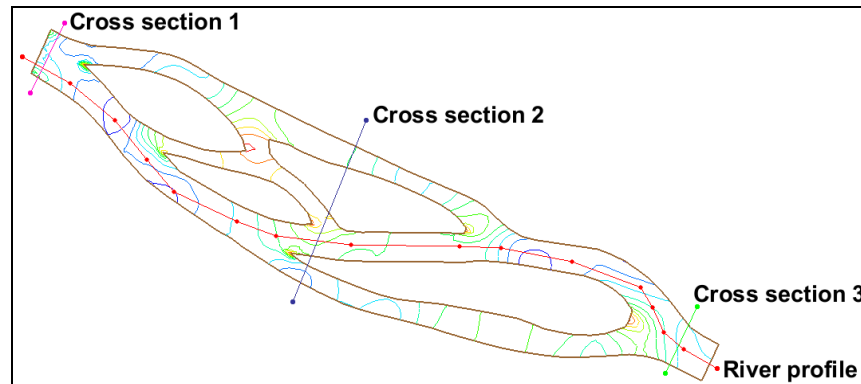10. Using the **Create Feature Arcs** tool, create each of the three cross section arcs shown in Figure 18.

Figure 18   Profile and cross section arcs

11. Using the **Select Feature Arc** tool, double-click any one of the arcs to open the *Observation Coverage* dialog.

12. Notice that there are now four arcs listed in the *Observation Arcs* section.

13. If desired, assign a different color to each arc using the color buttons in the *Color* column in the *Observation Arcs* section, or leave them at the default colors.

14. Enter "cs1" in the *Name* column for the "Cross section 1" arc (Figure 18).

15. Repeat step 13, entering "cs2" and "cs3" (respectively) in the *Name* column for "Cross section 2" and "Cross section 3".

16. Click **OK** to close the *Observation Coverage* dialog.

With the arcs created, the plots can now be generated by doing the following:

17. Select *Display* / **Plot Wizard…** to bring up the *Step 1 of 2* page of the *Plot Wizard* dialog.

18. In the *Plot Type* section, select "Observation Profile" from the list on the left/

19. Click **Next** to go to the *Step 2 of 2* page of the *Plot Wizard* dialog.

20. In the *Coverage* section, turn off *cs1*, *cs2*, and *cs3* in the *Show* column of the spreadsheet.

21. In the *Dataset(s)* section, select *Specified*.

22. Under "mesh (3)", turn on "Z".

23. Under the "observe4.flo" solution, turn on "water surface".

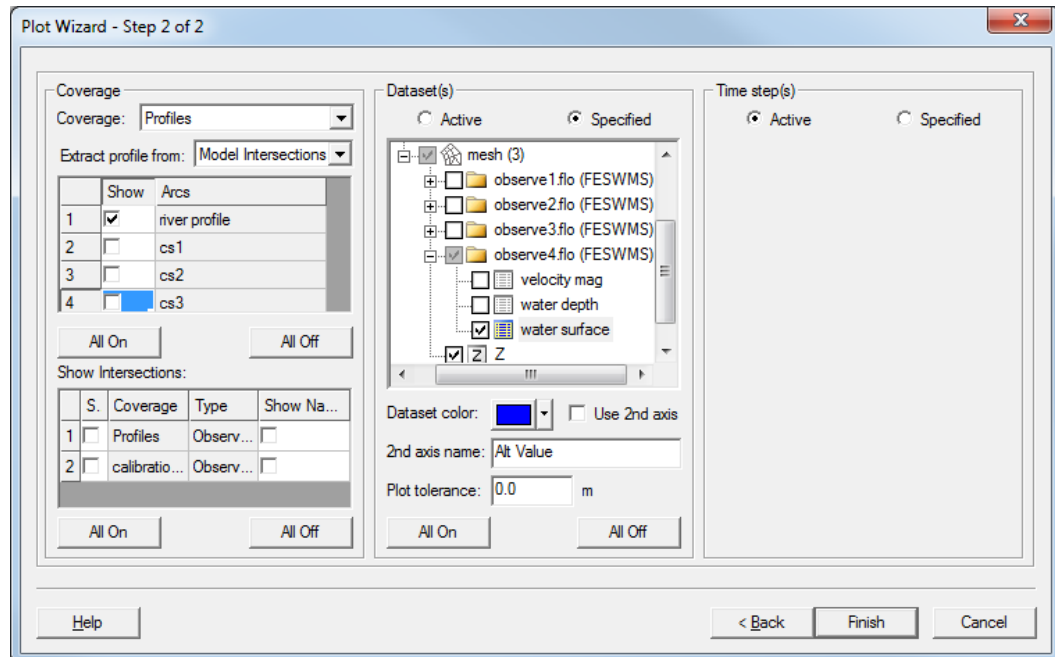The dialog should appear similar to Figure 19.

Figure 19   Settings for generating the plot

24. Click **Finish** to close the *Plot Wizard* and generate the plot.

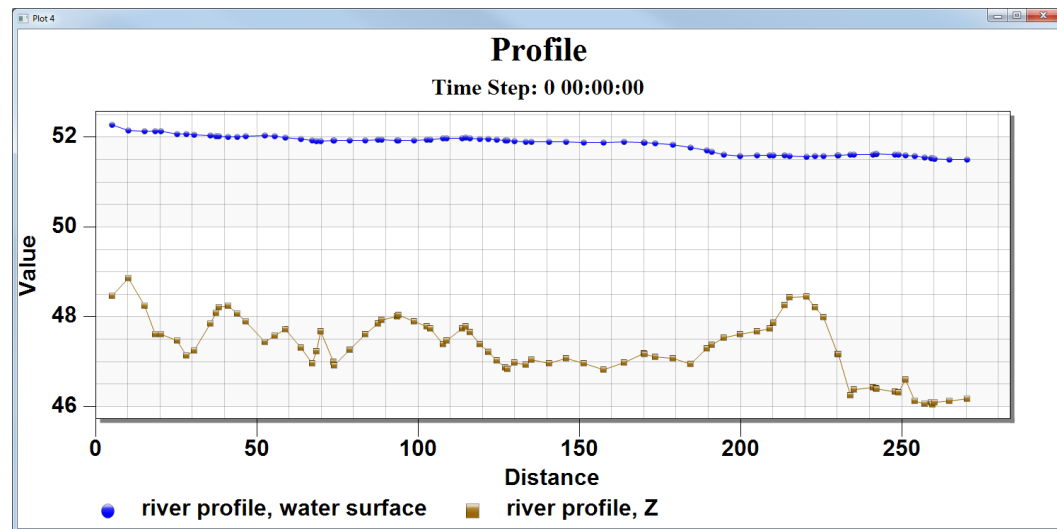The profile plot of the geometry of the stream should appear similar to Figure 20.

Figure 20   Profile plot of the stream geometry

To view the velocity distribution across the three cross sections:

25. Right-click on the profile plot and select **Plot Data…** to bring up the *Data Options* dialog.

26. In the *Coverage* section, turn off *river profile* and turn on *cs1*, *cs2*, and *cs3* in the *Show* column of the spreadsheet.

27. In the *Dataset(s)* section, under the "observe4.flo" solution under "mesh (3)", turn on "water surface".

---

28. Click **Ok** to close the *Data Options* dialog.

When done reviewing the cross section profile plot, click on the [×] at the top right corner of the *Plot 4* dialog to close it.

12   Generating Time Series Plots

As mentioned earlier, observation arcs are used to compute fluxes. One flux value that is often observed and measured in the field is flow rate. Observed flow rates can be used in model calibration in the same way observed velocities and water surface elevations are used.

In addition to normal model calibration, time series plots can be generated showing how the flow rate flux changes with time. This type of time series plot is commonly known as a hydrograph. Hydrographs created using calculated data from the model are useful to see if the model properly predicts flow rate patterns.

To create a time series plot:

1. Select *File* | **Open…** to bring up the *Open* dialog.

2. Select "noyo1.sms" and click **Open** to import the project and exit the *Open* dialog.

3. If asked to delete existing data, click **Yes**.
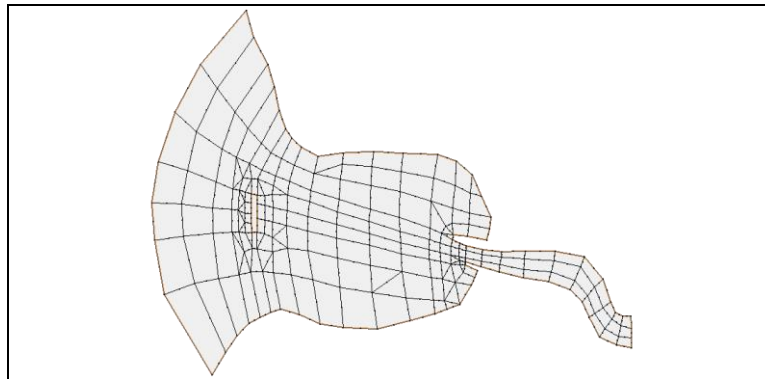
The project should appear similar to Figure 21.

Figure 21   Initial "noyo1.sms" project

Now create the coverage and adjust display options.

1. Right-click on "🗺 Map Data" and select **New Coverage** to open the *New Coverage* dialog.

2. In the *Coverage Type* section, select *Generic* | **Observation** and enter "Fluxes" as the *Coverage Name*.

3. Click **OK** to create the new coverage and exit the *New Coverage* dialog.

4. **Frame** ⊕ the project.

5. Click **Display Options** 🖥 to bring up the *Display Options* dialog.

6. Select "2D Mesh" from the list on the left.

7. On the *2D Mesh* tab, turn on *Elements* and *Nodes*.

8. Click **OK** to close the *Display Options* dialog.

9. Select "⬙ Fluxes" to make it active.

10. Using the **Create Feature Arc** ⌐ tool, create an observation arc across the mesh (Figure 22).
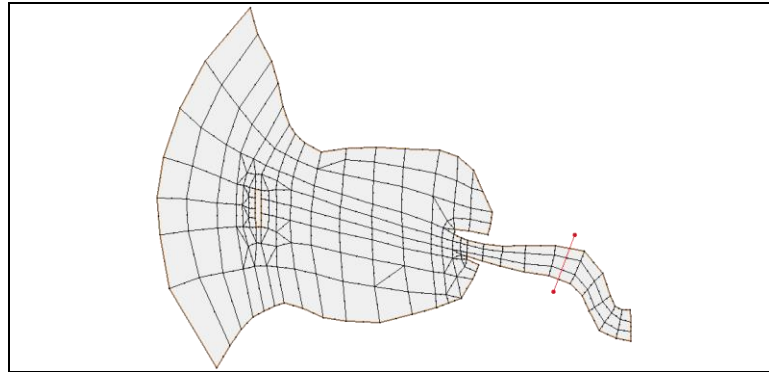

Figure 22   Observation arc across the noyo1 mesh

11. Click **Plot Wizard** ⬙ to bring up the *Step 1 of 2* page of the *Plot Wizard* dialog.

12. In the *Plot Type* section, select "Time Series" from the list on the left and click **Next** to go to the *Step 2 of 2* page of the *Plot Wizard* dialog.

13. Select "Flux" from the *Function Type* drop-down.

14. Check the box in the *Show* column in the spreadsheet on the *Arc 1* row.

15. In the *Scalar Dataset* section, select "water depth" from the tree list.

16. In the *Vector Dataset* section, select "velocity" from the tree list.

17. Click **Finish** to close the *Plot Wizard* dialog, generate the plot, and open the *Plot 5* dialog.

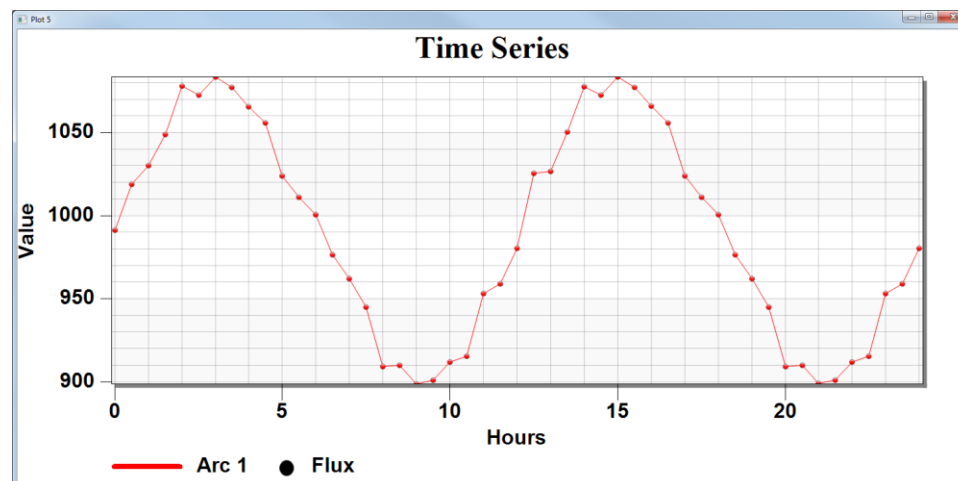A plot of the flow rate flux measurement should appear in the Plot 5 dialog (Figure 23).

Figure 23   Time Series plot of flow rate

---

13   Conclusion

This concludes the "Observations" tutorial. Feel free to continue experimenting with the program, or exit SMS.