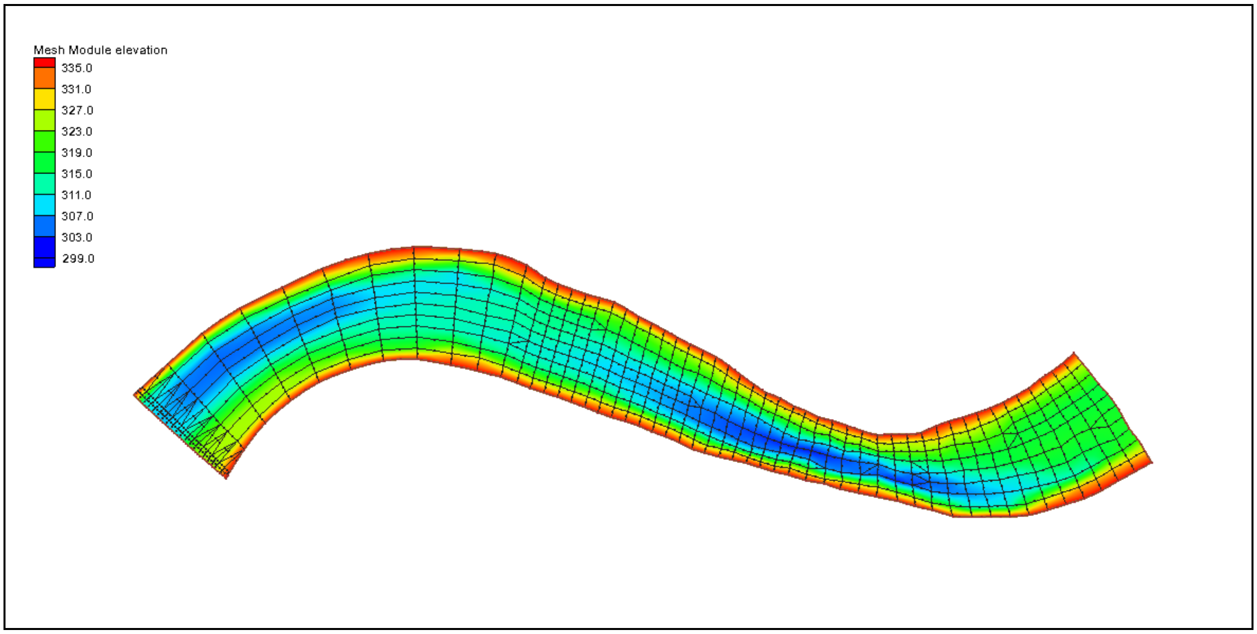
v. 11.1

SMS 11.1 Tutorial
Mesh Editing



Objectives

This tutorial lesson teaches manual finite element mesh generation techniques that can be performed using *SMS*. It gives a brief introduction to tools in *SMS* that are useful for editing a finite element mesh. The mesh in this tutorial will be created by hand from survey points. These mesh editing methods should be used in conjunction with map module meshing to generate a good finite element mesh. This tutorial exists to show useful tools for editing small portions of a mesh after the mesh generation. All files for this tutorial are in the Data File Folder for this tutorial.

Prerequisites

- None

Requirements

- Mesh Module

Time

- 30-45 minutes

Importing Topographic Data

Data points for a finite element mesh can be generated directly from topographic data, such as a list of survey points. An *XYZ* file contains the header XYZ on the first line of the file and then the X, Y, and Z coordinates of each point on a single line in the file. This type of file can be opened by *SMS*. To open the *poway1.xyz* file:

1. Select *File | Open*.
2. Change to the *Data Files Folder* and open the file *poway1.xyz*.
3. The *File Import Wizard* will come up. Click *Next* in *Step 1*.
4. In *Step 2* change the *SMS data type* to *Mesh*, and uncheck the *Triangulate data* option.
5. Make sure the column headings for the three data columns show a mapping of (ie. The *Type*) are X, Y and Z respectively.
6. Click *Finish* to import the data points.

The data points from the file are converted to *mesh nodes*. From the *File Import Wizard*, a user can open any columnar data 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. Make sure that *Nodes* is toggled on in the Display Options.

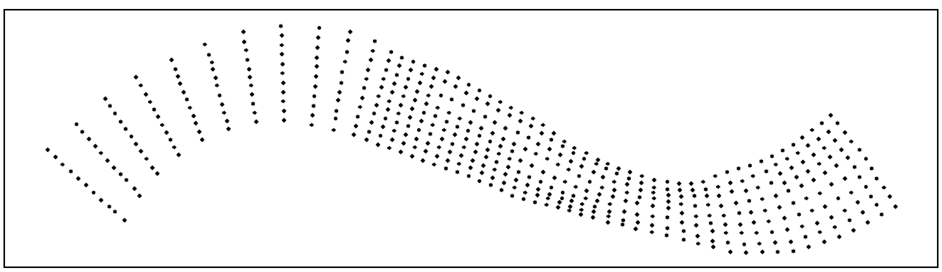


Figure 1 The poway1.xyz data points.

Triangulating the Nodes

After nodes have been created, elements are required to build a finite element 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 above, 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:

1. 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*, *FESWMS* and *RMA10* support quadratic elements, while *HIVEL*, *ADCIRC* 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.

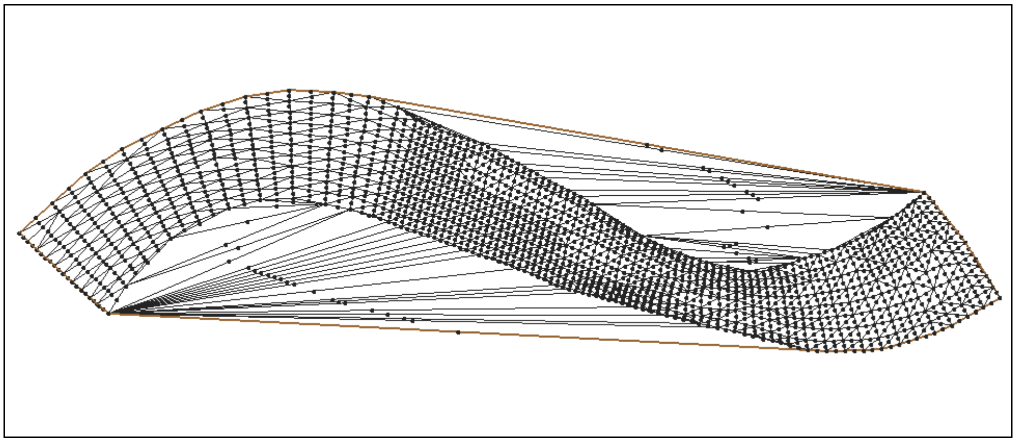


Figure 2. The results of triangulating the poway1.xyz data.

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. To remove these elements:

1. Make sure the *Select Element* tool  in the *Toolbox* is selected and click on an element to select it.
2. Select another element by holding the *SHIFT* key and clicking on it.
3. Select *Edit | Delete* or press the *DELETE* key to remove the selected elements and then click *Yes*.
4. Refresh the display.

It is tedious to individually select every element that needs to be deleted. *SMS* provides a hot key to help selecting groups of adjacent elements. To select a group of adjacent elements:

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.

Select *Edit* | *Delete* or press the *DELETE* key to remove the selected elements and then click *Yes*.

Refresh the display.

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

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 a 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:

1. Select *Elements* | *Options*.
2. Set the aspect ratio in the *Select thin triangle aspect ratio* box to 0.1. (The default value is 0.04.) Triangular elements with an aspect ratio less than this are considered to be thin triangles.
3. Click the *OK* button.

The best aspect ratio to use for selecting thin triangles depends on the finite element 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:

1. 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.
2. Select *Edit* | *Delete* or press the *DELETE* key and then click *Yes*.
3. Refresh the display.

The mesh should now look like Figure 3.

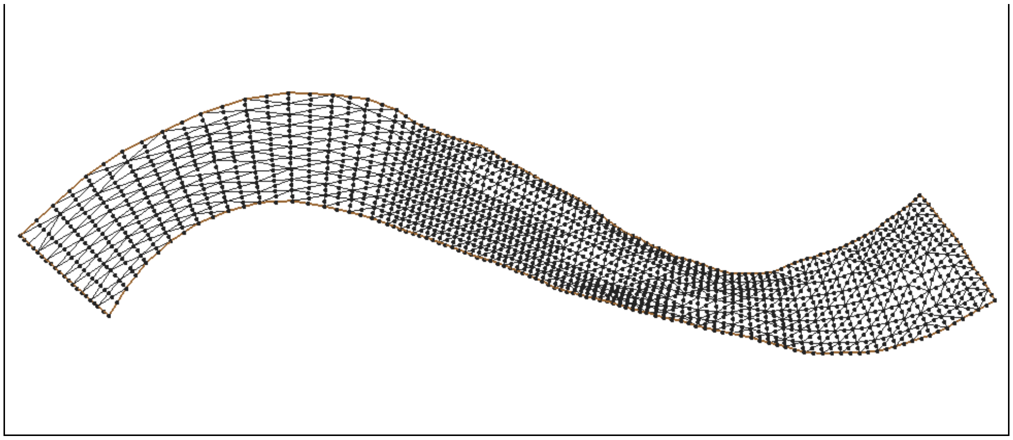


Figure 3 The poway1 mesh after deleting excess triangles.

Merging Triangles

The mesh is composed entirely of triangles. Both ADCIRC and CGWAVE support only triangles. If you are using one of these models, you may 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*, *RMA10*, *FESWMS* or *HIVEL* because:

- 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:

1. Select *Elements* | *Options*.
2. Enter a value of 55.0 in the *Merge triangles feature angle* box. (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. Click the *OK* button.

The finite element method is 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. To merge triangular elements into quadrilateral elements:

1. Select *Elements | Merge Triangles*.
2. Since no elements are selected, you will be prompted 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.

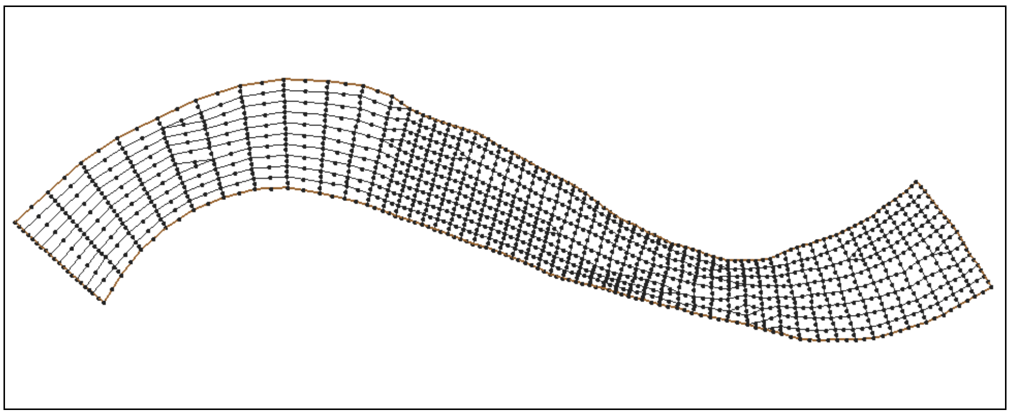


Figure 4. The poway1 mesh after merging triangles.

Editing Individual Elements

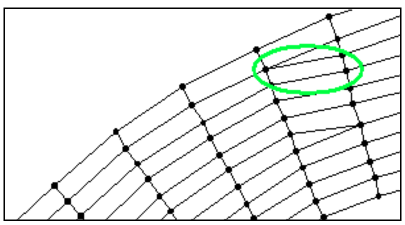
After triangulating the nodes, deleting elements outside the boundaries and merging triangles, the mesh often needs further manipulation to add model stability. For a main river channel such as this model, lines of elements should run parallel to the mesh boundary. This is especially important in cases 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 Help* for a better description of these tools.

Using the Split/Merge Tool

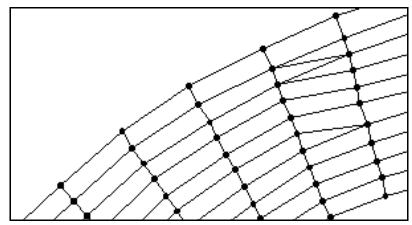
Most triangular elements in this mesh were merged into quadrilateral elements when the *Merge Triangles* command was performed in section 5. Some of the elements that were not automatically merged can be merged manually. To do this:

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.)

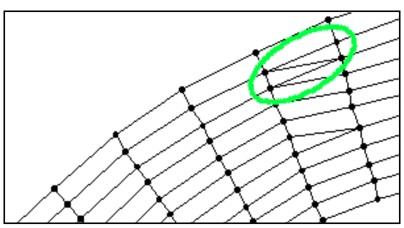
The result of this split/merge operation is shown in Figure 5d. There is now one fewer quadrilateral between the two triangles.



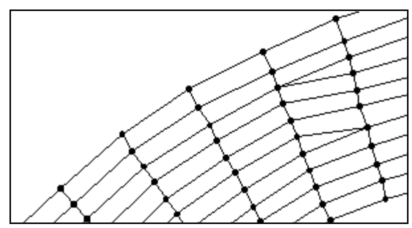
(a). Initial elements.



(b). After splitting quadrilateral.



(c). Merge elements.



(d). Final elements.

Figure 5 Example of manual split / merge procedure.

To finish editing this section:

- 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.

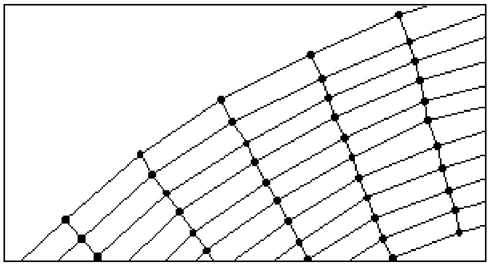


Figure 6. The mesh section after merging triangles.

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.

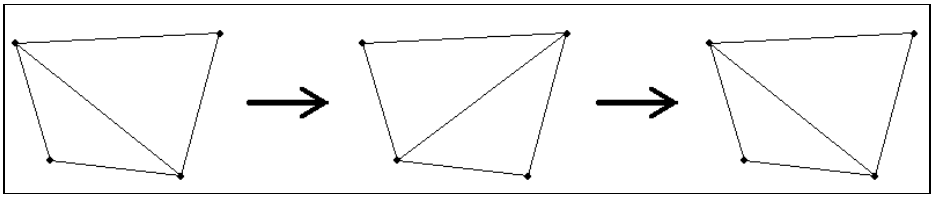


Figure 7 The Swap Edge technique.

One place in this mesh requires the use of the *Swap Edge* tool together with the *Split/Merge* tool to be able to merge the triangles. This is located toward the middle of the finite element mesh, at the constriction. The easiest way to find this location is to set the window boundaries to the correct location. To do this:

1. Select *Display* | *View* | *View Options …* .
2. In the *Display Options* dialog, 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: *Left (min X)* = 25,200; *Right (max X)* = 25,500; *Top (maxY)* = 9560.
4. Press the *OK* button.

You should now be able to see the portion of the finite element mesh shown in Figure 8. In this part of the mesh, there are two triangles that need to be merged together, separated by a single quadrilateral. To do this:

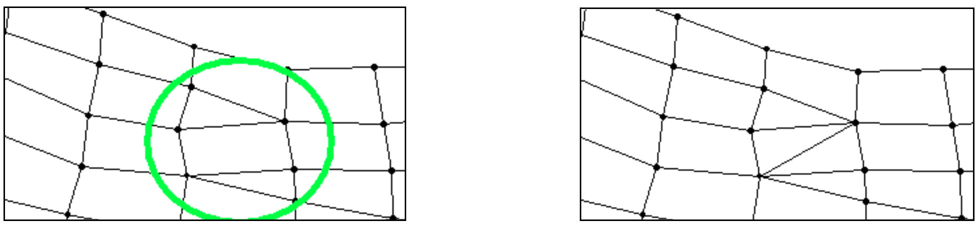
1. Choose the *Split/Merge* tool from the *Toolbox*.

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

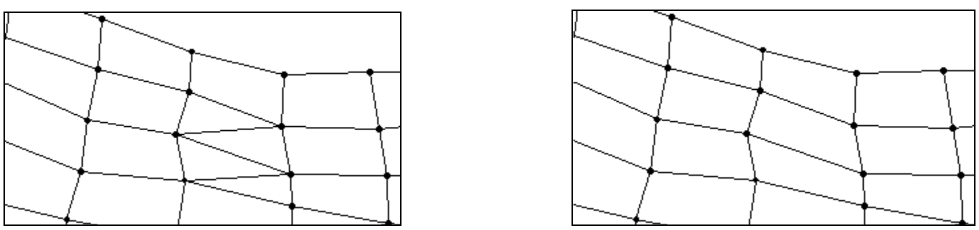
3. Choose the *Swap Edge* tool from the *Toolbox*. Click only once, directly on the edge that was just created inside the quadrilateral. The edge will swap to the other diagonal of the quadrilateral. This result is shown in Figure 8c.

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

5. Merge the top two triangles to form one quadrilateral, and then merge the bottom two triangles to form another quadrilateral. 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 you understand this operation, it is easier to use. The *Split/Merge* and *Swap Edge* tools are very useful for manually adjusting small areas of the finite element mesh.

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

1. Continue to merge triangles in the areas that you are able to do so. Not all of the triangles can be merged. When you are done, there should be only six triangles left in the finite element mesh, and it should look like that shown in Figure 9.

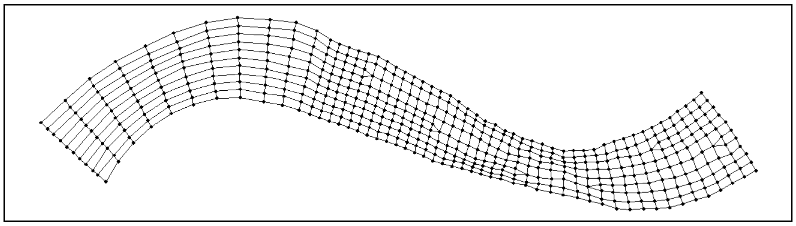


Figure 9 The finite element mesh after merging triangles.

Smoothing the Boundary

When dealing with quadratic finite element meshes, mass loss can occur through a jagged boundary. It is good to smooth the boundary of a quadratic mesh to prevent these losses. Smoothing can only be performed with quadratic models, because the midside nodes get moved while corner nodes do not. *SMS* currently supports three quadratic finite element models, *RMA2*, *RMA10* and *FST2DH.* If you are not using one of these quadratic models, you can skip this section. The quadratic models still support the creation of linear elements. To make sure you have quadratic elements:

1. Select *File | Get Info.*
2. In the top right corner of the *Mesh Info* tab, look at the *Element type*. This will be either *quadratic* or *linear*.
3. Click *OK* to close the dialog.
4. If the element type was *linear*, select *Elements | Linear <-> Quadratic* to switch the element type. If it was *quadratic*, you are already set.

The easiest way to smooth the entire mesh boundary is by creating a nodestring around the entire mesh boundary. To do this:

1. Choose the *Create Nodestring*  tool from the *Toolbox.*
2. Click the node labeled *Node 1* in Figure 10.
3. Hold the *CTRL* key and double-click the node labeled *Node 2* in Figure 10. When holding the *CTRL* key, *SMS* creates a nodestring counter-clockwise around the mesh boundary from the first node to the second node.

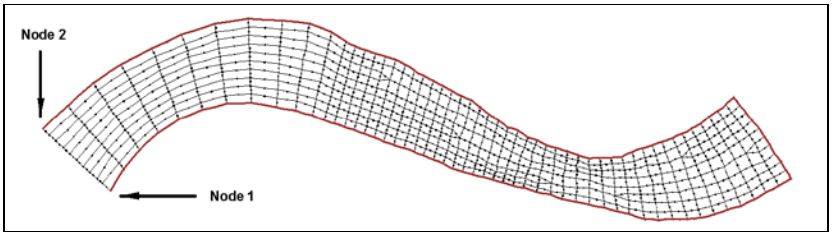


Figure 10 The nodestring to create for smoothing.

This nodestring starts from *Node 1*, and runs counter clockwise around the entire boundary to *Node 2*. Notice that this nodestring goes around two sharp corners on the right side of the mesh. To assure that these corners remain sharp:

1. Select *Elements* | *Options*.
2. In the *Element Options* dialog, change the *Smooth nodestring feature angle* to be 45.0. A midside node will not move if it is at a corner that is sharper than this angle.
3. Click the *OK* button.

Now that the nodestring is created and the feature angle is set, the boundary is ready to be smoothed. To do this:

1. Choose the *Select Nodestring*  tool from the *Toolbox*. A small icon will appear at the center of the nodestring (to the right side of the mesh).
2. Click on the icon to select the nodestring. The icon will be filled and the nodestring will be highlighted in red.
3. Select *Nodestrings* | *Smooth*. The mesh boundary will be smoothed as shown in Figure 11.

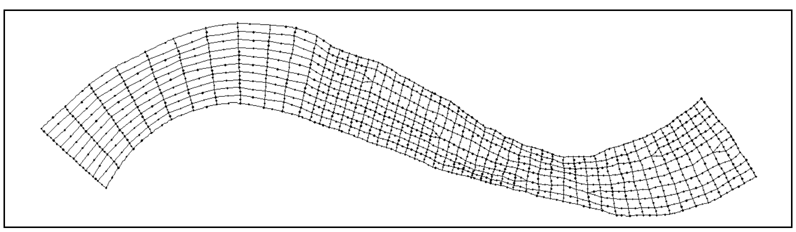


Figure 11. Example of mesh after smoothing.

In general, it is sufficient to smooth the finite element mesh boundary. However, it may be desirable to further smooth interior elements at sharp bends or where dry elements may change the boundary. Any nodestring can be used for smoothing. See the *SMS Help* for more information on creating interior nodestrings and the smoothing operation.

Renumbering the Mesh

The process of creating and editing a finite element mesh, as performed in the previous few sections, causes the node and element ordering to become disorganized. This random mesh ordering increases the size of the matrices required by the finite element analysis codes. Renumbering the mesh restores a good mesh ordering, making it faster to run the analysis. Renumbering starts from a nodestring. To renumber this mesh:

1. Choose the *Create Nodestring* tool from the *Toolbox*.
2. Create a nodestring across the left section, as shown in Figure 12.
3. Choose the *Select Nodestring* tool from the *Toolbox* and click in the selection box of the nodestring that was just created.
4. Choose *Nodestrings | Renumber*. It will not be evident that anything has happened, but the nodes have been numbered from the left to the right. This makes the solution process more efficient and should always be done before running a model.

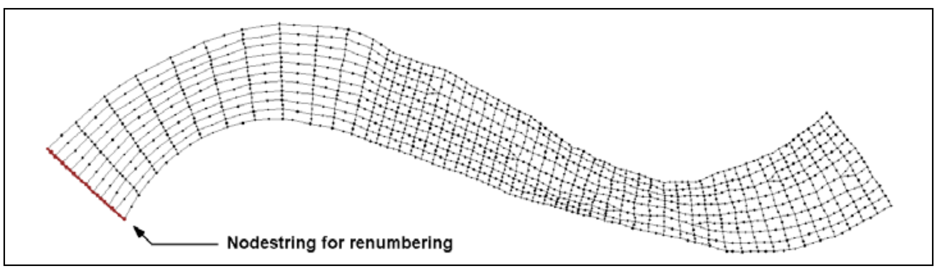


Figure 12 The position of the nodestring for renumbering.

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 creation process. Editing a mesh invalidates any boundary condition and/or solution files that have previously been saved. (Boundary condition and solution files are discussed in later tutorials.)

Changing the Contour Options

When the finite element 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* dialog. It is always a good idea to look at a color contour map after a new finite element mesh has been created. This helps you better visualize the bathymetry of the model. To set the color fill contours:

1. Choose *Data | Contour Options*.

2. In the *Display Options* dialog select *Color Fill* as the *Contour Method*.
3. In the *2D Mesh* tab, turn on the *Contours* and turn off the *Nodes*.
4. Click the *OK* button.

The display will refresh with color filled contours such as those shown in Figure 13.

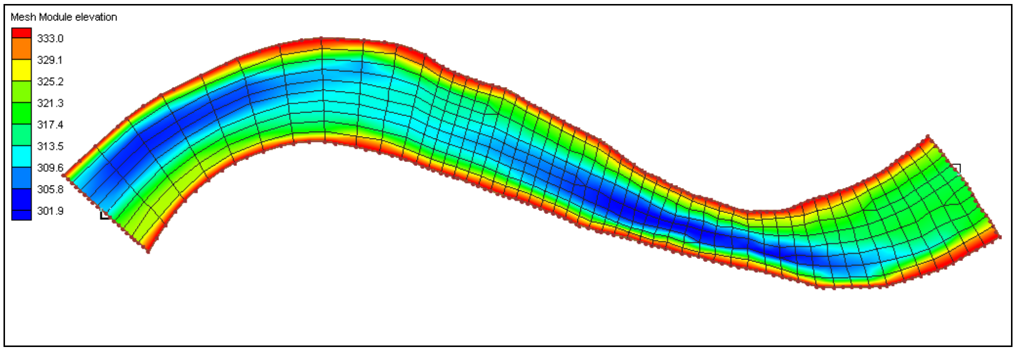


Figure 13 Elevation contours of the poway1 mesh.

In this plot, you can see that there are two pits in the river, while both banks are the highest part. If your contours are displaying red in the pits and blue along the banks, you can reverse the color ramp to match that of Figure 13.

1. Choose *Data* | *Contour Options*.

Select the Color Ramp button and then click the Reverse button at the bottom of the Color Options dialog.

Click OK and then OK again to exit Display Options.

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

Checking the Mesh Quality

Another important thing to check with a newly created finite element mesh is the mesh quality. There are various things that *SMS* looks at when checking this. To turn on the mesh quality:

1. Select *Display* | *Display Options* or right click in the *Project Explorer* to bring up the mesh display options.
2. Select the *2D Mesh* tab if it is not already selected.
3. Turn off the *Contours* and turn on the *Mesh quality*.
4. Click the *OK* button.

The display will refresh without contours and with the mesh quality, as shown in Figure 14. Mesh quality shows where problem areas may occur. A legend shows the color corresponding with each quality item. See the *SMS Help* for more information on these mesh quality options.

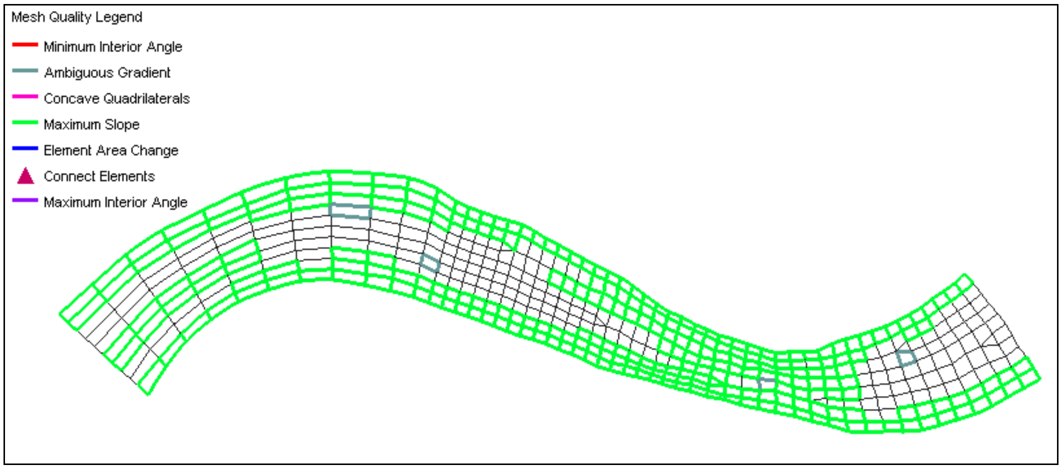


Figure 14 Mesh quality for the Poway1 finite element mesh.

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 on an element that spans a large range of elevation dries, 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:

1. Bring up the display options.
2. Select the *2D Mesh* tab if it is not already selected.
3. Click the *Options* button next to the *Mesh quality* item.
4. In the *Element Quality Checks* dialog, turn off the *Maximum slope* option.
5. Click the *OK* button in both dialogs.

Once again, the display will refresh (see Figure 15, 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

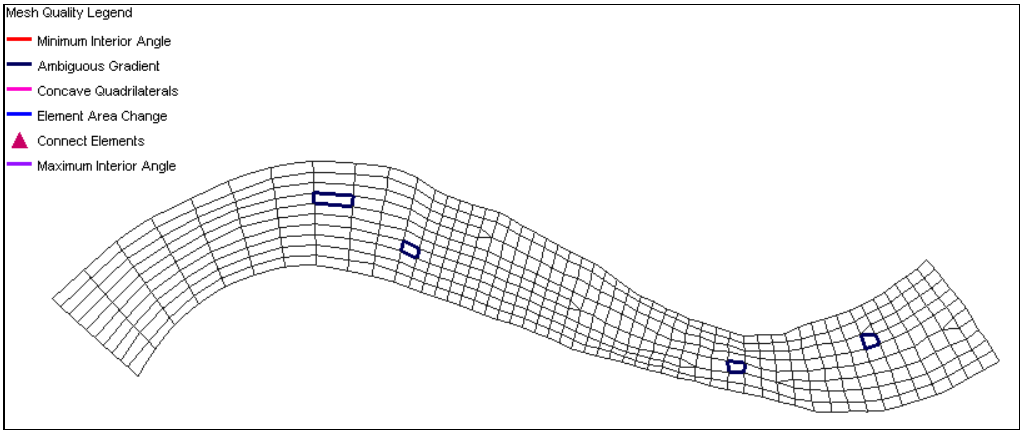


Figure 15 Mesh Quality without the Maximum Slope quality check.

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 and viewing the elevation of these nodes.

After making these modifications, you do not need to worry about the element quality warnings. The following three things should be done (in no particular order):

- *Turn off the display of element quality checks*. You are done looking at the mesh quality, so this should be turned off to make the screen less cluttered.
- *Turn on the display of color filled contours* to check that the adjustments you made did not make funny looking 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 funny looking contours result, you may want to put things back the way they were and make some different changes. When finished, turn the contours back off and turn on the nodes.
- *Renumber the mesh*. Remember, whenever you adjust the finite element mesh (splitting a quadrilateral into triangles), it should be renumbered. If you had only modified elevation values, then you would not need to renumber.

Refining Elements

At times, it is desirous to refine part of a mesh so that there is more definition in that area. More definition helps to increase accuracy and decrease divergence problems. It is important to not refine a mesh too much, however, because more nodes and elements increase the time required for finite element computations. In this section, you will refine the section of elements on the left edge of the mesh.

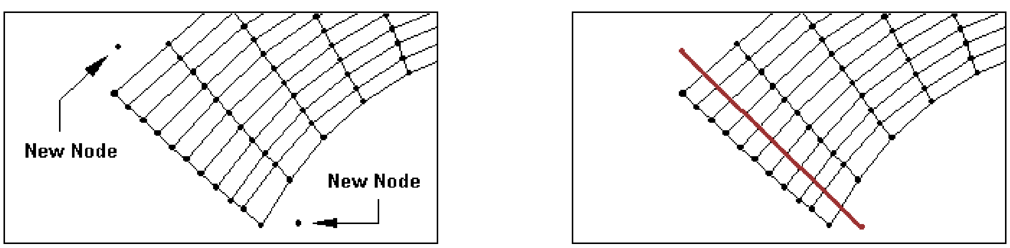
Inserting Breaklines

The elements at the left of the mesh are already rather skinny. The first refinement will be to cut them across their width. This can be done using a nodestring as a breakline. In order to create a nodestring to cut the elements, you must create two nodes, one on either side of the channel. To do this:

1. Zoom into the area shown in Figure 16
2. Choose the *Create Mesh Node* tool from the *Toolbox*.
3. Click once on each side of the channel, near the middle of the left-most column of elements, as shown in Figure 16a. After creating the first node, assign the z-value in the edit window to be 335.0. This will assign the elevation of the new nodes to be similar to the existing nodes.

A nodestring can now be created from one of these new nodes to the other. This nodestring will be used as a breakline. To create the nodestring:

1. Choose the *Create Nodestring* tool from the *Toolbox*.
2. Click on one of the new nodes. Double-click on the other. The nodestring will appear, as shown in Figure 16b.



(a). Two nodes to create. *(b). The nodestring to create.*

Figure 16 Adding the nodestring to use as a breakline.

With the nodestring created, it can be used as a breakline. A breakline splits all the elements that it crosses, forcing element edges to appear along the line. To make a breakline from the nodestring:

1. Choose the *Select Nodestring* tool from the *Toolbox*.
2. Select the icon that appears in the center of the nodestring.
3. Select *Nodestrings* | *Force Breaklines*. The elements will be split along the nodestring, as shown in Figure 17.

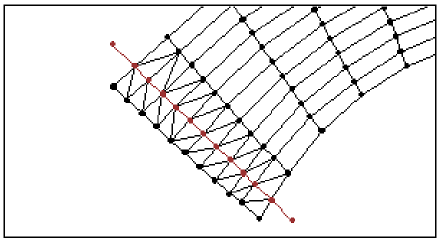


Figure 17 The breakline has been inserted.

Now that the nodestring has been used as a breakline, it is no longer needed. It should still be selected. To remove the nodestring:

- Select *Edit* | *Delete* or click the *Delete* macro.

When the elements get broken along the breakline, triangular elements are created. These should be merged into quadrilateral elements. To do this:

1. Choose the *Select Element* tool from the *Toolbox*.
2. Select *Edit* | *Select With Poly*. This allows you to select a specific set of elements by drawing a polygon around them.
3. Click out a polygon that surrounds all the triangular elements that were created by the breakline. Double-click to end the polygon.
4. With the triangular elements highlighted, select *Elements* | *Merge Triangles*.

All of the triangular elements that were created by the breakline will be merged into quadrilateral elements. With these elements created, you just need to get rid of the two nodes that were created to define the breakline. These nodes are not connected to any elements, and are thus called *disjoint*. To remove the disjoint nodes:

1. Select *Nodes* | *Select Disjoint*. You should get a message that two disjoint nodes were found and selected. Click *OK* to this prompt.
2. Select *Edit* | *Delete* or the delete key and then click *Yes*.

Now that the breakline has been inserted, triangular elements have been merged into quadrilaterals, and the disjoint nodes have been deleted, the mesh should look like that in Figure 18.

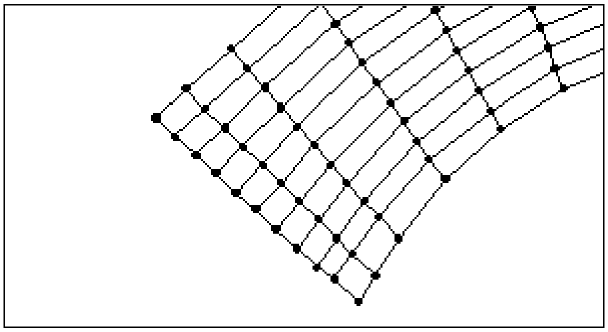
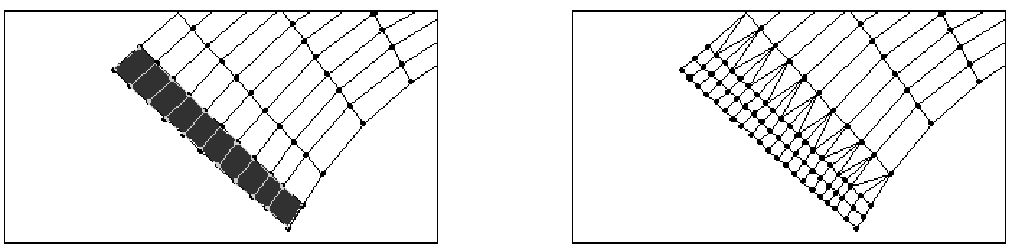


Figure 18 The final mesh after inserting the breakline.

Using the Refine Command

Now that the breakline has been inserted, you are ready to use the refine command. This command splits a quadrilateral element into fourths. The reason the breakline was added is so that the refined elements would not be too skinny. You will refine the first column of elements on the very left side. To refine these elements:

1. Choose the *Select Element* tool from the *Toolbox*.
2. Hold the *CTRL* key and drag a line through the left-most column of elements, as shown in Figure 19a.
3. Select *Elements* | *Refine*. Each of the selected quadrilaterals will be split into four smaller quadrilaterals, and triangles will transition these small quadrilaterals to the larger quadrilaterals, as shown in Figure 19b.



(a). The elements to select. *(b). After the refine command.*

Figure 19 The section of the mesh to refine.

Finishing the Mesh

Now that elements have been created and edited, the following things should be done before using this mesh in a finite element analysis:

- The *Mesh Quality* should be checked. You will see element area change warnings in addition to the same types of warnings as in section 10. See the *SMS help* for a description of why elemental size transitions could present a problem. (If you left the contours on, you may want to turn them off now since the display could become cluttered.)
- The mesh should be renumbered. Remember that whenever nodes and elements are created, the mesh order should be fixed as in section 8

Saving the Mesh

If *SMS* is registered, then the finite element mesh can be saved. This mesh will not be used in other tutorials, so saving it is not required. To save the mesh:

1. Select *File | Save As*.
2. Make sure the *Save as type* is set to *Project Files*.
3. Enter the name *poway1* and click the *Save* button.

Conclusion

This concludes the *Mesh Editing* tutorial. Although not every option was discussed, you should be familiar with many of the tools that *SMS* provides for mesh editing. You may continue to experiment with the *SMS* interface or you may quit the program.