Tutorial: 2D Pipe Junction Using Hexa Meshing

Introduction

In this tutorial, you will generate a mesh for a two-dimensional pipe junction, composed of two inlets and one outlet. After generating an initial mesh, you will check the quality of the mesh and refine it for a Navier-Stokes solution.

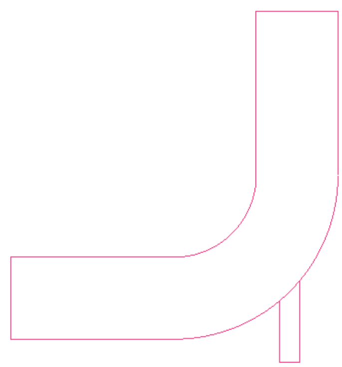


Figure 1: 2D Pipe Geometry

This tutorial demonstrates how to do the following:

- Blocking the geometry.
- Associating to geometry.
- Moving the vertices.
- Applying mesh parameters.
- Generating the mesh.
- Adjusting the edge distribution and refining the mesh.
- Matching the edges.
- Verifying and saving the mesh.

Prerequisites

This tutorial assumes that you are familiar with the menu structure in ANSYS ICEM CFD and that you have read about this functionality. Some of the steps in setup and the procedure will not be shown explicitly.

For details about hexa mesh generation, refer to the Chapter, Hexa, in ANSYS ICEM CFD user manual.

Conventions

Some of the basic conventions used in this tutorial are:

- The icon to the left of the text (here, Blocking) suggests that you have to select the option from the display tree.

  Blocking

- The arrow mark with the text LMB in the box the suggests that you have to click the left-mouse button to enable or disable an option (here, Vertices).

  [LMB] ⟶ Vertices

- The arrow mark with the text RMB in the box the suggests that you have to click the right-mouse button to enable or disable an option (here, Numbers).

  [RMB] ⟶ Numbers

For detailed information about GUI and text conventions, refer to the document, Getting Started with ANSYS ICEM CFD.

Preparation

1. Download the `ICEM_hexa_2dpipe_FILES.zip` file from the ANSYS Customer Portal. It contains the necessary input geometry file (`hexa_2dpipe.tin`).

2. Start ANSYS ICEM CFD and open the geometry (`hexa_2dpipe.tin`).

   File > Geometry > Open Geometry...

Blocking Strategy

Decide the blocking strategy to generate a mesh with blocking.

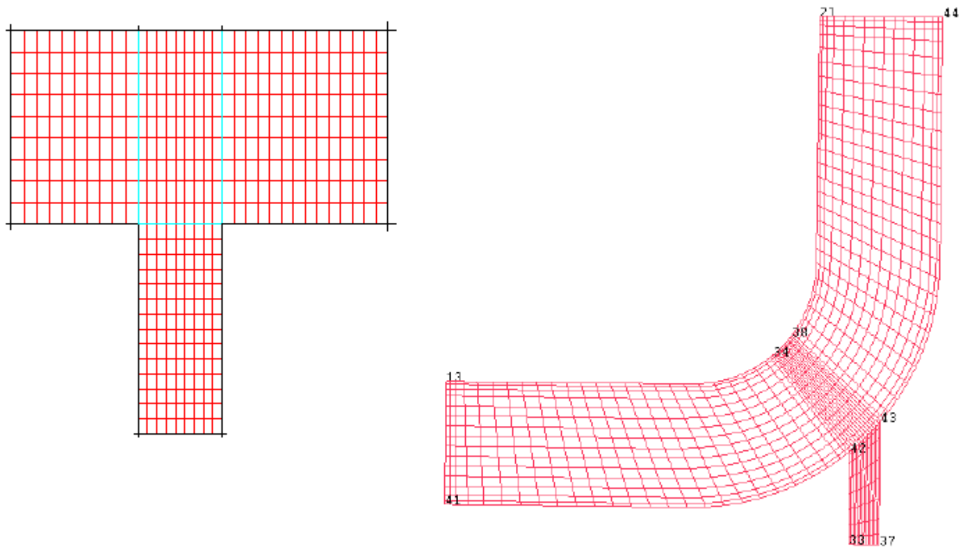


Figure 2: The Mesh and Topologies

Note: *The geometry is equivalent to a T shape. You need to bend the right side of the blocking crossbar upward to resemble the geometry. See Figure 2.*

To fit the T shaped blocking material to the geometry do the following:

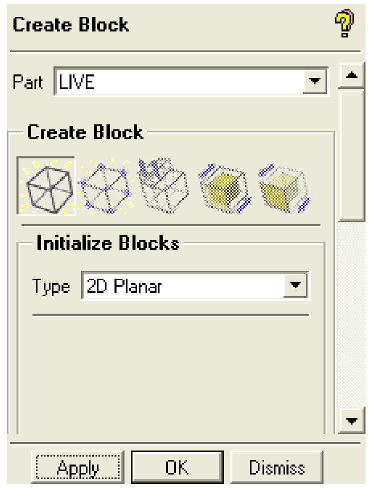
1. Create associations between the Edges of the blocks and the Curves in the geometry.
2. Move the Vertices of the blocks to the corners of the geometry.

Now, the mesh sizes is set and the mesh is computed. The program will automatically project the edge nodes onto the curves of the geometry and the internal 2D volume mesh will be interpolated.

Step 1: Blocking the Geometry

1. Create initial block.
   (a) Initialize the 2D blocking.

   Blocking > Create Block  > Initialize Blocks 

   i. Enter LIVE in the Part field.
   ii. Change the Type to 2D Planar.
   iii. Click Apply.

   (b) Enable Vertices under Blocking.

    Blocking LMB ⟶ Vertices

   (c) Select Numbers under Vertices.

    Blocking RMB ⟶ Vertices LMB ⟶ Numbers

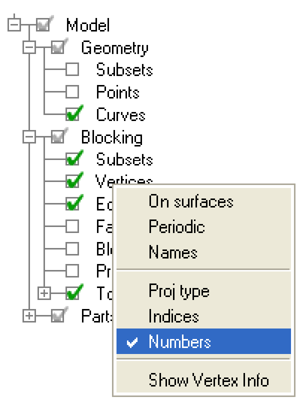


Figure 3: Numbering the Vertices

The white block encloses the geometry as shown in Figure 4. This initial block will be used to create the topology of the model.

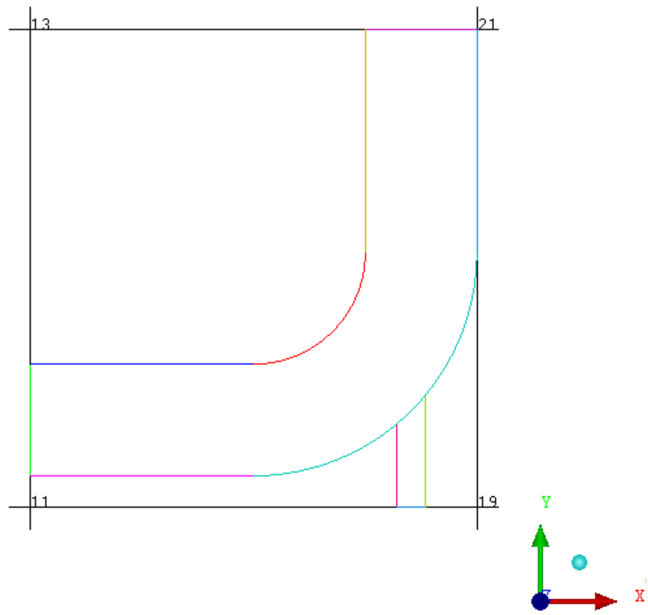


Figure 4: Initial FLUID Block

The curves are now colored separately instead of by part. This is done so that the individual curve entities can be distinguished from each other, which is necessary for some of the blocking operations. You can enable or disable the color coding by doing the following:

i. *Select* Curves *in the* Model *display control tree.*

ii. *Select/deselect* Show Composite.

Geometry RMB ⟶ Curves LMB ⟶ Show Composite

2. Split the initial block into sub-blocks.

Blocking > Split Block > Split Block

In this case, you will first do two vertical splits and one horizontal split.

(a) Create verticle split.

i. Ensure that Curves under Geometry is enabled.

Geometry LMB ⟶ Curves

ii. Retain default selection of Screen select from the Split Method drop-down list in Split Block DEZ.

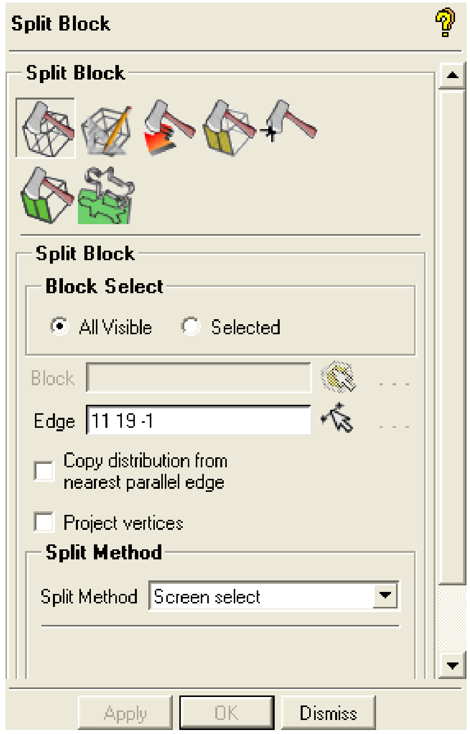


Figure 5: The Split Window

Note: *In this case, the split may be done by approximation because only the topology of the T shape is essential, the exact proportion is not.*

iii. Click  (Select edge(s)).

You will be prompted to select an edge (red text at the bottom of the view screen).

iv. Select the edge defined by vertices 11 and 19 or 13 and 21.

v. To position the new edge, click the left-mouse button, slide the new edge to the desired location and click middle-mouse button.

The split is shown in Figure 6. Check the color of the edge—blue (cyan) designates an internal edge.

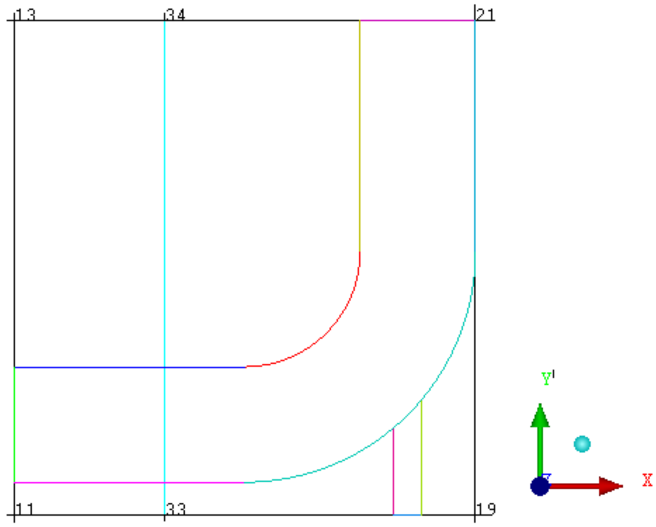


Figure 6: First Split Edge 11-19

Note: *To cancel the previous selection, click the right-mouse button while in selection mode.*

vi. Similarly, select the edge defined by vertices 33 and 19 or 34 and 21. See Figure 7.

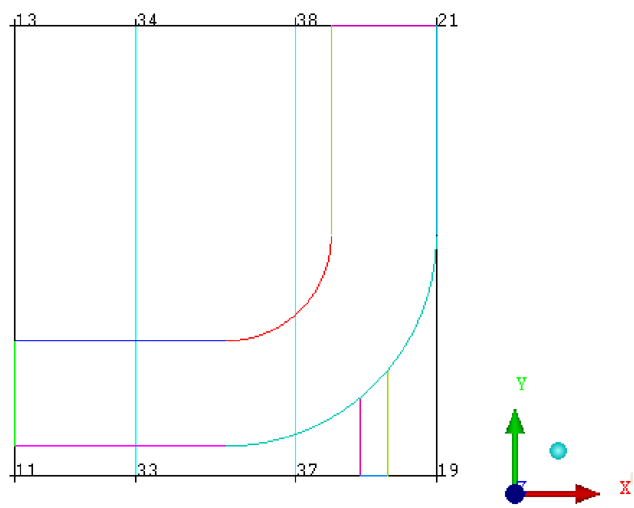


Figure 7: Second Split Edge 33-19

(b) Create horizontal split.

i. Change Split Method to Relative in Split Block DEZ.

ii. Enter `0.5` for the value of Parameter (mid-point of selected edge).

iii. Select any one of the four vertical edges in the graphics window and click Apply.

The horizontal split is shown in Figure 8.

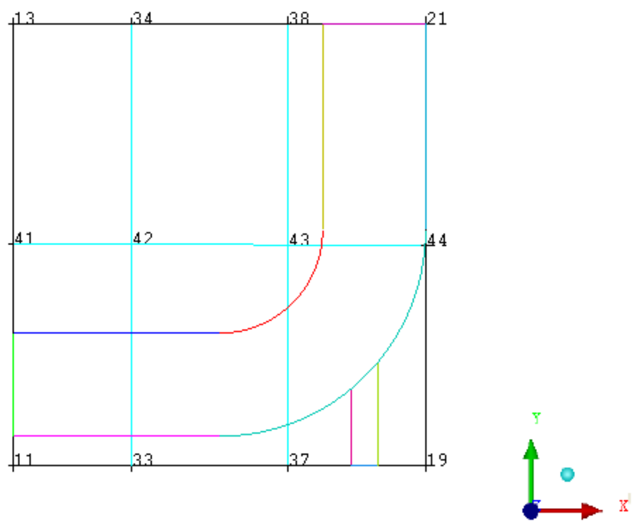


Figure 8: Curves and FLUID Block After Three Splits

3. Delete unnecessary blocks.

The next step in this "top down" approach is to remove/delete the blocks those are not required.

Blocking > Delete Block 

(a) Disable Delete permanently.

(b) Click  (Select block(s)).

(c) Select the blocks to be deleted as shown in Figure 9 and click the middle-mouse button to accept the selection.

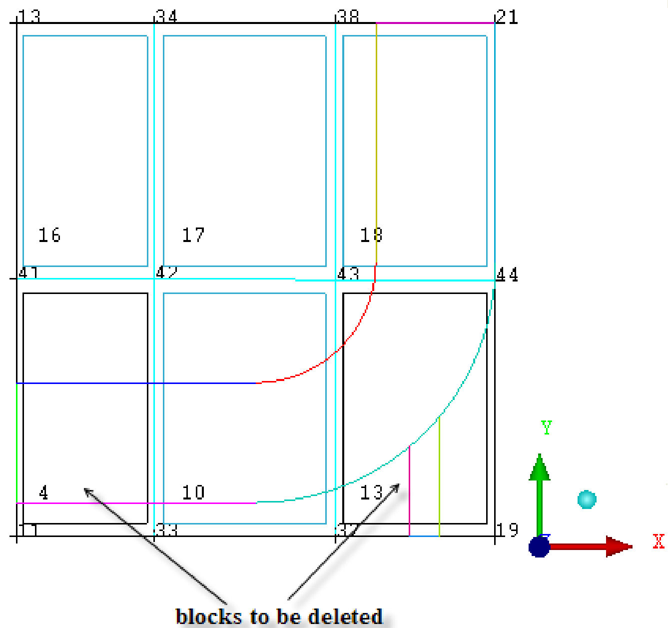


Figure 9: Blocks to be Deleted

(d) Click Apply in the Delete Block DEZ.

Note: *The deleted blocks with* Delete permanently *disabled (default) are actually put into the VORFN part, a default dead zone that is usually deactivated.*

The geometry and blocking of the model now resemble Figure 10.

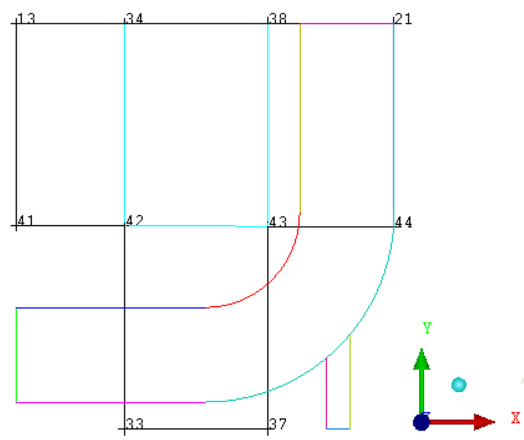


Figure 10: Final T Shape Topology

Step 2: Associating to Geometry

In this step, you will associate edges of the blocking to the curves of the CAD geometry. You should first select edges and then curves to which you want to associate the edges. If two or more curves are selected per operation, those curves will automatically be grouped (concatenated).

For reference, select Show Curve Names. See Figure 11.

Geometry RMB ⟶ Curves LMB ⟶ Show Curve Names

This is not required for edge to curve association, but it helps to illustrate the fact that each blocking edge is associated to named curve(s).

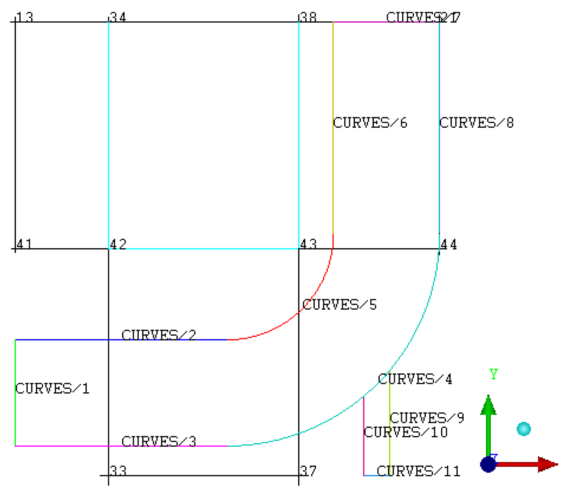


Figure 11: Vertex Numbers and Curve Names

1. Associate the inlet, the leftmost end of the large pipe.

   Blocking > Associate > Associate Edge to Curve

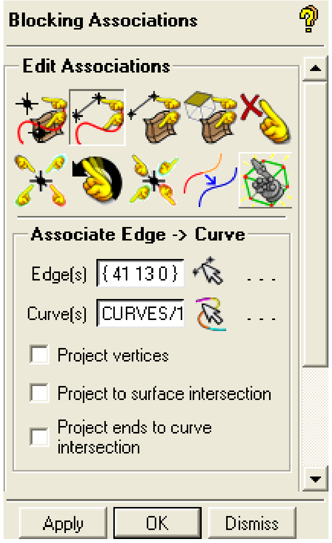
   (a) Select the required edges.

      i. Ensure that Project vertices are disabled (default).

      ii. Click (Select edge(s)).

      iii. Select edge 13-41.

      iv. Click the middle-mouse button to accept the selection.



(b) Select the appropriate curves.

i. Click  (Select compcurve(s)).

ii. Select the curve, CURVES/1.

iii. Click the middle-mouse button to accept the selection.

(c) Click Apply in the Associate Edge -> Curve DEZ.

The associated edge will turn green.

Note: *Associate edge to curve operation runs in continuation mode, allowing you to select the next set of edges and curves without reinvoking the function. The function will be cancelled, if you click the middle-mouse button or click* Dismiss*, without selecting entities.*

2. Similarly, associate the following edge/curve combinations to make the T fit the geometry:

- For small pipe, associate the following:
  - Edge 33-42 to curve CURVES/10.
  - Edge 33-37 to CURVES/11.
  - Edge 37-43 to CURVES/9.
- For outlet (top horizontal end of large pipe), associate the following:
  - Edge 21-44 to CURVES/7.

    This vertical edge will eventually be moved to capture the horizontal curve.

    Note: *When the entities are overlapped with other entities, disable the entity types. This will enable you to identify the right entity. For example, disable* Vertices *and* Edges *to verify the curve names. Enable the* Edges *to proceed with the selection.*

- For large pipe, associate the following:
  (a) Select all three edges (13-34, 34-38, and 38-21) and click the middle-mouse button to confirm the selection.
  (b) Select the three curves (CURVES/2, CURVES/5, and CURVES/6) and click the middle-mouse button.

  The three curves will automatically be grouped as one logical composite entity. Geometrically, they are still three separate curves.

  (c) Click Apply in the Associate Edge -> Curve DEZ.

3. Similarly, associate the edges 41-42, 43-44 to CURVES 3, 4, and 8.

   The blue (cyan) edges 42-43, 34-42, 38-43 *do not have to be associated. They are internal and will interpolate on the geometry when the mesh is computed.*

4. Verify that the correct associations have been set (Figure 12).

   Blocking [RMB] ⟶ Edges [LMB] ⟶ Show association

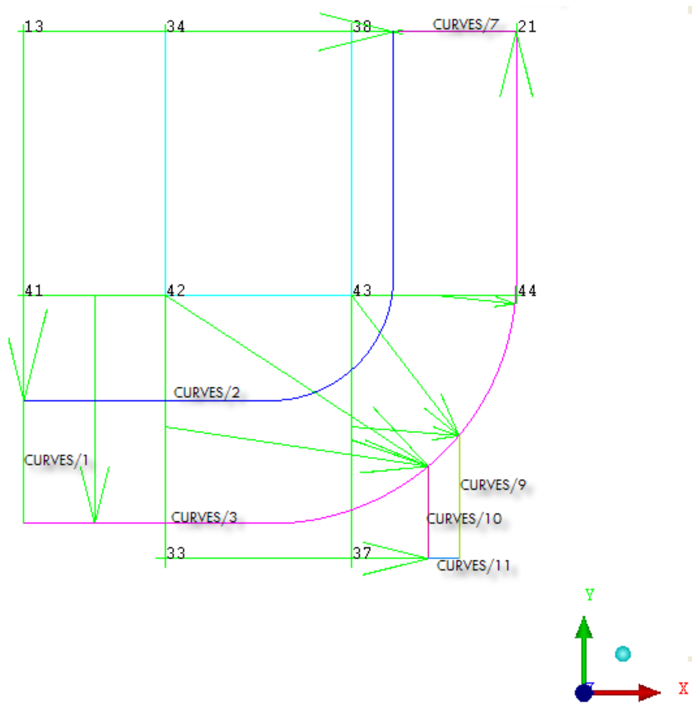


Figure 12: Projection of Edges to Curve

The green arrows in the display point from an edge to its associated curve. Nodes and vertices of these edges will project on to the associated geometry.

Note: *After completion, if the associations do not appear correctly, you can associate the edges to the correct curves again. It is not necessary to disassociate and then re-associate. Associating the edge to a new curve will overwrite the previous association. The steps of operation can also be retraced using* Undo *and* Redo *buttons.*

5. Deselect Show association.

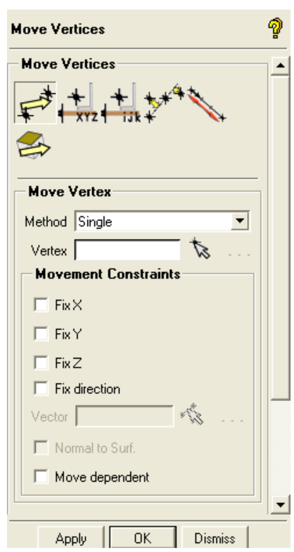
 Blocking RMB ⟶ Edges LMB ⟶ Show association

Step 3: Moving the Vertices

1. Manually move the vertices of inlets and outlet (ends of large pipe).

Blocking > Move Vertex  > Move Vertex 

Note: *Selecting* Move Vertex *from the function tabs will prompt you to select from the screen. It is usually not necessary to select* Move Vertex *from the DEZ unless another option was previously selected.*



(a) Click  (Select vert(s)) and move the vertices. (Figure 13).

Select the Vertex*. Keeping the left-mouse button pressed, drag the vertex along the curve.*

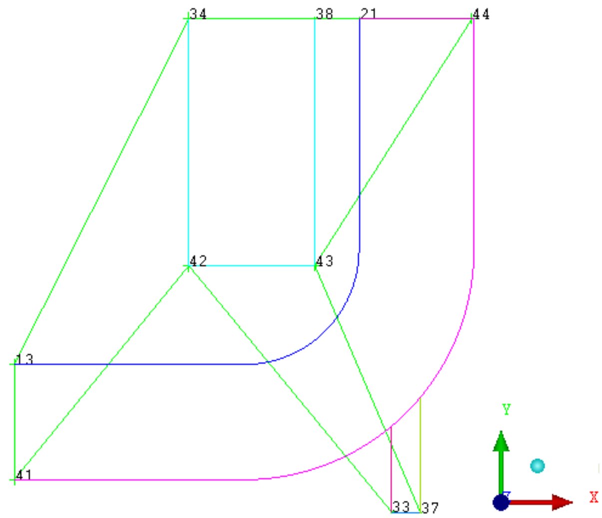


Figure 13: Moving Vertices

Notes:

- *Due to the associations made between the edges and curves, many of these vertices will snap to the correct position. Vertices can also be moved along the curve by dragging the mouse. To capture the ends of the curves:*
  i. *Select the vertex.*
  ii. *Keep left-mouse button pressed and drag the vertex along the curve until the vertex can be moved no further.*
  iii. *Position the cursor beyond the end of the curve so that the end is surely captured.*

  You may also prefer to associate the vertex with the points at the ends of the curves as described later in step 3.

- *The ends of the pipe are straight and it is possible to block this example without using the curve associations. However, the curve associations also create line elements on curves they are associated to. If you skip performing the curve associations, the boundary line elements will not be created. This will make it impossible to apply boundary conditions to that edge (such as inlet or wall). Hence, most CFD solvers give errors if any of the perimeter edges are not associated with perimeter curves.*

2. Move the remaining vertices to their appropriate positions on the geometry. See Figure 14.

   Try to make the blocks as orthogonal (good internal angles) as possible.

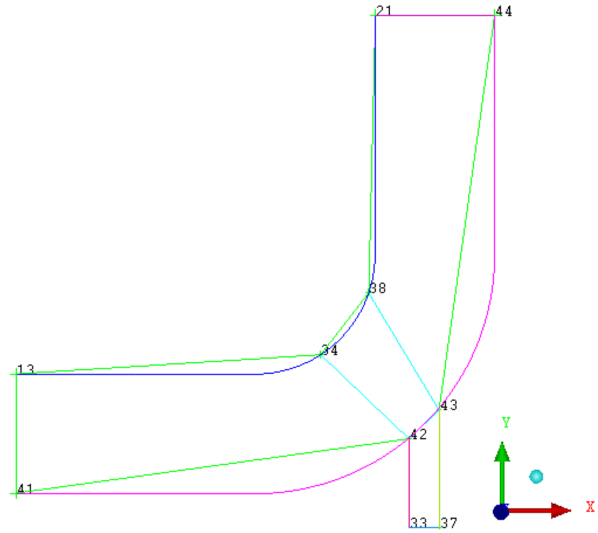


Figure 14: Moving Rest of the Vertices to Their Position

3. Associate the vertices to the points.

   This is an optional step.

   (a) Select Show Point Names.

   Geometry RMB ⟶ Points LMB ⟶ Show Point Names

   (b) Enable Associate Vertex.

   Blocking > Associate > Associate Vertex

   The Point *option is enabled by default.*

   (c) Select the Vertex and select the Point to which you want to associate the vertex.

   (d) Associate the vertices 13, 21, 41, 42, 33, 37, 43, 44 to points POINTS/2, POINTS/5, POINTS/1, POINTS/10, POINTS/9, POINTS/8, POINTS/11, POINTS/6.

   (e) Deselect Show Point Names.

   Geometry RMB ⟶ Points LMB ⟶ Show Point Names

4. Save the blocking.

   File > Blocking > Save Blocking As...

5. Provide a filename (`2D-pipe-geometry.blk`) so that the file can be reloaded at a later time, using File > Blocking > Open Blocking....

Step 4: Applying Mesh Parameters

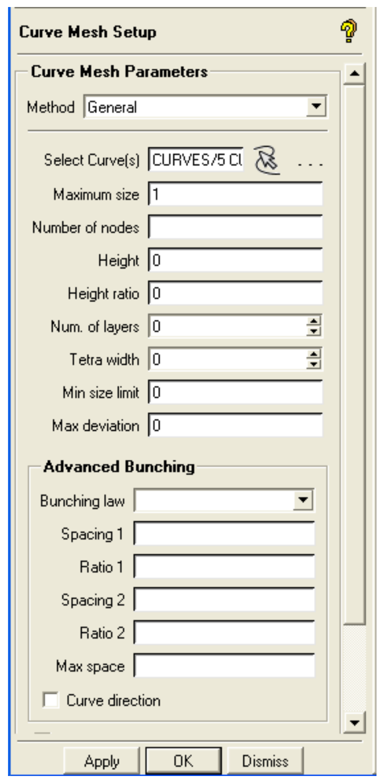
Set Mesh *parameters (sizes) on the geometry (curves in this 2D case). This is done at the geometry level and can be done before or after the blocking.*

Mesh > Curve Mesh Setup  > Select curve(s) 

1. Select Select all appropriate visible objects  from the selection tool bar.

   You can enter v *for visible or* a *for all.*

2. Set Maximum size to 1.



- Maximum size determines the length of the edges on the curve (or surface for 3D).
- Height determines the length of the edge of the first layer normal to the curve.
- Ratio (Ratio 1 and Ratio 2) determines the normal heights of the subsequent layers.

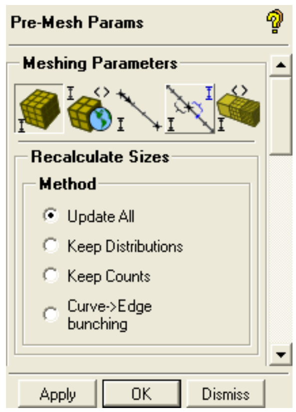
In this case, height and ratio are determined by the perpendicular curves whose Maximum size *will override any height or ratio settings.*

3. Ignore all other parameters and click Apply.

Step 5: Generating the Mesh

In this step, you will generate an initial mesh.

Blocking > Pre Mesh Params > Update Sizes



1. Enable Update Sizes.
   (a) Select Update All.
   This will automatically determine the number of nodes on the edges from the mesh sizes set on the curves.
   (b) Click Apply.
2. Enable Pre-Mesh.
   Blocking LMB ⟶ Pre-Mesh
   The Mesh *dialog box will appear, asking if you want to recompute the mesh. Click* Yes*.*

3. Disable Edges and Vertices.
   Blocking LMB ⟶ Edges
   Blocking LMB ⟶ Vertices
   The initial mesh is shown in Figure 15.

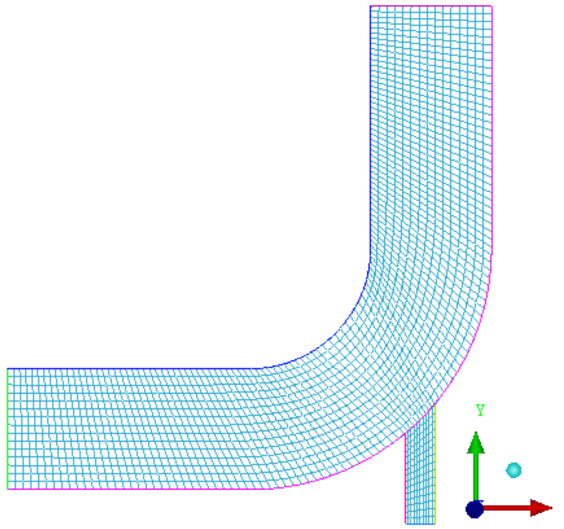


Figure 15: The Initial Mesh

Note: *The number of elements in initial mesh is sensitive to exact vertex placement (longest edge length in an index divided by max size found along that index). Hence, it may vary slightly from Figure 15.*

Step 6: Adjusting the Edge Distribution and Refining the Mesh

In this step, you will use advanced edge meshing features to re-distribute grid points to resolve the salient features of the flow.

1. Disable Pre-Mesh.

   Blocking LMB ⟶ Pre-Mesh

2. Re-display Curves and Edges.

3. See the distribution of grid points along the edges (Figure 16).

   Blocking RMB ⟶ Edges LMB ⟶ Bunching

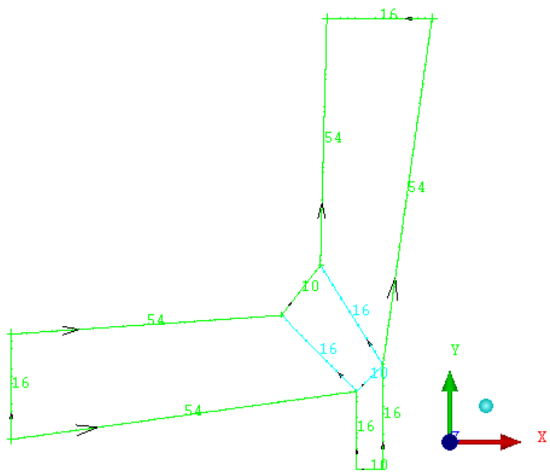


Figure 16: Bunching the Edges

4. Reduce the number of nodes along the length of the large pipe.

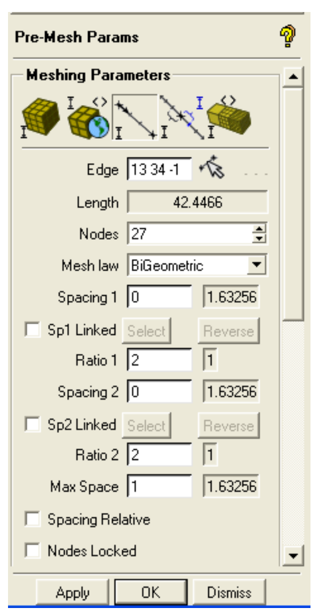
   Blocking [RMB] ⟶ Vertices [LMB] ⟶ Numbers

   (a) Display the edge meshing parameters.

   Blocking > Pre-Mesh Params > Edge Params

   (b) Select Select edge(s) or Edge Params and select edge 13-34 when prompted.

   (c) In the Pre-Mesh Params DEZ, change the number of Nodes to 27. Click Apply.



5. Similarly, re-select  or , select edge 21-38, change Nodes to 27, and click Apply.

6. Enable Pre-Mesh and recompute to view the new mesh.

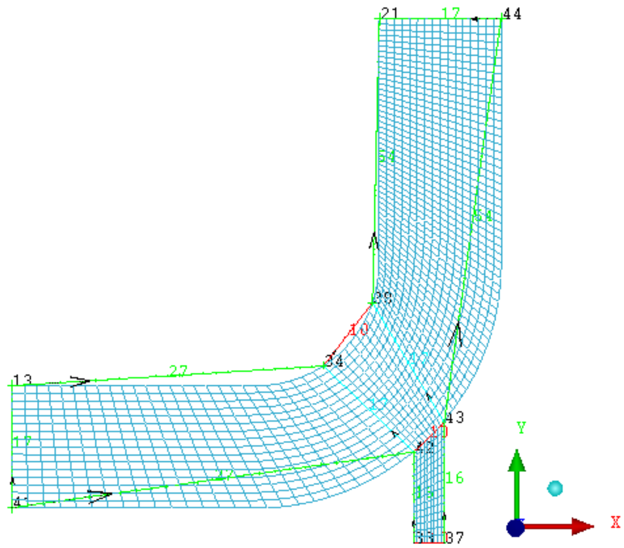


Figure 17: Edges Parameter

Note: *Figure 17 shows a structured grid. When the number of nodes is changed on one edge, all parallel opposing edges will automatically have the same number of nodes. In this case, edges* 41-42 *and* 43-44 *will have the same number of nodes as edges* 13-34 *and* 38-21 *respectively.*

7. Disable Pre-Mesh and Curves to view the bunching on the edges. See Figure 18.

 Blocking LMB ⟶ Pre-Mesh

 Geometry LMB ⟶ Curves

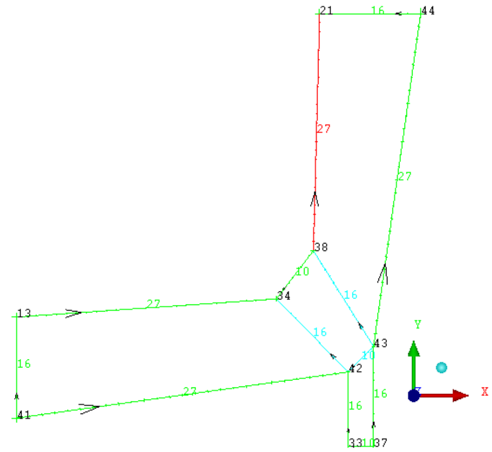


Figure 18: Bunching on Edges After Changing the Number of Nodes

8. Bias the nodes closer to the wall boundaries of the large pipe.

   (a) Click  and select edge 13-41.

      i. In the Pre-Mesh Params DEZ, enter `0.5` for Spacing 1 and Spacing 2.

         Note: Spacing 1 *refers to the node spacing at the beginning of the edge, and* Spacing 2 *refers to the spacing at the end of the edge. The beginning of the edge is shown by the white arrow after the edge is selected.*

      ii. Enter `1.2` for Ratio 1 and Ratio 2.

      iii. Click Apply.

      Requested values for spacing and ratio are entered in the first column. Actual values are displayed in the second column. The requested ratios cannot be attained due to the number of Nodes, Mesh law, Spacing1, *and* Spacing2. *Increase the number of* Nodes *using the arrow until the ratios are close to the entered value,* 1.2.

      Note: *The* Mesh law *is by default set to* BiGeometric. *This allows the nodes to be biased towards both ends of the edge. The expansion rate from the end is a linear progression. Several other mathematical progression functions (laws) are available.*

   (b) Ensure that the parallel edges 34-42, 38-43, and 21-44 have the same spacing.

      i. Enable Copy Parameters in the Pre-Mesh Params DEZ.

      ii. Ensure that To All Parallel Edges is selected from Method drop-down list.

      iii. Click Apply.

(c) Select edge 21-38 and enter 0.5 for Spacing 1 and Spacing 2 and click Apply.

This will concentrate grid points toward the outlet and toward the small pipe. To have these changes reflected in edge 43-44 *as well, ensure that* To All Parallel Edges *is selected from* Method *drop-down list.*

9. Copy the same distribution to the other section of the large pipe.

   (a) Select To Selected Edges Reversed from the Method drop-down list.

   (b) Select the Select edge(s) icon immediately underneath the Method field.

   (c) Select the edge 13-34.

   (d) Click the middle-mouse button or click Apply.

10. Refine the nodes along the small pipe.

    (a) Click Select edge(s) (icon toward the top of the menu) and select the edge 33-42.

    (b) Enter 9 for the Nodes in the DEZ.

    (c) Enter 1.0 and 0.5 for Spacing 1 and Spacing 2 respectively.

    (d) Select To All Parallel Edges from the Method drop-down list.

    (e) Click Apply.

11. Select edge 34-38 and enter 9 for the Nodes. Click Apply.

12. Enable Pre-Mesh and recompute to view the refined mesh shown in Figure 19.

    Blocking LMB ⟶ Pre-Mesh

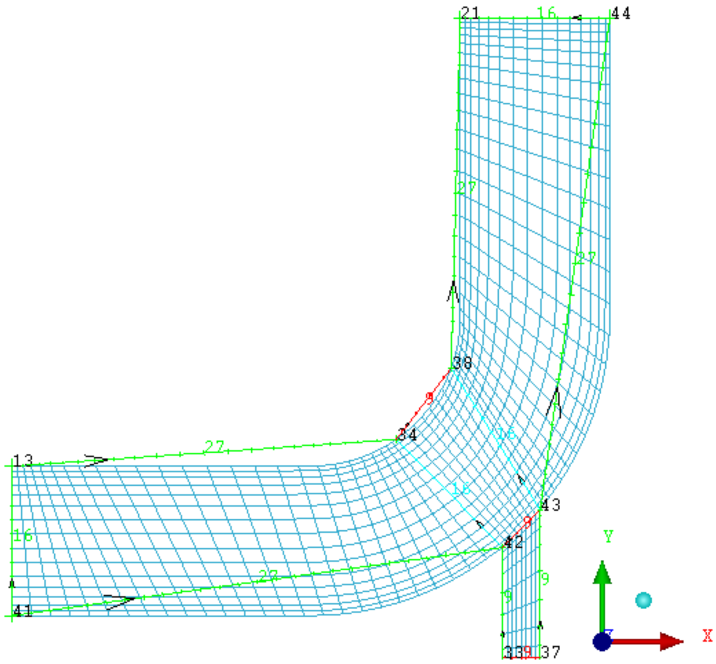


Figure 19: The Refined Mesh

Step 7: Matching the Edges

In this step, you will match the edge spacing of a Reference Edge *to a connecting* Target Edge(s). *You will modify the node spacing on the end of the target edge that connects to the reference edge to match the node spacing on the reference edge.*

Blocking > Pre-Mesh Params > Match Edges

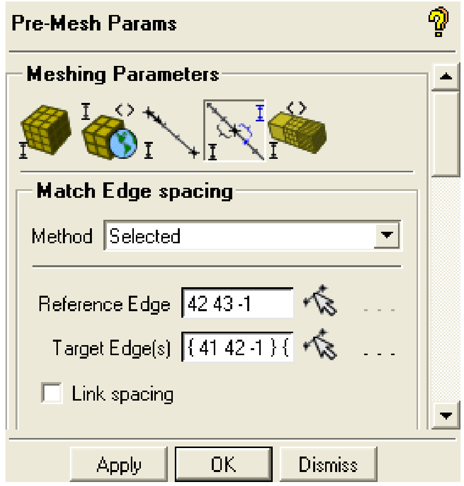
1. Match the edge spacing manually.

   (a) Select Selected from the Method drop-down list.

   (b) Click for the Reference Edge.

   (c) Select the edge, 42-43 and click middle-mouse button to accept the selection.

   (d) Click for the Target Edge(s).



(e) Select the edges, 41-42 and 42-33 and click middle-mouse button to accept the selection.

(f) Click Apply in the Match Edge spacing DEZ.

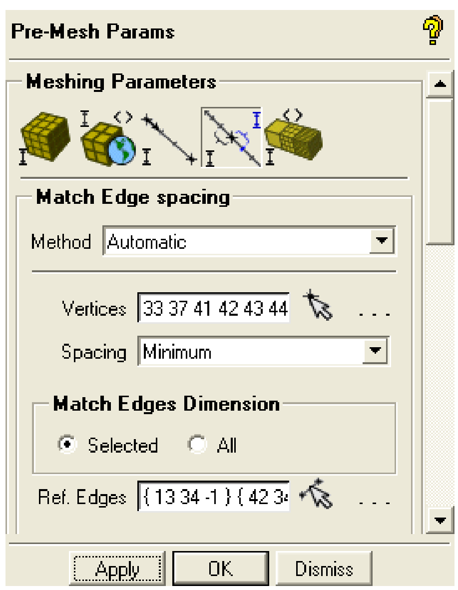
(g) Enable Pre-Mesh.

Blocking LMB ⟶ Pre-Mesh

The Mesh *dialog box will appear, asking if you want to recompute the mesh. Click* Yes*.*

2. Match the edge spacing automatically.

(a) Select Automatic from the Method drop-down list.



(b) Ensure that the Spacing is Minimum.

(c) Click Vertices.

(d) Select Select all appropriate visible objects from the selection tool bar.

*You can also enter **v** for all visible vertices or drag a box to select all vertices.*

(e) Click middle-mouse button to accept the selection.

(f) Click for the Ref. Edges.

(g) Select the edges, 13-34 and 34-42 and click middle-mouse button to accept the selection.

This selection chooses the i and j index at each vertex for matching.

(h) Click Apply.

(i) Disable Pre-Mesh and re-enable it.

Blocking [LMB] ⟶ Pre-Mesh

The Mesh *dialog box will appear, asking if you want to recompute the mesh. Click* Yes.

Figures 20 and 21 show the comparison of mesh before and after matching the edges.

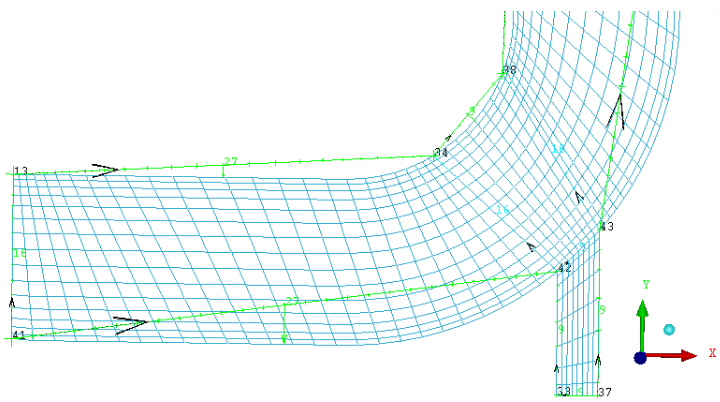


Figure 20: Mesh Before Edge Matching

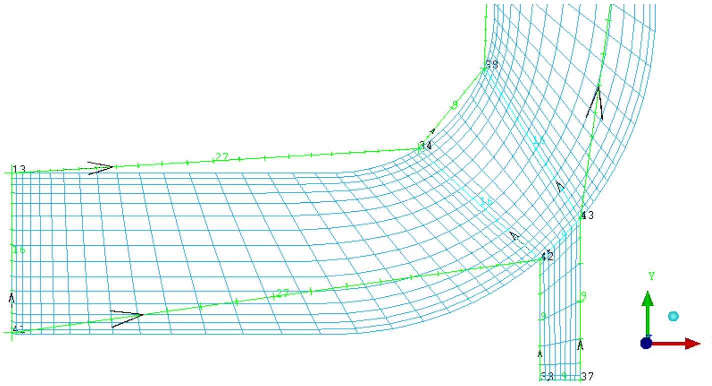
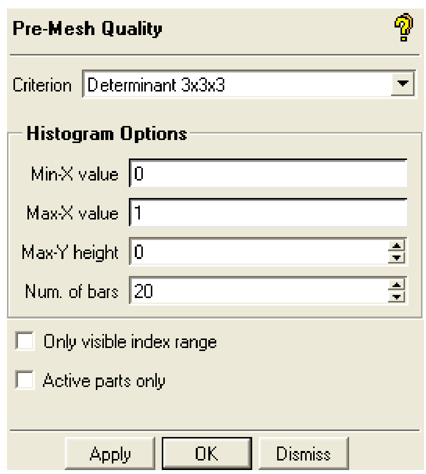


Figure 21: Final Mesh After Edge Matching

3. Check the quality of the mesh.

   Blocking > Pre-Mesh Quality Histograms



   (a) Retain the default settings.

   (b) Click Apply.

   (c) Select the worst two bars from the histogram.

   The selected bars will be highlighted in pink (Figure 22).

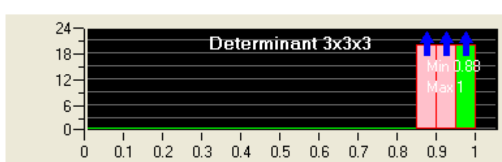


Figure 22: Histogram

The highlighted elements are shown in Figure 23.

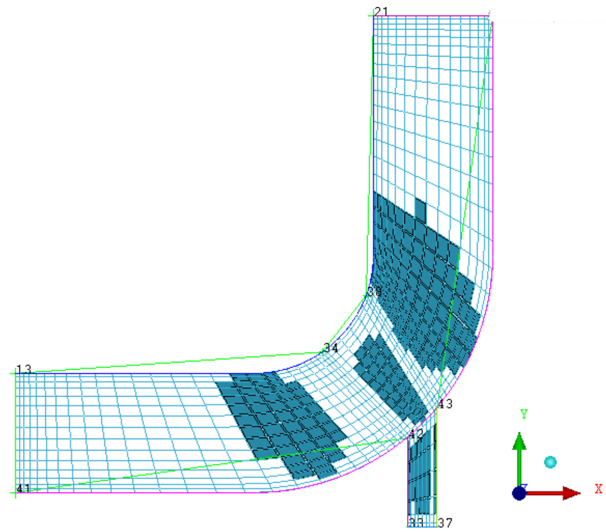


Figure 23: Worst Quality Elements Highlighted

Step 8: Verifying and Saving the Mesh

1. Save the mesh in unstructured format.

   Pre-Mesh > Convert to Unstruct Mesh

2. Save the blocking file (`2D-pipe-geometry-final.blk`).

   File > Blocking > Save Blocking As...

   *This block file can be loaded in a future session (*File > Blocking > Open Blocking*) for additional modification or to mesh a similar geometry. Save each blocking to a separate file instead of overwriting a previous one. In more complex models, you may have to back track and load a previous blocking.*

3. Save the project file (`2D-pipe-geometry-final.prj`).

   File > Save Project As...

   This will save all the files—tetin, blocking, and unstructured mesh.

4. Exit the current session.

   File > Exit