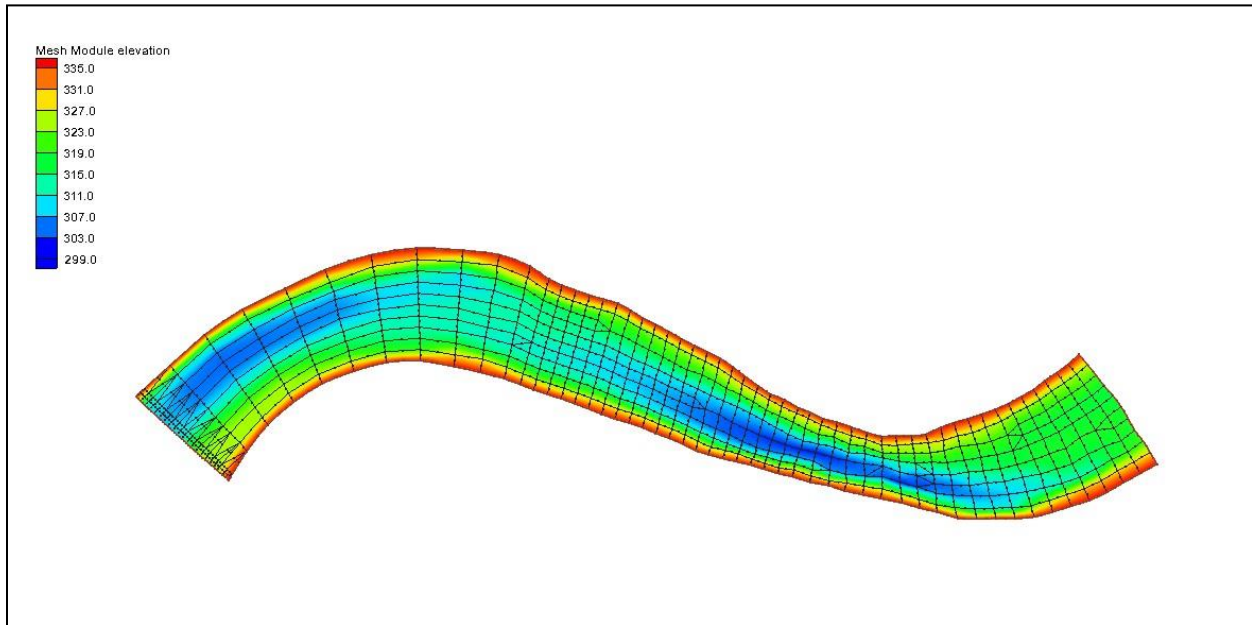*SMS 12.2 Tutorial*
Mesh Editing

Objectives

This tutorial teaches manual mesh generation and editing techniques that can be performed using SMS. It should be noted that manual methods are NOT recommended. Automated mesh generation is preferred both for mesh generation as well as mesh revision to ensure the mesh generation can be replicated and preserved. For minor edits, and for historical support, this tutorial gives a brief introduction to tools in SMS that are useful for editing a mesh.

Prerequisites
- None

Requirements
- Mesh Module

Time
- 20–35 minutes

AQUAVEO™

1   Importing Topographic Data

A mesh can be generated directly from topographic data points. Normally these points are not spaced in a way that would result in a useful mesh, so this type of data is more commonly used to create a scatter set, or "TIN".

However, for illustration of the mesh editing tools, this tutorial will use a preconditioned set of points contained in an XYZ file. This file defines a point on each row of the file with an X, Y, and Z coordinate. A file of this type may also contain one or more header rows defining the contents of the file, source of the data, or identifying the intent of each column. This type of file can be opened by SMS.

To open an XYZ file:

1. First check that the default mesh model has been set to a quadratic mesh model. To do this:

   a. Go to *Edit | **Preferences…*** to bring up the *SMS Preferences* dialog.

   b. Select the *Defaults* tab and check that the *Default 2D Mesh Model* is set to "Generic Mesh". Click **OK** when done. After this tutorial, change the default mesh model if desired using these same steps. It is possible to go through this tutorial using either a quadratic or a linear mesh model.

2. Select *File | **Open***, to bring up the *Open* dialog.

3. Navigate to the *tutorial\SMS_MeshEditing\data files* directory.

4. Select the file "poway1.xyz".

5. Click **Open** to bring up the *File Import Wizard*.

6. In *Step 1* of the *File Import Wizard*, check the *Space* option, then click **Next >**.

7. Change the *SMS data type* to "Mesh", uncheck the *Triangulate data* option, and make sure the column headings for the three data columns show a mapping of "X", "Y" and "Z", respectively, in the *Type* row of the *File preview* section. Click **Finish** to close the *File Import Wizard* and import the data points.

The data points from the file are converted to mesh nodes. From the *File Import Wizard*, any columnar data can be opened into SMS. See the *SMS Online Help* for

more information on the *File Import Wizard*. The data points created from "poway1.xyz" are shown in Figure 1.

If the data points are not visible:

1. Select the *Display* | **Display Options** command to bring up the *Display Options* dialog.

2. Select the "2D Mesh" option from the list on the left, and make sure that *Nodes* is checked on. Click **OK** when done.
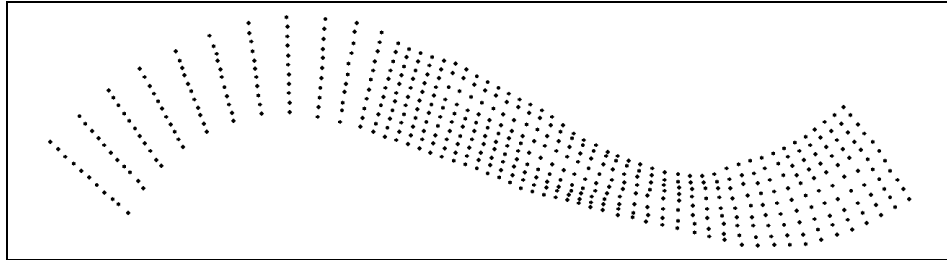

Figure 1   The poway1.xyz data points

2   Triangulating the Nodes

After nodes have been created, elements or cells are required to build a mesh. Elements connect the nodes to define the extents of the flow area. SMS provides numerous automatic mesh generation techniques. This section will review a very simple technique: triangulation.

If the *Triangulate data* option had not been unchecked in the *File Import Wizard*, this step would have been done automatically when the file was imported. The file would then have looked like Figure 2 when it was opened.

To create a triangulated mesh from the data points, do the following:

1. Click on the "Mesh Data" folder to make it active.

2. Select *Elements* / **Triangulate**.

When SMS triangulates data points, it creates either quadratic triangles or linear triangles from the mesh nodes. Different numerical models support different types of elements. RMA2 and FESWMS support quadratic elements, while ADCIRC, SRH-2D, and CGWAVE support only linear elements.

After the nodes are triangulated, the mesh will look like that in Figure 2. It may or may not have midside nodes, depending on whether the elements are linear or quadratic. Quadratic elements usually will have them, and linear elements usually will not.
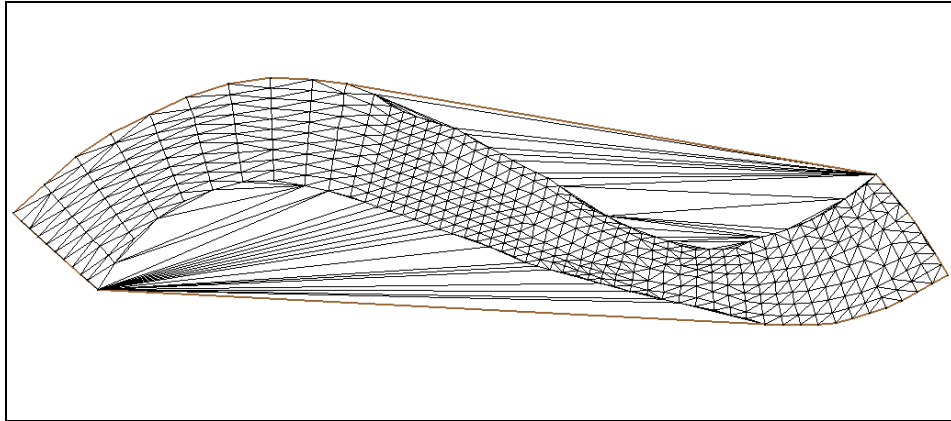
Figure 2   The results of triangulating the "poway1.xyz" data

3   Deleting Outer Elements

The triangulation process always creates elements outside the real mesh boundaries. For this tutorial, the mesh should be in the shape of a rotated *S*, so any elements outside of this boundary must be deleted. Do as follows to remove these elements:

1. Select the **Select Element** ⬆ tool in the toolbox.

2. Click on an element outside the rotated *S* shape to select it.

3. Select another element by holding the *Shift* key and clicking on it.

4. Select *Edit* | **Delete** or press the *Delete* key to remove the selected elements.

5. Click **Yes** at the prompt to confirm the deletion.

It is tedious to individually select every element that needs to be deleted. SMS provides a method to help select groups of adjacent elements by doing the following:

1. Hold the *Ctrl* key and click and drag a line through some elements to select them. Be careful to only select elements outside the *S* shape.

2. Select *Edit* | **Delete** or press the *Delete* key to remove the selected elements.

3. Click **Yes** at the prompt to confirm the deletion.

4. Finish deleting elements that are outside the boundaries of the *S* shape.

3.1   Deleting Thin Triangles

It is not uncommon for the triangulation process to create very thin triangular elements outside the desired mesh boundary. The three corner nodes of thin triangles are almost collinear and the elements may be too thin to see or select. If these are not deleted, numerical errors in the model solution can result.

SMS provides a way to define what is meant by "thin triangle" using the aspect ratio element. The element "aspect ratio" is the ratio of the element width to its height. Perfect equilateral triangles have an aspect ratio of 1.0 while that of thin triangles is

much less. To define the element aspect ratio, do the following:

1. Select *Elements* | **Options** to bring up the *Element Options* dialog.

2. Set the aspect ratio in the *Select thin triangle aspect ratio* box to "0.1". Triangular elements with an aspect ratio less than this are considered to be thin triangles.

3. Accept the defaults for the other options and c lick **OK** to close the *Element Options* dialog.

The best aspect ratio to use for selecting thin triangles depends on the mesh. For this mesh, the distribution of nodes is rather uniform, so a large aspect ratio will suffice. After this value is set, SMS can check for and select thin triangles. To delete any remaining thin triangles, follow these steps:

4. Select *Elements* | **Select Thin Triangles**. The lower right portion of the Status Bar in the Graphics Window shows how many elements became selected due to this operation, along with the total area of the selected elements. There may be quite a few elements selected.

5. Select *Edit* | **Delete** or press the *Delete* key.

6. Click **Yes** to confirm the deletions.
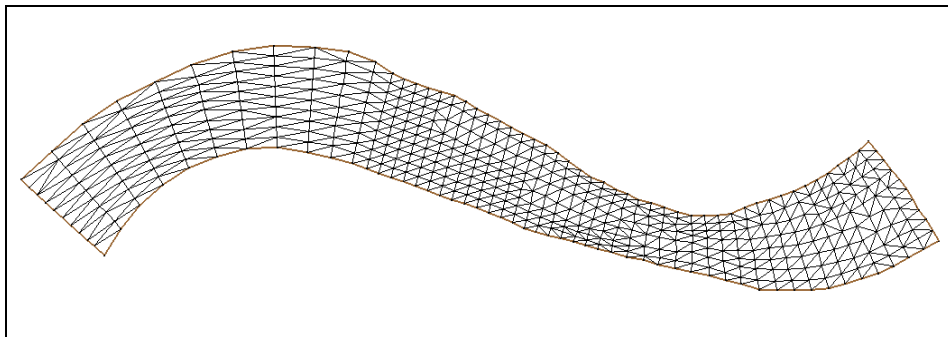
The mesh should now look similar to Figure 3.

Figure 3   The poway1 mesh after deleting excess triangles

4   Merging Triangles

The mesh is composed entirely of triangles. Both ADCIRC and CGWAVE support only triangles. If using one of these models, skip this section of this tutorial.

Using quadrilateral elements can reduce the number of elements required for a simulation and speed up analysis when using RMA2 or FESWMS because of the following:

- A quadrilateral element covers more area than a triangular element.

- A quadrilateral can maintain good interior angles (90 degrees) and still have high resolution in one direction. This makes these elements more numerically stable.

SMS can automatically merge a pair of triangles into a quadrilateral. Before merging

triangles, the *Merge triangles feature angle* should be set. To do this, follow these steps:

1. Select *Elements* / **Options** to bring up the *Element Options* dialog again.

2. Enter a value of "55.0" in the *Merge triangles feature angle* field. (The default value is 65.0.) Two triangles may be merged if all angles of the resulting quadrilateral are greater than the value specified.

3. Leave the other options and click **OK** to close the *Element Options* dialog.

Numeric solvers are more stable and accurate when quadrilateral elements are rectangular and triangular elements are equilateral. Although it is not practical for a mesh to exist entirely of these perfect shapes, the elements should approach these shapes as close as possible. For this reason, SMS merges triangles in an iterative manner.

First, it merges elements using the angle criterion of 90°. Then, the angle criterion is decreased by a number of steps to the feature angle specified. Slowly decreasing the feature angle and testing all triangles against this specified angle will form the best-shaped elements.

SMS can merge the triangles in either a selected portion of elements or all elements. In order to merge triangles in the entire mesh, no elements should be selected. Do the following to merge triangular elements into quadrilateral elements:

1. Select *Elements* / **Merge Triangles**.

2. Since no elements are selected, a prompt will ask if wanting to merge all triangles. Click the **Yes** button at this prompt.

With most meshes, as is the case for this example, not all triangles will be merged. The mesh will appear as in Figure 4 after SMS merges the triangles.
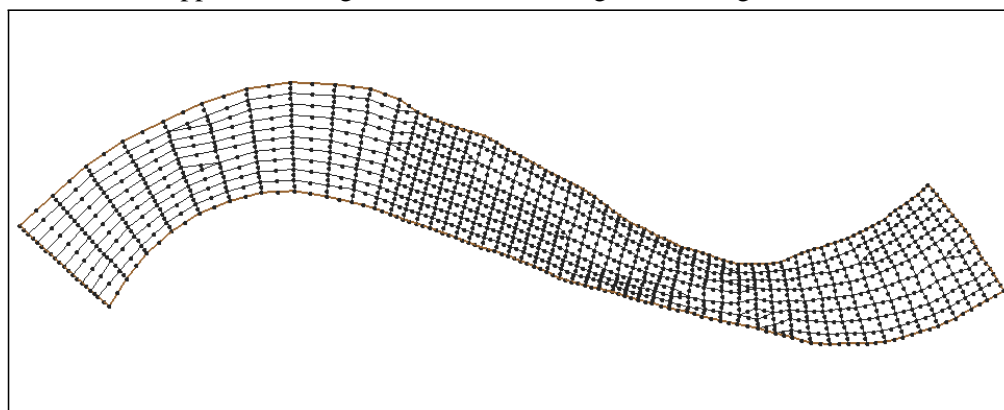

Figure 4   The poway1 mesh after merging triangles

5   Editing Individual Elements

After triangulating the nodes, deleting elements outside the boundaries, and merging triangles, the mesh often needs further manipulation to add model stability. For a main river channel, such as this model, lines of elements should run parallel to the mesh boundary. This is especially important in cases where a portion of the mesh may become dry so that the mesh will dry parallel to the boundary.

Two of the tools in SMS used for manipulating individual elements are the **Split/Merge** 🔧 tool and **Swap Edge** 🔧 tool. With the **Split/Merge** 🔧 tool, two adjacent triangular elements can be merged into a quadrilateral element or a single quadrilateral element can be split into two triangular elements. With the **Swap Edge** 🔧 tool, the common edge of two adjacent triangular elements can be swapped.

See the *SMS Online Help* for a better description of these tools.

5.1   Using the Split/Merge Tool

Most triangular elements in this mesh were merged into quadrilateral elements when the **Merge Triangles** command was performed in section 4. Some of the elements that were not automatically merged can be merged manually. To do this, follow these steps:

1. **Zoom** 🔍 into the portion of the mesh shown in Figure 5a. Notice the two triangular elements separated by a number of quadrilateral elements.

2. Select the **Split/Merge** 🔧 tool from the toolbox.

3. Split the quadrilateral, highlighted in Figure 5a, by clicking inside it. There should now be three triangles, as shown in Figure 5b.

4. Merge the top two triangles, highlighted in Figure 5c, by clicking on the edge between them. (The **Split/Merge** 🔧 tool should still be selected.)

The result of this split/merge operation is shown in Figure 5d. There is now one fewer quadrilateral between the two triangles.
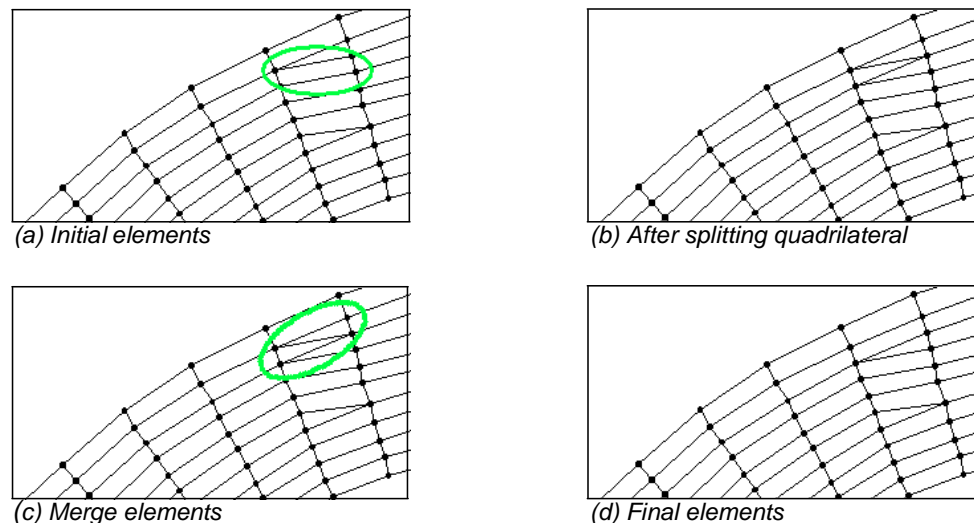

(a) Initial elements

(b) After splitting quadrilateral

(c) Merge elements

(d) Final elements

Figure 5   Example of manual split / merge procedure

To finish editing this section:

5. Repeat the above split/merge process until there are no more triangles across the section. This part of the mesh should look like Figure 6.
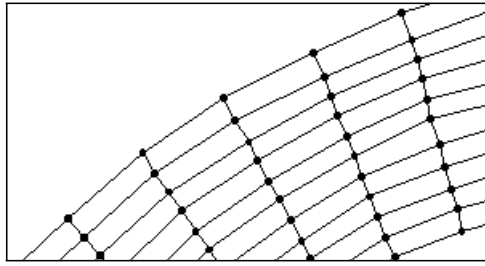
Figure 6   The mesh section after merging triangles

5.2   Using the Swap Edge Tool

The common edge between two triangles can be swapped. The best way to understand this is to think of the two triangles as a quadrilateral, and the common edge between them is a diagonal of the quadrilateral. By swapping this common edge, it changes to be along the opposite diagonal of the quadrilateral. If this edge is clicked again, it returns back to its original state. This can be seen in Figure 7.
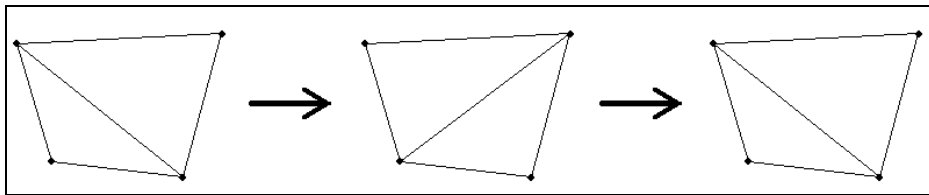


Figure 7   The swap edge technique

One place in this mesh requires the use of the **Swap Edge** tool as well as the **Split/Merge** tool to be able to merge the triangles. This is located toward the middle of the mesh, at the constriction. The easiest way to find this location is to set the window boundaries to the correct location by following these steps:

1. Select *Display* | *View* | **View Options** ... which will bring up the *Display Options* dialog.

2. In the *View* tab, select to use the *Specify width with height dependent on aspect ratio* option (this will disable the *Bottom (min. Y)* field).

3. Enter these values:

   - *Top (maxY)* = "9567"

   - *Left (min X)* = "25164"

   - *Right (max X)* = "25577"

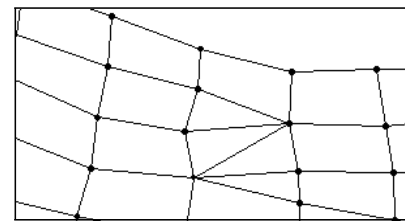4. When done, click **OK** to close the *Display Options* dialog.

The portion of the mesh shown in Figure 8 is now visible. In this part of the mesh, there are two triangles that need to be merged together, separated by a single quadrilateral.

To do this:

1. Choose the **Split/Merge** tool from the toolbox.

2. Click inside the quadrilateral, highlighted in Figure 8a, that separates the two triangles. The quadrilateral gets split as shown in Figure 8b. However, the new edge was not created in the direction necessary to merge the outer triangles.

3. Choose the **Swap Edge** tool from the toolbox. Click only once, directly on the edge that was just created inside the quadrilateral. The edge will swap to the other diagonal of the quadrilateral. This result is shown in Figure 8c.

4. Once again choose the **Split/Merge** tool from the toolbox.

5. Merge the top two triangles to form one quadrilateral by clicking on the line between the triangles.

6. Then merge the bottom two triangles to form another quadrilateral by clicking on the line between the triangles. The result is shown in Figure 8d.
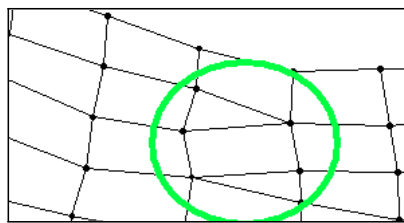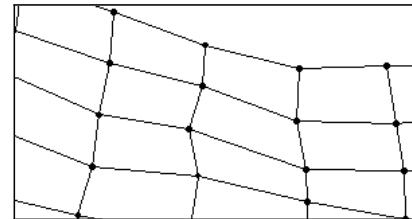

(a) The original elements   *(b) Elements after splitting quad*
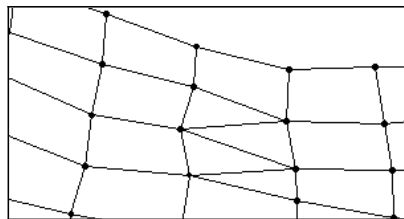
(c) Elements after swapping edge   *(d) Final quadrilateral elements*

Figure 8   Example of manual swapping procedure

Although this operation appears simple, it is one that takes some time to get used to performing. Most people do not get through this without making a mistake. However, after understanding this operation it becomes easier to use. The **Split/Merge** and **Swap Edge** tools are useful for manually adjusting small areas of the mesh.

Since the **Split/Merge** and **Swap Edge** tools are often used together, use the opposite tool that is selected by holding down the *Shift* key.

7. Continue to merge triangles in the where able to do so. Not all of the triangles can be merged. When finished, there should be only six triangles left in the finite element mesh. It should look like the image shown in Figure 9.
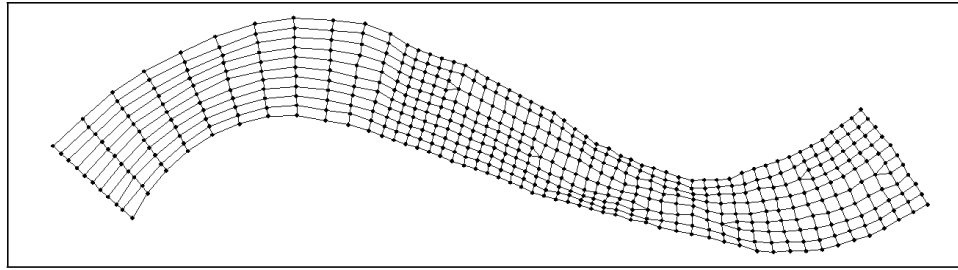
Figure 9   The finite element mesh after merging triangles

5.3   Renumbering the Mesh

Any process which edits a mesh, such as those performed in the previous few sections, causes the node and element ordering to become disorganized. This random mesh ordering decreases the order and efficiency of the matrices required by the analysis codes. Renumbering the mesh restores a good mesh ordering, making it more efficient to run the analysis.

To renumber this mesh, follow these steps:

1.   Choose *Nodes* / **Renumber**.

2.   SMS will bring up a dialog reporting the bandwidth before renumbering and the bandwidth after renumbering. Lower bandwidths make the solution process more efficient and should always be done before running a model. Click **OK** to start the renumbering process.
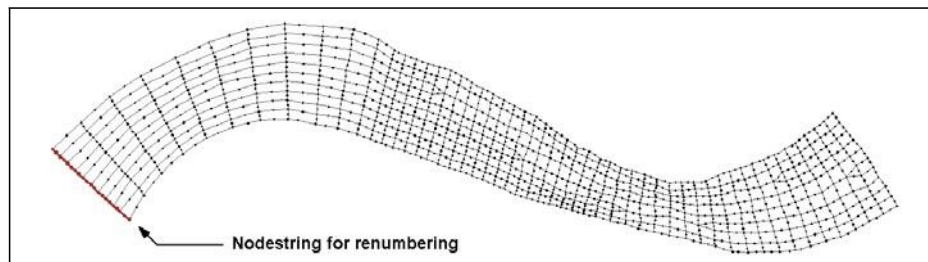
Figure 10   The position of the nodestring for renumbering

When SMS is finished renumbering the mesh, the display will refresh. Remember that adding and deleting nodes or elements changes the mesh order. It is important that renumbering be the last step of the mesh editing process. Editing a mesh invalidates any solution files that have previously been saved. (Solution files are discussed in later tutorials.)

5.4   Changing the Contour Options

When the mesh is created, contour lines are drawn to connect points of equal elevation. By default, these contours are displayed as constant green lines. The contour display can be changed using the contour options in the *Display Options* dialog. It is

always a good idea to look at a color contour map after a new mesh has been created. This helps better visualize the bathymetry of the model.

Do the following to set the color fill contours:

1. Choose *Display* / **Display Options** to bring up the *Display Options* dialog.

2. On the *2D Mesh* tab, turn on the *Contours* option and turn off the *Nodes* option.

3. Select the *Contours* tab then set the *Contour* method as "Color Fill".

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

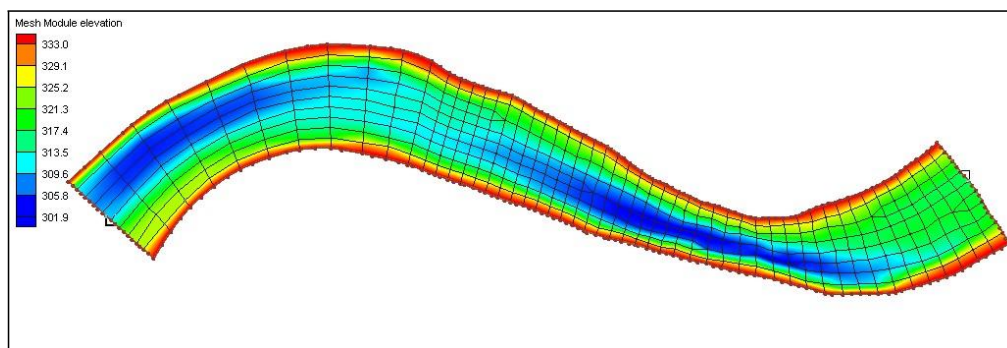The display will refresh with color-filled contours such as those shown in Figure 11.

Figure 11   Elevation contours of the poway1 mesh

In this plot, see that there are two pits in the river, while both banks are the highest part.

If the contours are displaying red in the pits and blue along the banks, reverse the color ramp to match that of Figure 11 by doing the following:

1. Choose *Data* / **Contour Options** to bring up the *Contours* tab of the *Display Options* dialog

2. Select the **Color Ramp** button to bring up the *Color Options* dialog.

3. Click the **Reverse** button at the bottom of the dialog.

4. Click **OK** to close the *Color Options* dialog then **OK** again to exit the *Display Options* dialog.

For more examples of how to work with display and contour options in SMS, see *SMS Online Help*.

5.5   Checking the Mesh Quality

Another important item to check with a newly created mesh is the element quality. SMS looks at various things when checking this. Do as follows to turn on the mesh quality:

1. Select *Display* / **Display Options** to bring up the *Display Options* dialog again.

2. Select the *2D Mesh* tab if it is not already selected.

3. Turn off the *Contours* option then turn on the *Mesh quality* option.

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

The display will refresh without contours and with the mesh quality, as shown in Figure 12. Mesh quality shows where problem areas may occur. A legend shows the color corresponding with each quality item. See the *SMS Online Help* for more information on these mesh quality options.
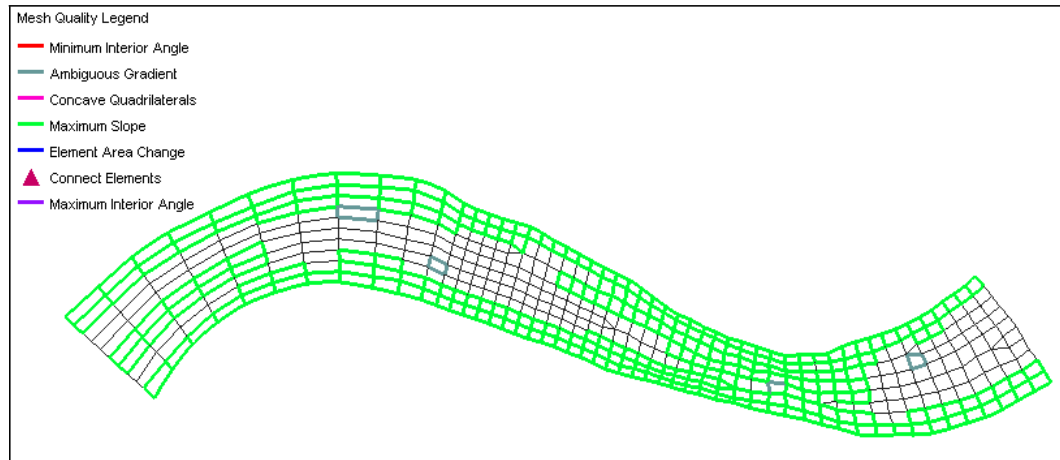

Figure 12   Mesh quality for the "poway1" mesh

Many elements are highlighted because of the maximum slope warning. Elements that are steep in the flow direction may cause supercritical flow to occur. They also could be related to an area where the depth averaged flow assumptions are invalid.

In this mesh, however, the elements are steep in the direction perpendicular to the flow. This is a potential area of numerical instability if drying will take place. If a node dries on an element that spans a large range of elevation, all the flow through that element must be redistributed to other elements. In this case, the entire mesh will be wet, so this warning can be ignored.

To turn off this mesh quality check, do as follows:

1. Select the Display | **Display Options** to bring up the *Display Options* dialog.

2. Select the *2D Mesh* tab if not already selected.

3. Click the **Options** button next to the *Mesh quality* item. This will bring up the *Element Quality Checks* dialog.

4. Turn off the *Maximum slope* option and click **OK** to close the *Element Quality Checks* dialog.

5. Click **OK** to exit the *Display Options* dialog.

Once again, the display will refresh (see Figure 13), but this time, no slope warnings will be shown. There are only ambiguous gradient warnings left for four elements, which are shown in the figure.
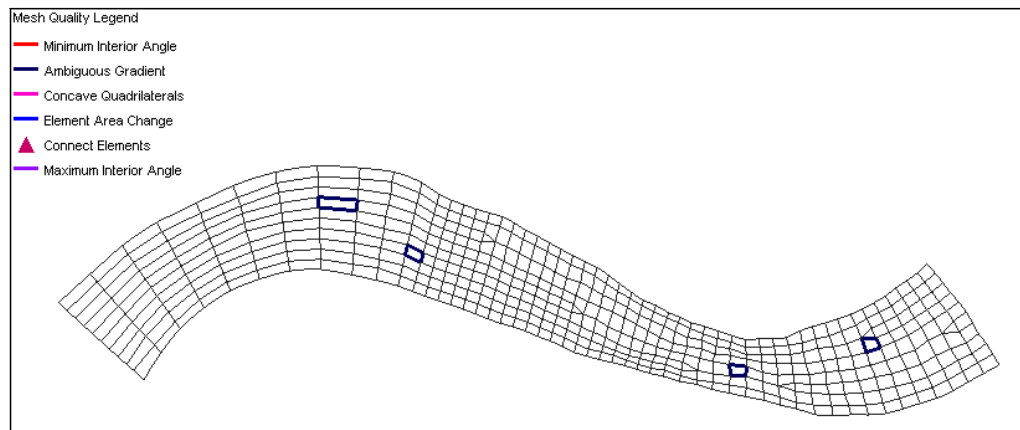
Figure 13   Mesh Quality without the Maximum Slope quality check

If the ambiguous gradient is very small, it can be ignored because the surface is really almost planar. This is the case with the two ambiguous gradients on the left and the rightmost case. Ambiguous gradient cases with a larger variation in elevation, such as the one in the middle of this case, should just be split into two triangular elements using the **Split/Merge** 🖼 tool.

The ambiguous gradients can be examined by selecting the nodes on the corners of the elements using the **Select Mesh Node** ⋅🢔 tool and viewing the elevation of these nodes.

After making these modifications, it is not necessary to worry about the element quality warnings. The following three tasks should be done (in no particular order):

- Turn off the display of mesh quality checks to make the screen less cluttered.

    1. Select the *Display* / **Display Options** command to bring up the *Display Options* dialog.

    2. Go to the *2D Mesh* tab, uncheck the *Mesh quality* option.

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

- Turn on the display of color-filled contours to check that the adjustments just made did not create unusual contours such as a spike or a pit in the mesh. When editing nodal elevation values, it is always important to check the contours. If unusual contours result, look at putting the display back the way it was then make some different changes.

    1. Select the *Display* / **Display Options** command to bring up the *Display Options* dialog.

    2. Go to the *2D Mesh* tab and turn on the *Contours* option.

    3. Click on the *Contours* tab. Under *Contour method*, make sure "Color Fill" is selected.

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

    When finished making changes, if there are any unusual contours, do the

following:

1. Select the *Display* | **Display Options** command to bring up the *Display Options* dialog.

2. In the *2D Mesh* tab, turn off the *Contours* option and turn on the *Nodes* option.

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

Refer to the *SMS Online Help* for additional information on working with contours.

- Renumber the mesh. Remember, whenever the adjusting the mesh (e.g., splitting a quadrilateral into triangles, merging triangles into a quadrilateral, inserting a new node), it should be renumbered. If only modifying elevation values, then it is not necessary to renumber.

1. Choose *Nodes* | **Renumber**.

2. Click **OK** to close the message dialog.

6   Saving the Mesh

If SMS is registered, then the mesh can be saved. This mesh will not be used in other tutorials, so saving it is not required.

If desired, do the following to save the mesh:

1. Select *File* | **Save As** to bring up the *Save As* dialog.

2. Make sure the *Save as type* is set to "Project Files (*.sms)".

3. Enter the *File name* "poway1".

4. Click **Save** to create the project file.

7   Conclusion

This concludes the *SMS Mesh Editing* tutorial. Although not every option was discussed, it is recommended to become familiar with many of the tools that SMS provides for mesh editing. Continue to experiment with the SMS interface or quit the program.