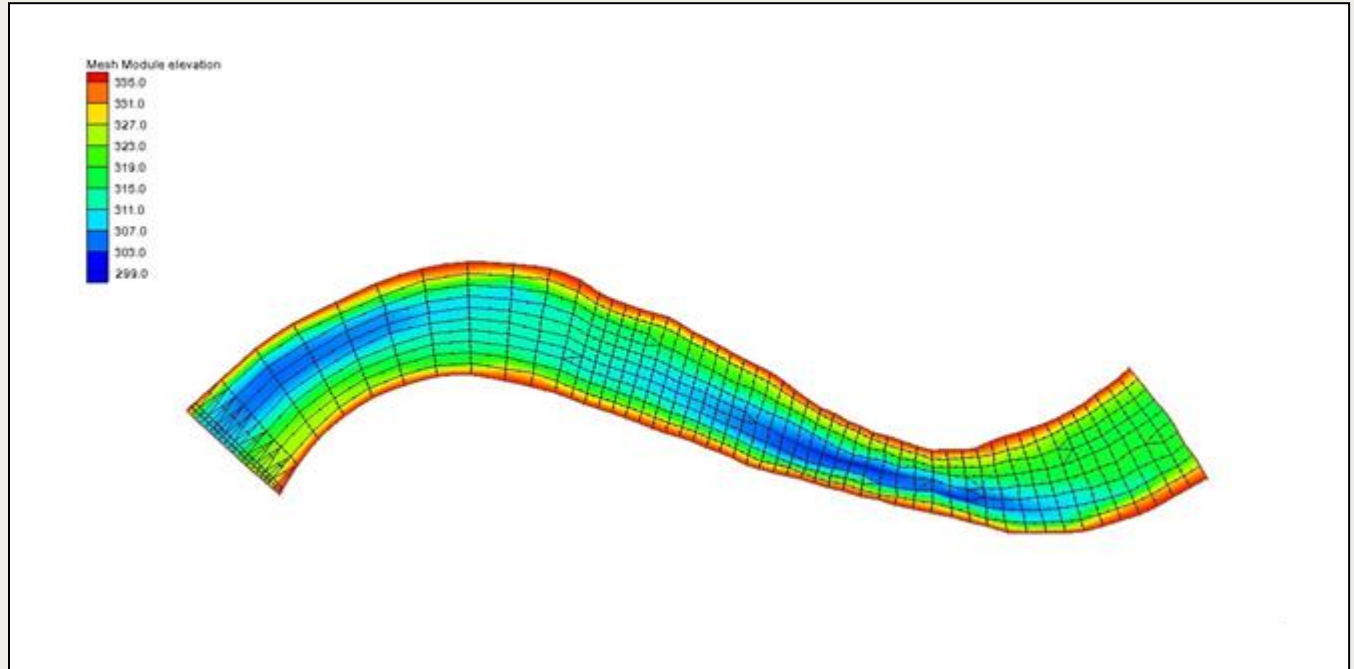*SMS 13.3 Tutorial*

Mesh Editing

Manual mesh editing in SMS

Objectives

Learn manual mesh generation and editing techniques that can be performed using SMS. For minor edits, and for historical support, this tutorial gives a brief introduction to tools in SMS that are useful for editing a mesh.

NOTE: Manually editing a mesh is strongly discouraged. Automated meshing alternatives are preferred.

Prerequisite Tutorials

- Overview
- Map Module
- Mesh Generation

Required Components

- SMS Core

Time

- 20–35 minutes

AQUAVEO™

1   Introduction

This tutorial teaches manual mesh generation and editing techniques that can be performed using SMS.

Note 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.

1. If necessary, start SMS.

2. If SMS is already open, close any existing data by selecting *File* | **New**. A dialog may appear asking to save changes. Click **Don't Save** to clear all data. The graphics window of SMS should refresh to show an empty space.

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

2. Navigate to the SMS_Mesh_Editing\data files directory.

3. Select the file "poway1.xyz".

4. Click **Open** to bring up the *Step 1 of 2* page of the *File Import Wizard*.

5. In *Step 1* of the *File Import Wizard*, turn on the *Space* option, then click **Next >**.

6. Change the *SMS data type* to "Mesh" 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.

7. Click **Finish** to close the *File Import Wizard* and import the data points.

The data points from the file are converted to mesh nodes and triangulated to form triangle elements or cells. 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*.

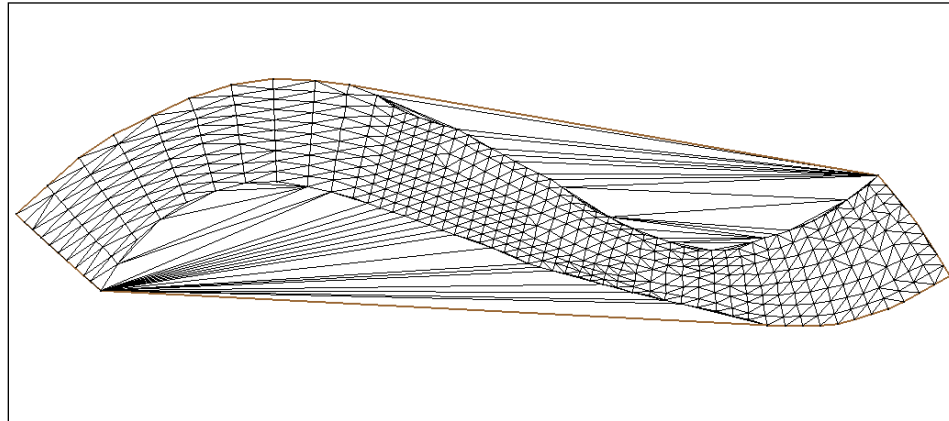If the display options are set to the factory defaults, the new mesh should appear as shown in Figure 1.

Figure 1   The results of triangulating the "poway1.xyz" data

3   Start Mesh Backup

Before manually editing the mesh, it is recommended to turn on the *Mesh Backup* tool. While running, this tool will create of a backup of the mesh so, if necessary, the mesh can be restored to a previous state. To start the *Mesh Backup* tool:

1. Click the **Toolbox** 🧰 macro to open the *Toolbox* dialog.

2. On the *Tools* tab, expand the "📁 Auto-Backup" folder and select the *Mesh Backup* tool.

3. Click **Run Tool** to bring up the *Mesh Backup* dialog.

4. For the *Maximum number of backups* option, enter "100".

This sets the tool to create a backup of the mesh every 20 seconds, only if a change has been made. It will create up to 100 backup meshes which should be more than enough for this tutorial.

5. Click **OK** to run the tool and bring up the *Turn on/off auto mesh backup* dialog.

Leave the *Turn on/off auto mesh backup* dialog open. The tool will detect changes to the mesh and make a backup mesh.

The *Mesh Reload* tool can be used to restore the mesh to a previous state if an error is made.

6. Return to (or reopen) the *Toolbox* dialog.

7. On the *Tools* tab, under the "🗀 Auto-Backup" folder, select the *Mesh Reload* tool.

8. Click **Run Tool** to bring up the *Mesh Reload* dialog.

For now, note that the *Timestamp* option. This option is not populated at this point. This option will list points where the *Mesh Backup* tool saved a change to the mesh.

9. Click **Cancel** to exit the *Mesh Reload* dialog.

4   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:

6. 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.

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

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

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

4.1   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 element aspect ratio. 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 click **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 Boundary 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.
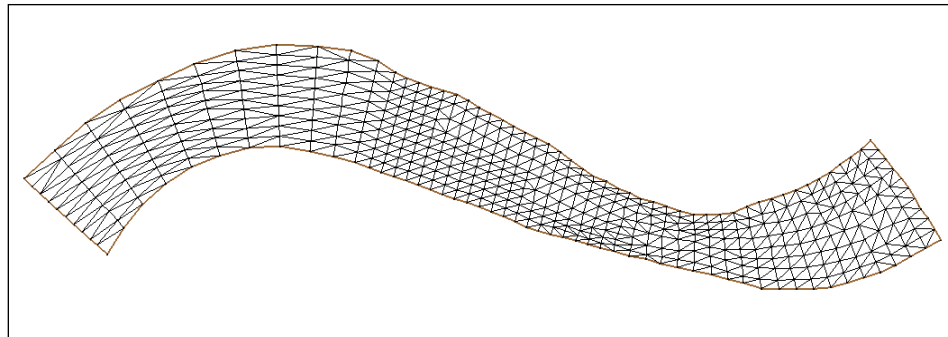
The mesh should now look similar to Figure 2.

Figure 2   The poway1 mesh after deleting excess triangles

5   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 similar models 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 closely as possible. For this reason, SMS merges triangles in an iterative manner.

First, SMS 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 3 after SMS merges the triangles.
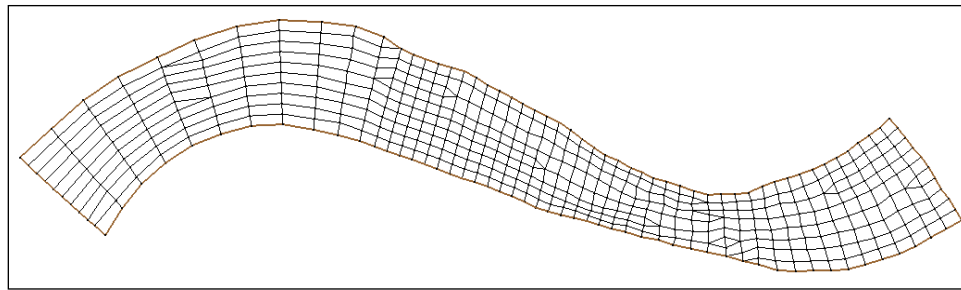

Figure 3   The poway1 mesh after merging triangles

6   Editing Individual Elements

After triangulating the nodes, deleting elements outside the boundaries, and merging triangles, the mesh could benefit from manipulation to increase stability. As noted above, automated methods for mesh generation are recommended and manual manipulation should be kept to a minimum.

For a main river channel, such as in 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.

SMS supports two tools used to manipulate individual elements.

6.1   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 4a. Notice the two triangular elements that are separated by a number of quadrilateral elements.

2. Using the **Split/Merge** tool, split the quadrilateral, highlighted in Figure 4a, by clicking inside it.

There should now be three triangles, as shown in Figure 4b.

3. Merge the top two triangles, highlighted in Figure 4c, 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 4d. There is now one fewer quadrilateral between the two triangles.
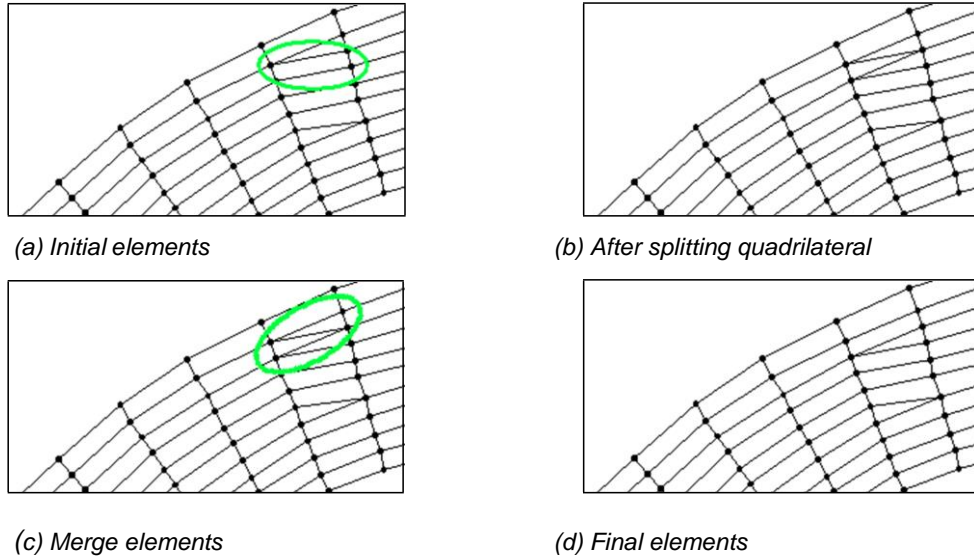
(a) Initial elements

(b) After splitting quadrilateral

(c) Merge elements

(d) Final elements

Figure 4   Example of manual split / merge procedure

To finish editing this section:

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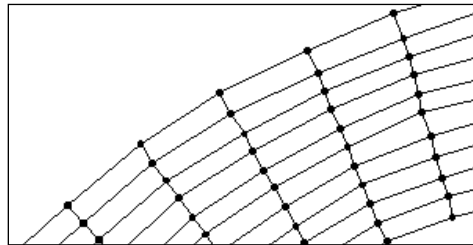
Figure 5   The mesh section after merging triangles

6.2   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 6.
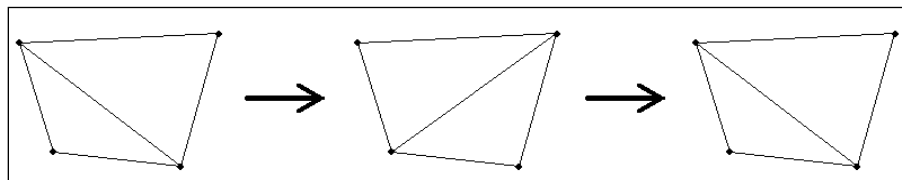
Figure 6   The swap edge technique

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 the *Specify width with height dependent on aspect ratio* option (this will disable the *Bottom (min. Y) field*).

3. Enter these values:

   - *Top (max. Y)* = "9567" (ft)

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

   - Right (max. X) = "25577" (ft)

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

The portion of the mesh shown in Figure 7 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. Using the **Split/Merge** tool, click inside the quadrilateral, highlighted in Figure 7a, that separates the two triangles.

The quadrilateral gets split as shown in Figure 7b. However, the new edge was not created in the direction necessary to merge the outer triangles.

2. Using the **Swap Edges** tool, 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 as shown in Figure 7c.
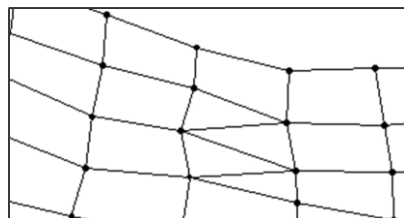
3. Using the **Split/Merge** tool, merge the top two triangles to form one quadrilateral by clicking on the line between the triangles.

4. Merge the bottom two triangles to form another quadrilateral by clicking on the line between the triangles. The result is shown in Figure 7d.
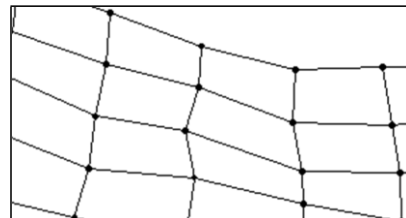

(a) The original elements
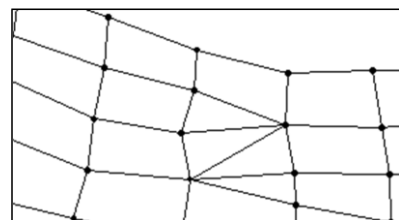
(b) Elements after splitting quad

(c) Elements after swapping edge

(d) Final quadrilateral elements

Figure 7   Example of manual swapping procedure

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

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

5. Continue to merge triangles throughout the domain.

6. **Frame** the mesh.

Not all of the triangles can be merged. When finished, there should be only six triangles left in the finite element mesh. If there is only one triangle in a column of quadrilateral elements, that can be a good indicator that this triangle cannot be merged. The final mesh should look like Figure 8.
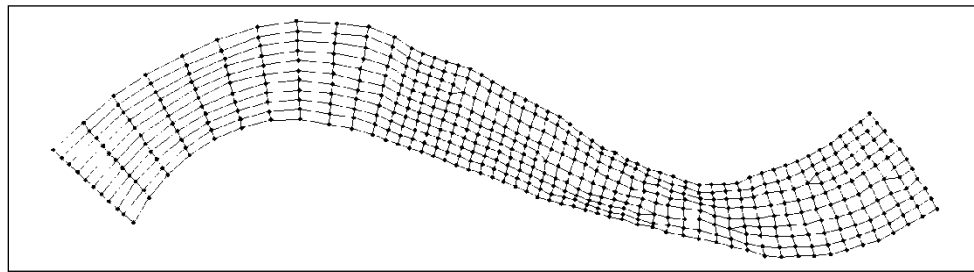

Figure 8   The finite element mesh after merging triangles

6.3   Renumbering the Mesh

Any process that 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** to bring up an *Info* dialog.

The *Info* dialog reports 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.

2. Click **OK** to close the *Info* dialog and start the renumbering process.

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.)

6.4   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 9. 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.
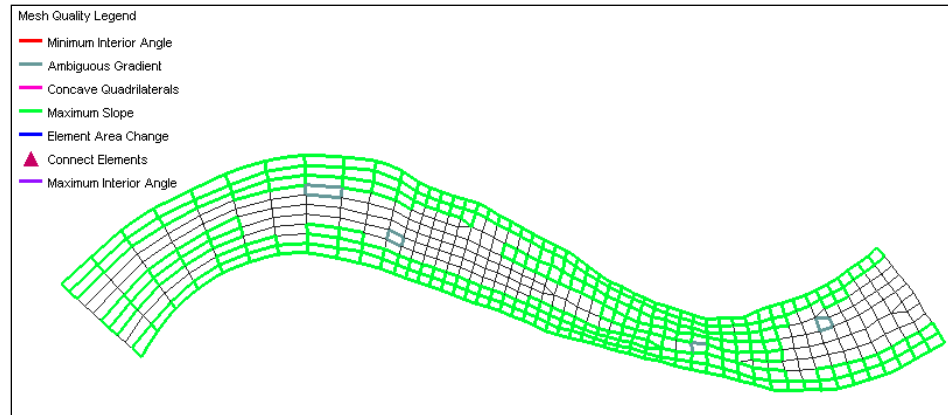

Figure 9   Mesh quality for the "poway1" mesh

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 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 10), but this time no slope warnings will be shown. There are only ambiguous gradient warnings left for four elements, which are shown in Figure 10.

If the ambiguous gradient is very small, it can be ignored because the surface is 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.
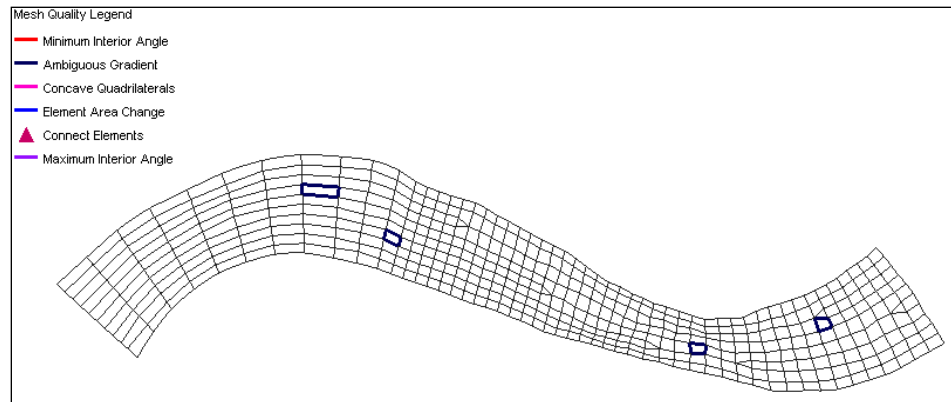

Figure 10   Mesh quality without the maximum slope quality check

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 in the *Edit Window* above the Graphics Window.

After making these modifications, it is not necessary to worry about the element quality warnings. The following tasks should be done:

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 and turn off the *Mesh quality* option.

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

7   Reloading a Previous Mesh

With the *Mesh Backup* tool running in the background, different versions of the mesh during the editing process can be loaded back into SMS by using the *Mesh Reload* tool. This can be used to fix a mistake made during the editing process or to compare the mesh changes. The reloaded mesh with be added to the project as a separate mesh, preserving the changed mesh. To see how this works, complete the following:

1.   Return to the *Turn on/off auto mesh backup* dialog.

2.   Click **Cancel** to close the *Turn on/off auto mesh backup* dialog.

3.   Return to (or reopen) the *Toolbox* dialog.

4.   On the *Tools* tab, under the "📁 Auto-Backup" folder, select the *Mesh Reload* tool.

5.   Click **Run Tool** to bring up the *Mesh Reload* dialog.

6.   Using the *Timestamp* dropdown option, select one of the earlier time stamps.

7.   For the *Reloaded mesh name*, enter "reloaded mesh".

8.   Click **OK** to close the *Mesh Reload* dialog and run the tool.

9.   When the tool finishes, click **OK** to close the *Reload from auto mesh backup* dialog.

A new mesh will appear in the Project Explorer that shows the mesh in an earlier state. If a mistake was made during the editing process, corrections can be made using the reloaded mesh. Comparisons between the earlier mesh and the edited mesh can also be made.

8   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" and click **Save** to create the project file and close the *Save As* dialog.

9   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. Feel free to continue experimenting with the SMS interface.