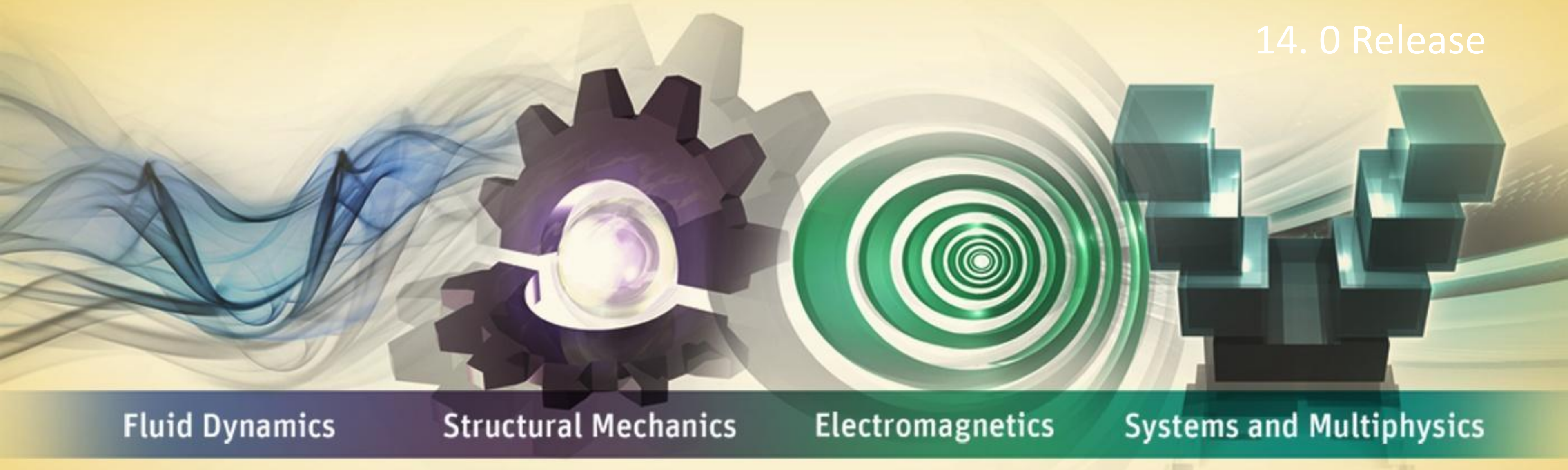
Realize Your Product Promise™

Workshop 8.1
2D Pipe Junction

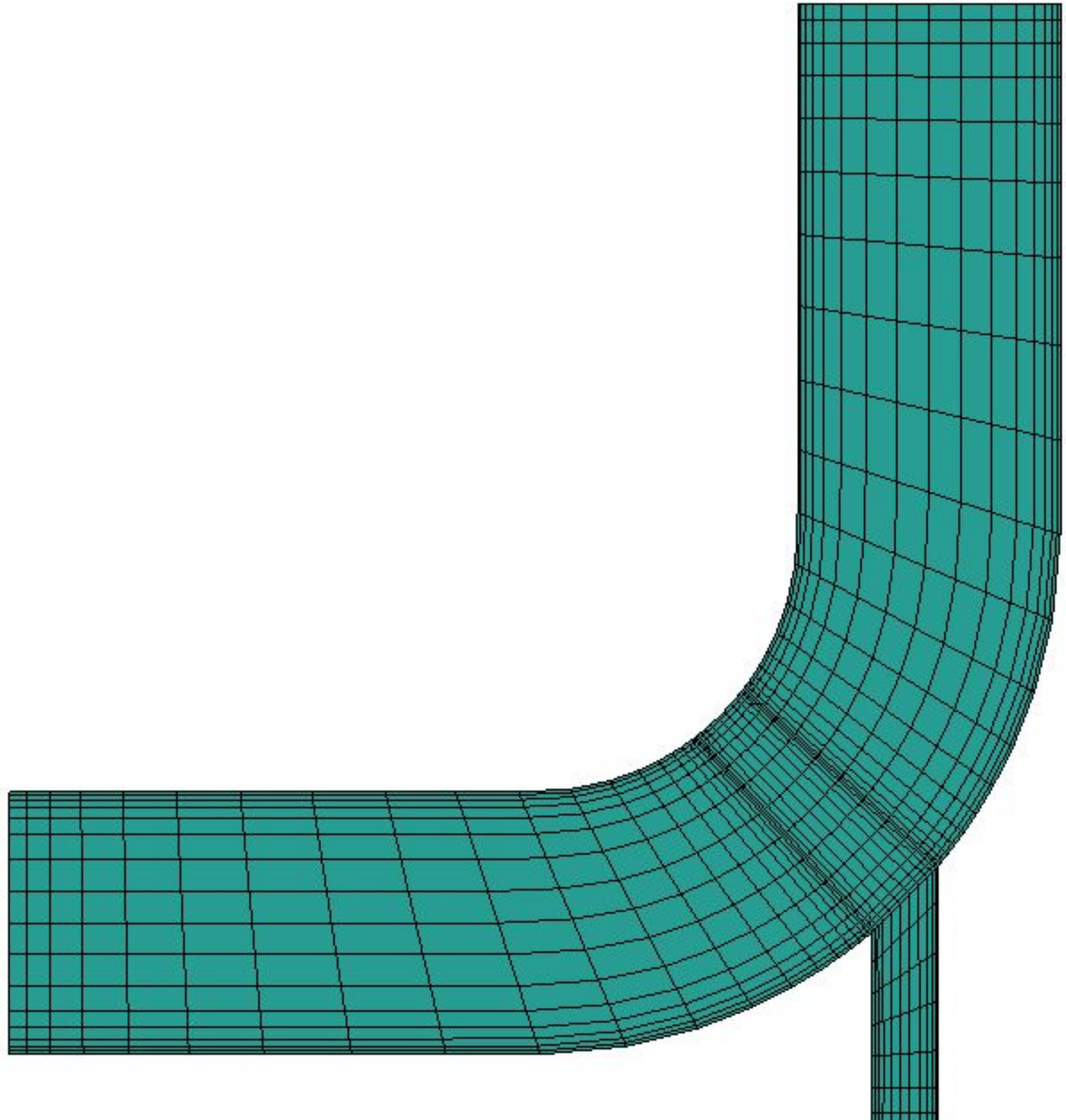
14. 0 Release



Introduction to ANSYS ICEM CFD

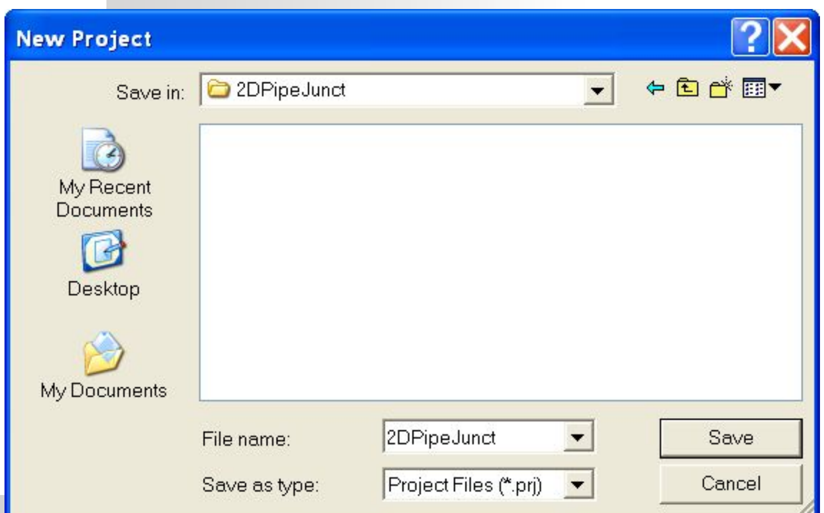
2D Pipe J

- **2D Pipe Junctior**
  - This 2D model s
    the basic proces
- **This tutorial dem**
  - Top down proce
    desired grid line
  - Edge associatio
  - Curve grouping
  - Match edges
  - Convert to an U



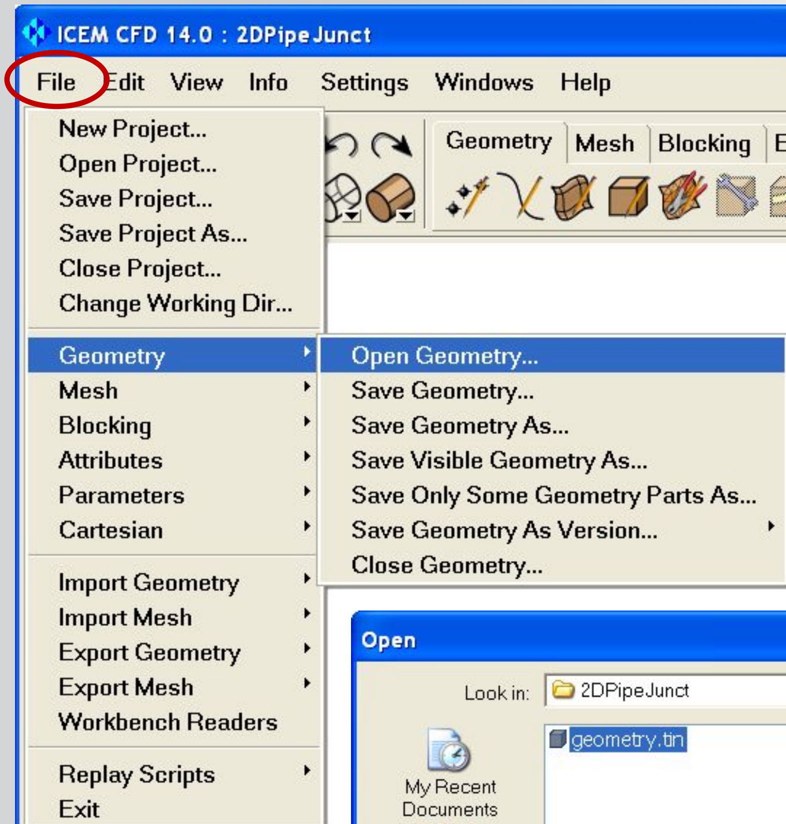
Create a Project

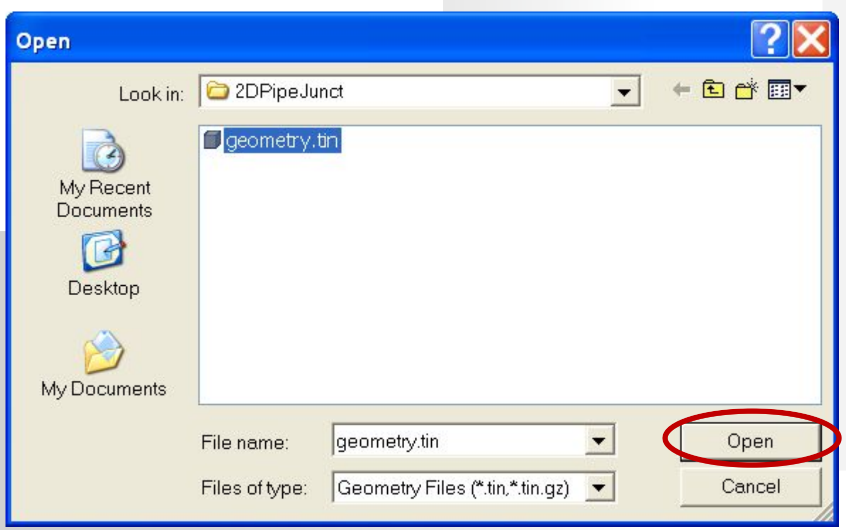
- Create the project
  - *File > New Project*
  - Browse to the working directory (2DPipeJunct)
  - Enter any new project name (2DPipeJunct in this case)
  - *Save* (it will append .prj if this extension is not already there)



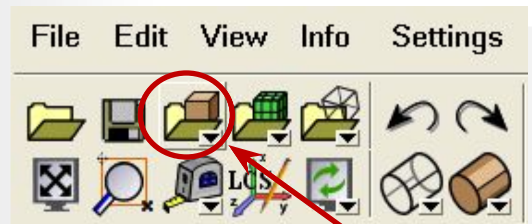
  - The pull down next to the file name can be used to quickly locate recently used projects.
  - The Project file contains information about project settings, the working folder and file associations.
  - After saving project, simply loading the project file will load all associated files within the working directory

Open Geometry

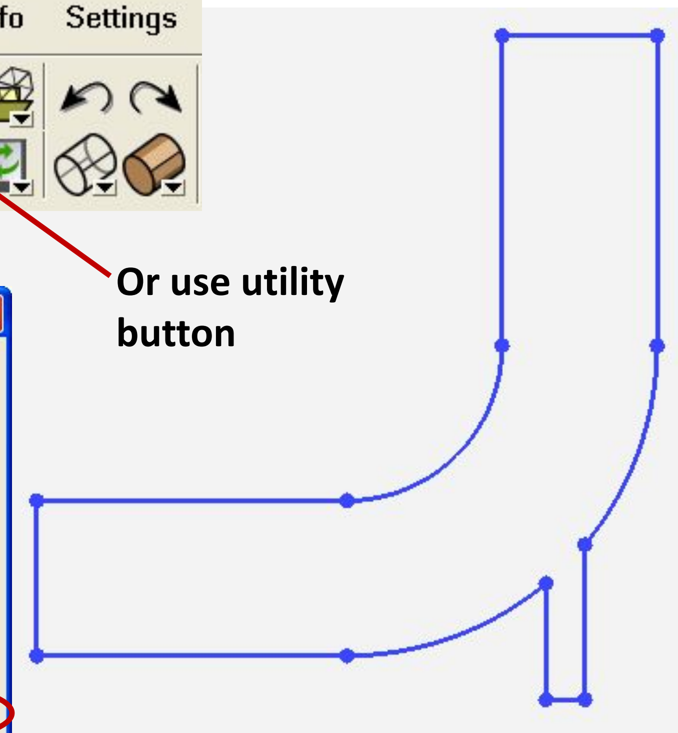




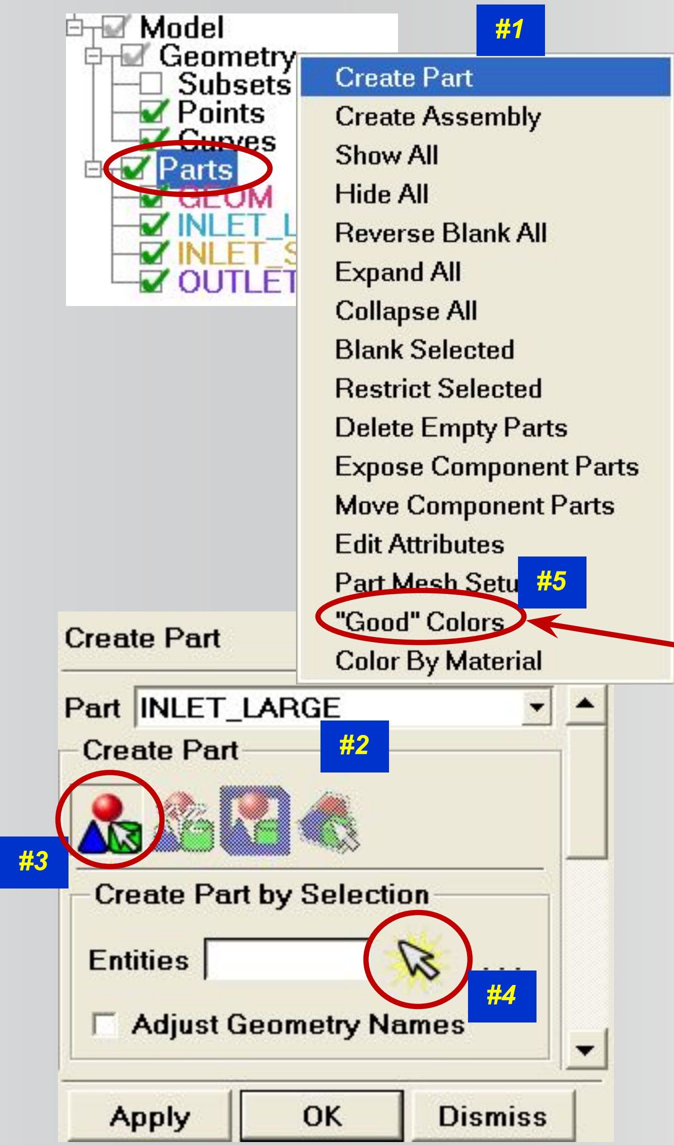
- **Open geometry**
  - *File > Geometry > Open Geometry*
  - Choose *geometry.tin*
  - *Open*



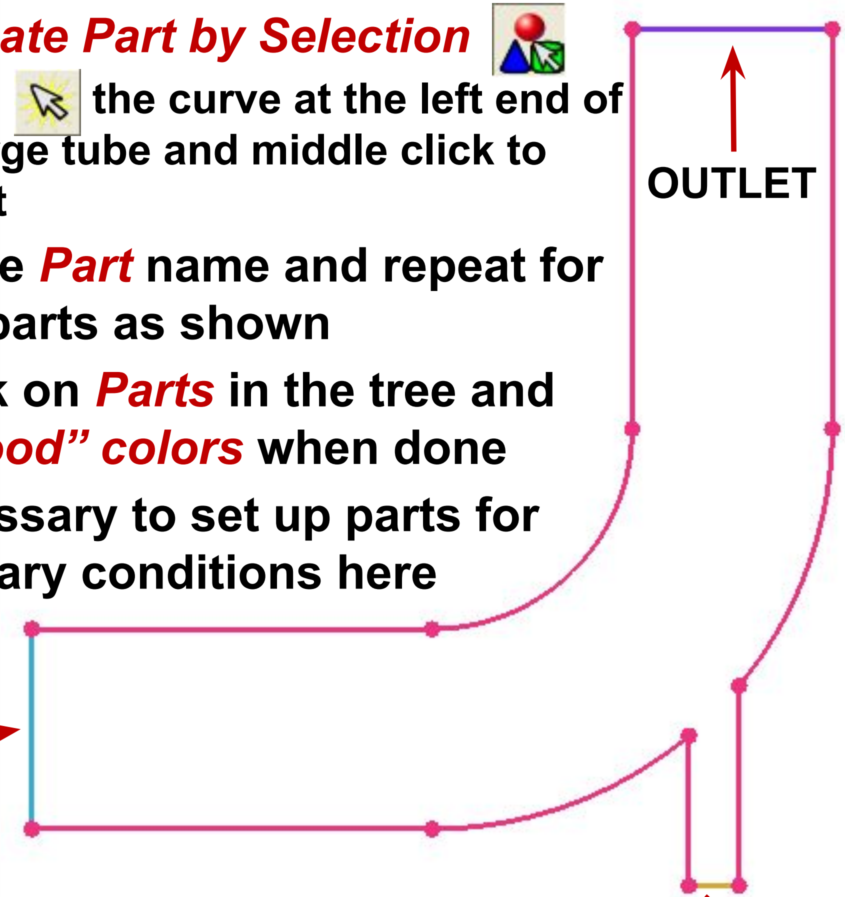
Or use utility button



Create Parts

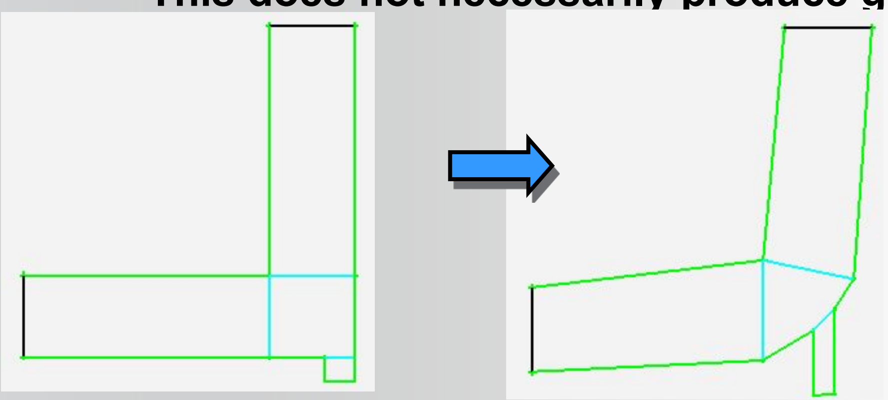


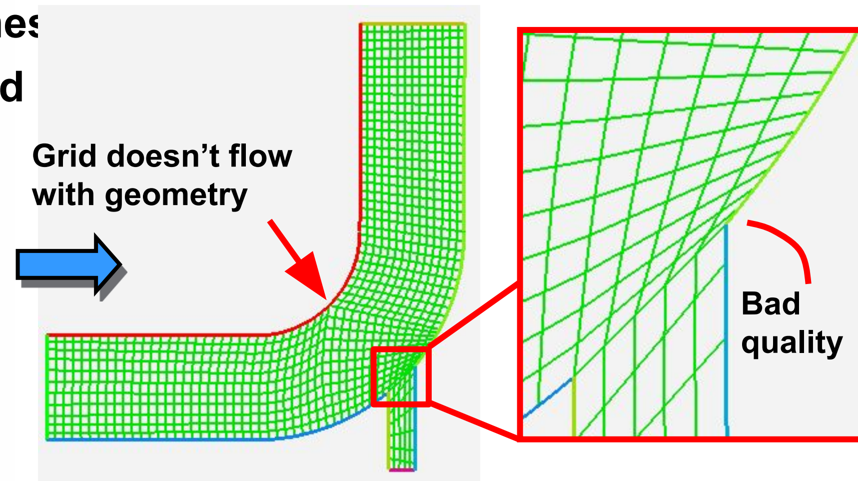
- Right-click on ***Parts*** in the tree and select ***Create Part***
- Enter **INLET_LARGE** as the ***Part*** name
- Select ***Create Part by Selection***
  - Select the curve at the left end of the large tube and middle click to accept
- Change the ***Part*** name and repeat for the other parts as shown
- Right-click on ***Parts*** in the tree and select ***"Good" colors*** when done
- Only necessary to set up parts for the boundary conditions here



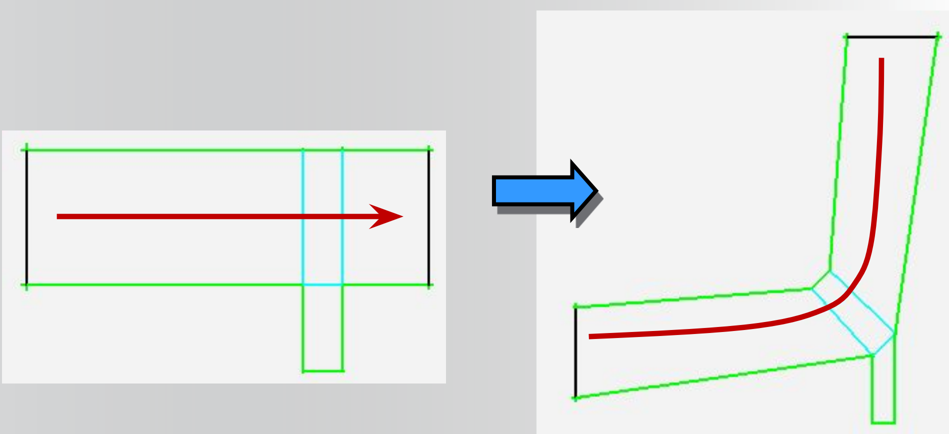
Decide on Topology (Blocking Structure)

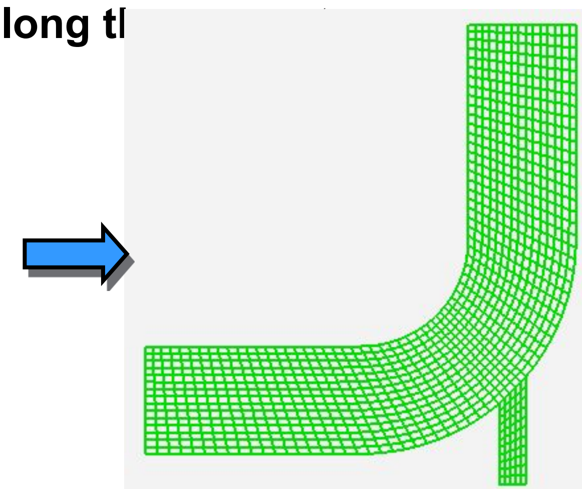
- Often, students just split wherever there are lines
  - This does not necessarily produce good



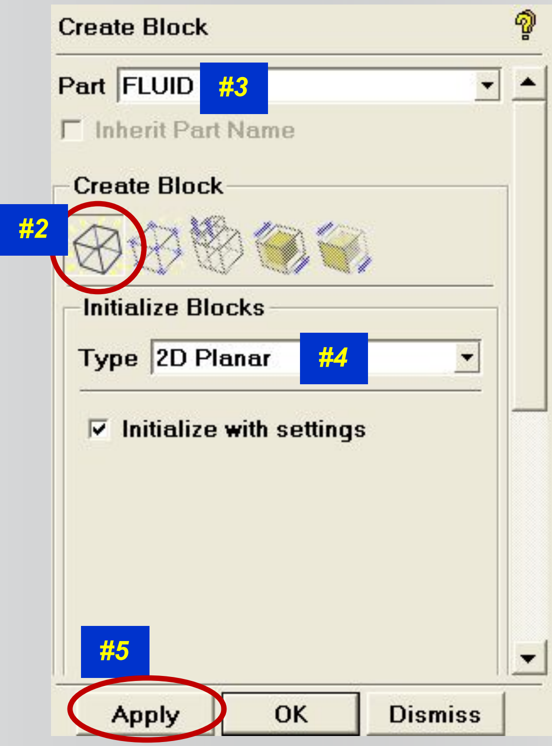


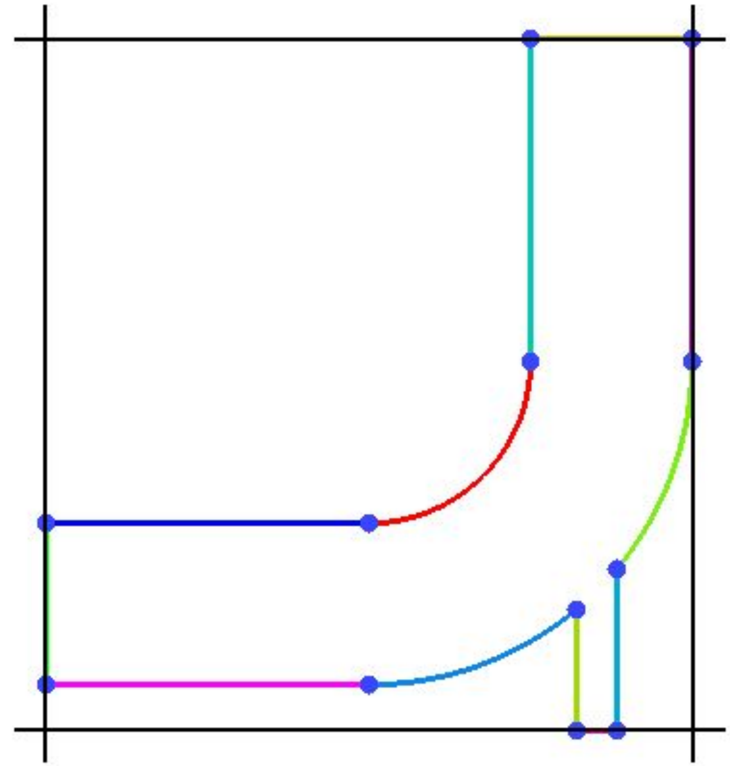
- A better topology is a “T” shape
  - This concept is initially difficult for many new users, but thinking “how do I structure the blocks to get grid lines to flow the way I want?” results in better mesh than simply trying to block along tl





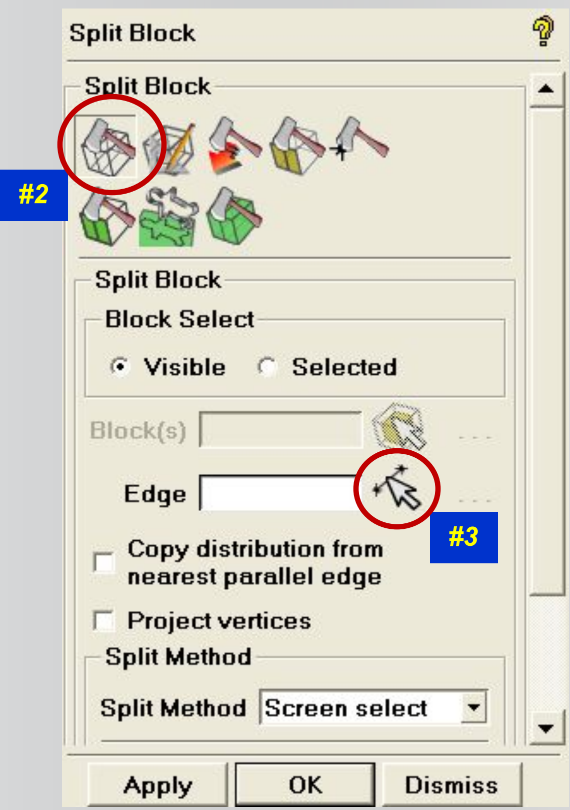
Initialize Blocking





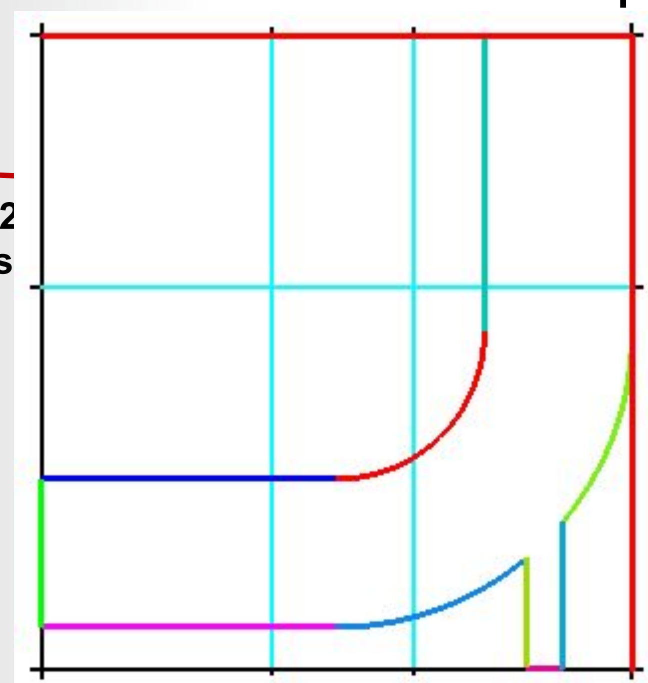
- **Begin blocking**
  - Select ***Blocking > Create Block > Initialize Blocks***
    - All other buttons will be grayed out until a first block is initialized
  - Type ***FLUID*** for the ***Part***
  - Change ***Type*** to ***2D Planar***
  - ***Apply***
  - This creates one block in the **FLUID** part which encloses the entire geometry
  - Curves automatically change color (colored separately instead of by part) to allow you to see where the ends of curves are

Split to Create Topology



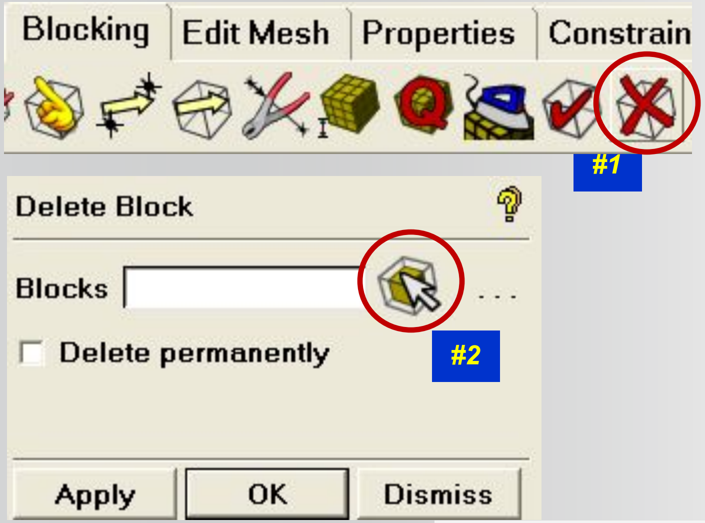
- Splitting blocks
  - Select ***Blocking > Split Block > Split Block***
    - Select the ***Select edges(s)*** button and left click on the edge to split
    - The new edge will be normal to the edge that you select
    - Hold left mouse button and drag the split to desired location
    - Middle mouse to complete or ***Apply***

Split this edge for the 2 vertical splits



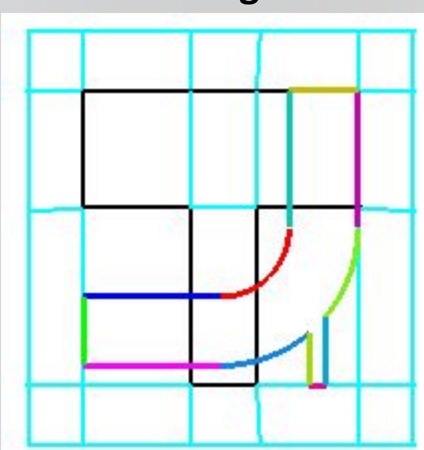
Split this edge for the horizontal split

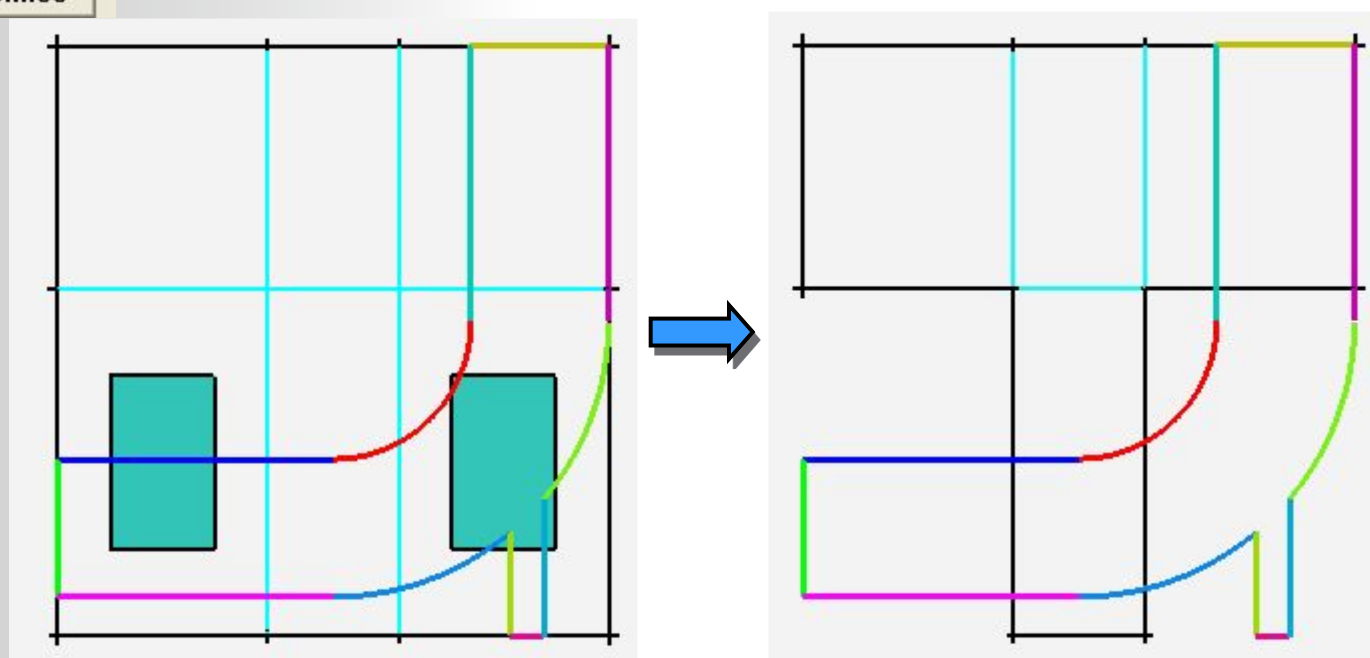
Delete Blocks



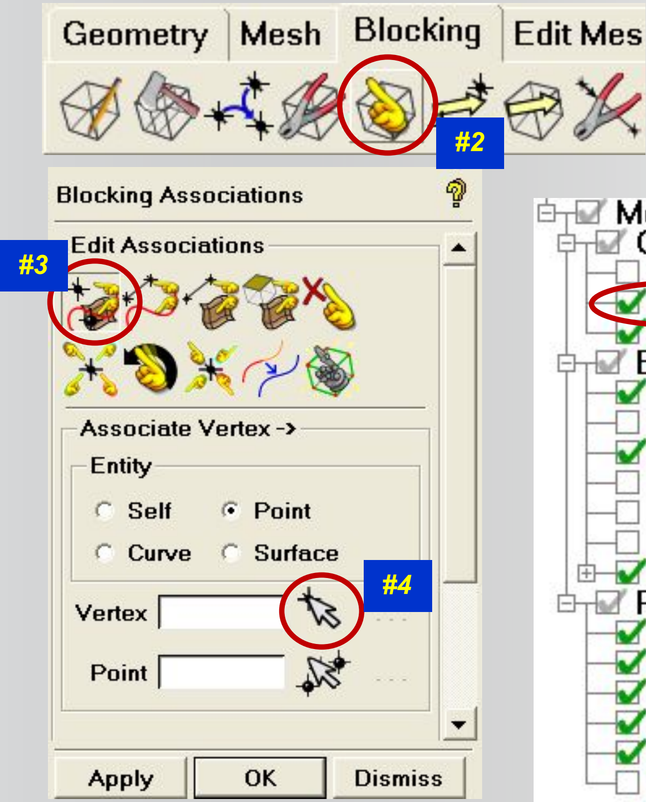
- Select ***Delete Block***
  - Select the lower corner blocks as shown
  - ***Apply*** or middle mouse
  - With ***Delete permanently*** OFF, this doesn't actually delete the blocks. It moves them to the part, ***VORFN***, so they can be used again, if desired
  - To actually delete the blocks, select ***Delete permanently***

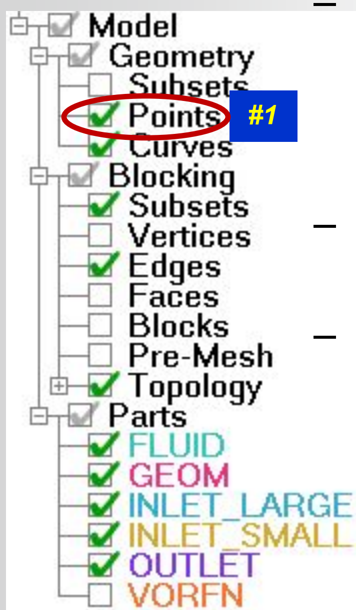
VORFN region



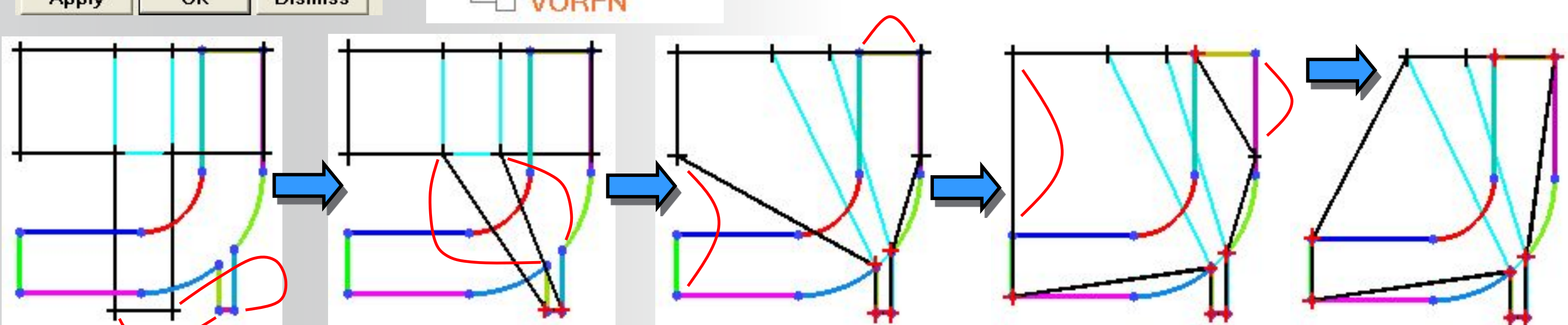


Vertex to Point Association

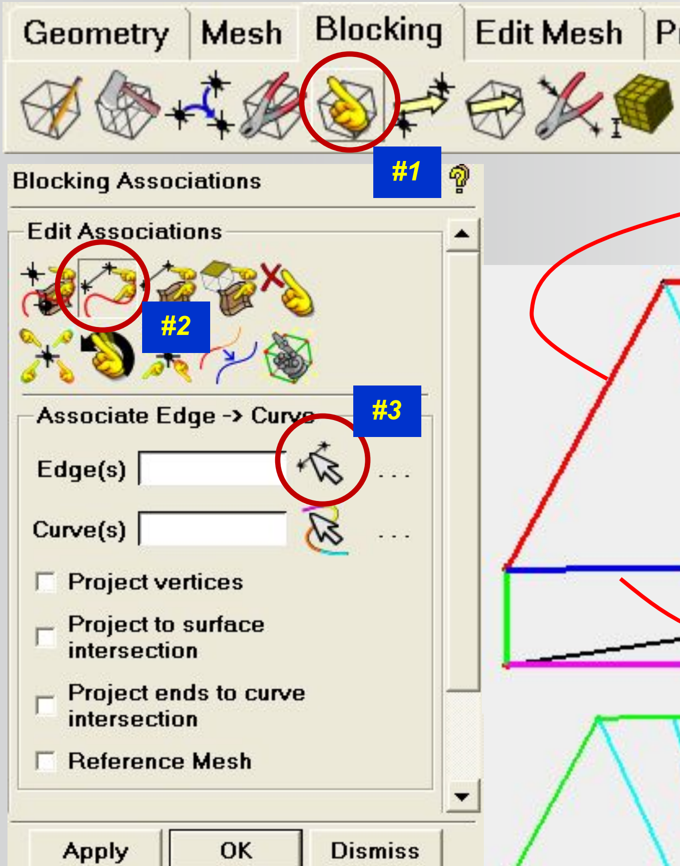




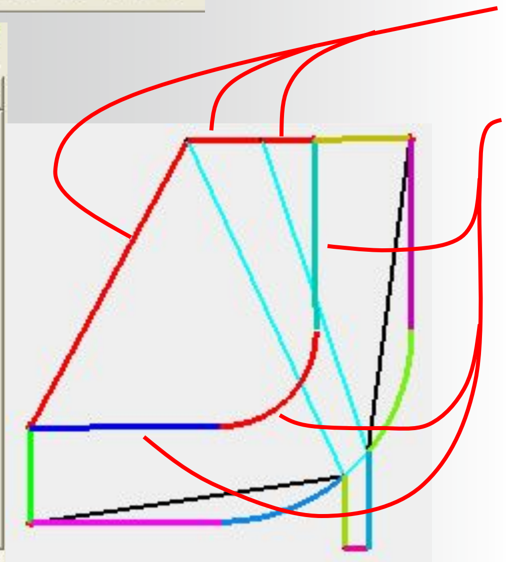
- **Associate vertices to points**
  - **Turn on *Points* in the model tree**
  - ***Select* Blocking > Associate > Associate Vertex**
    - ***Entity* type is already *Point*: Can proceed directly with selecting from screen (press *Select vert(s)* to enter selection)**
  - ***Select* one vertex, then select (left mouse) point, and it will jump there**
  - **Note change in color of vertices**
    - **White/black (boundary) to red (fixed – constrained to point)**

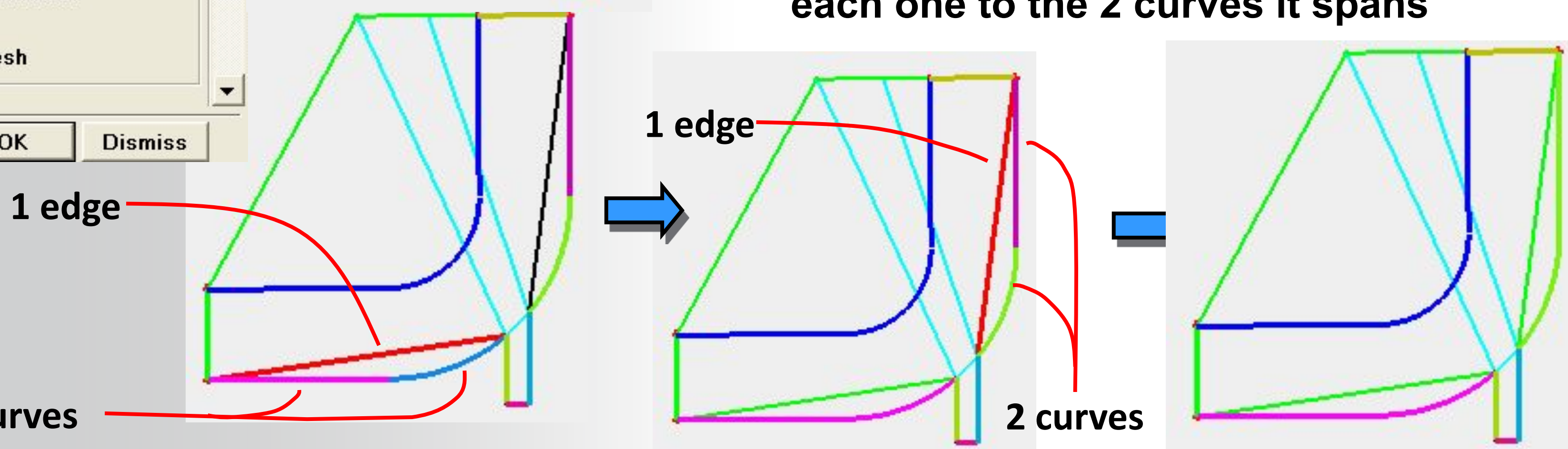


Associate Edges to Curves

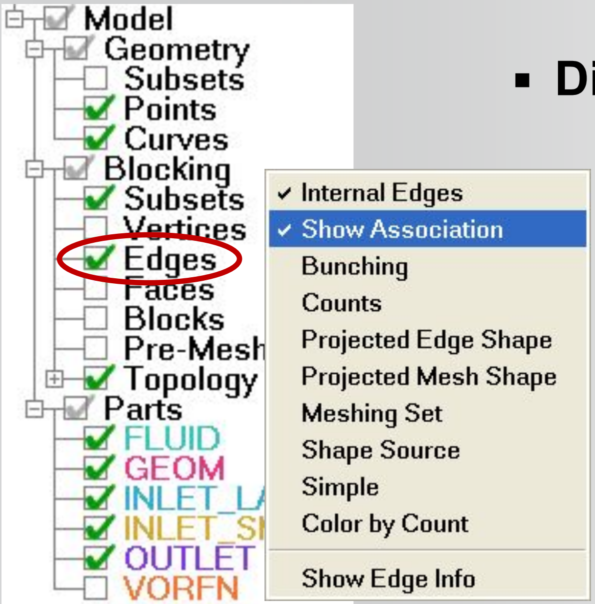


- Select ***Blocking > Associate > Associate Edge to Curve***
  - Select 3 edges as shown, then middle mouse
  - Select the 3 curves shown, then middle mouse
  - Curves automatically grouped into one
    - Note single color of grouped curve
  - Note single color of edges
    - White/black (boundary) to green (constrained to curve)
  - Repeat for 2 more edges, associating each one to the 2 curves it spans

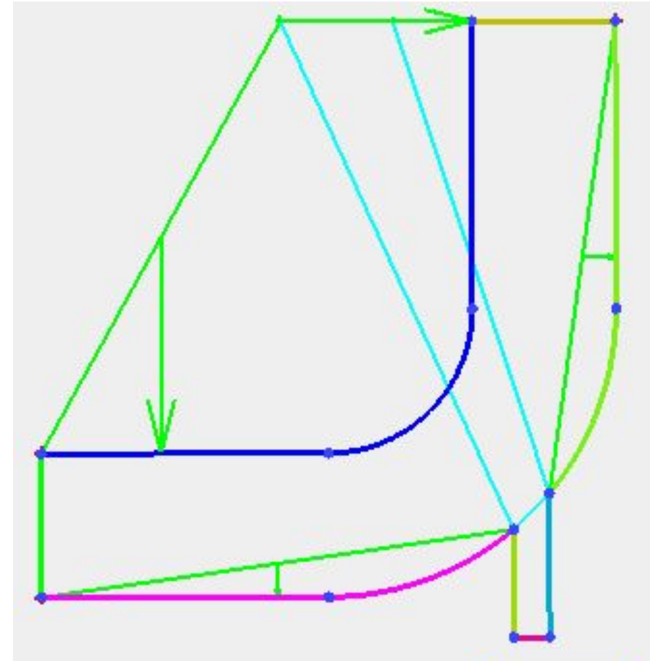




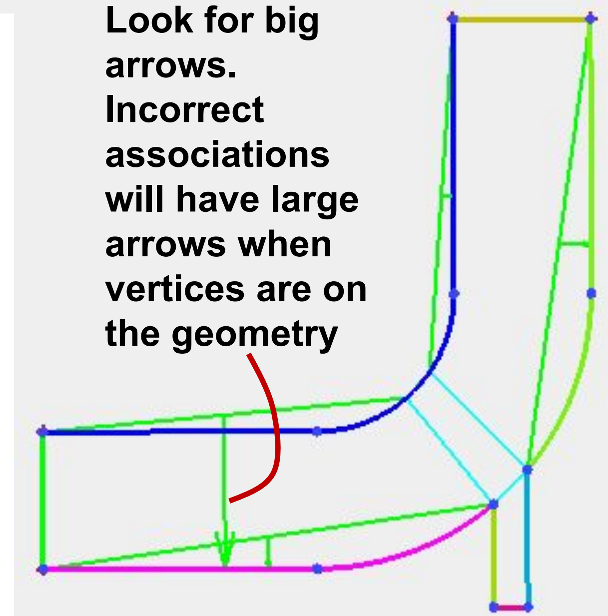
Displaying Associations



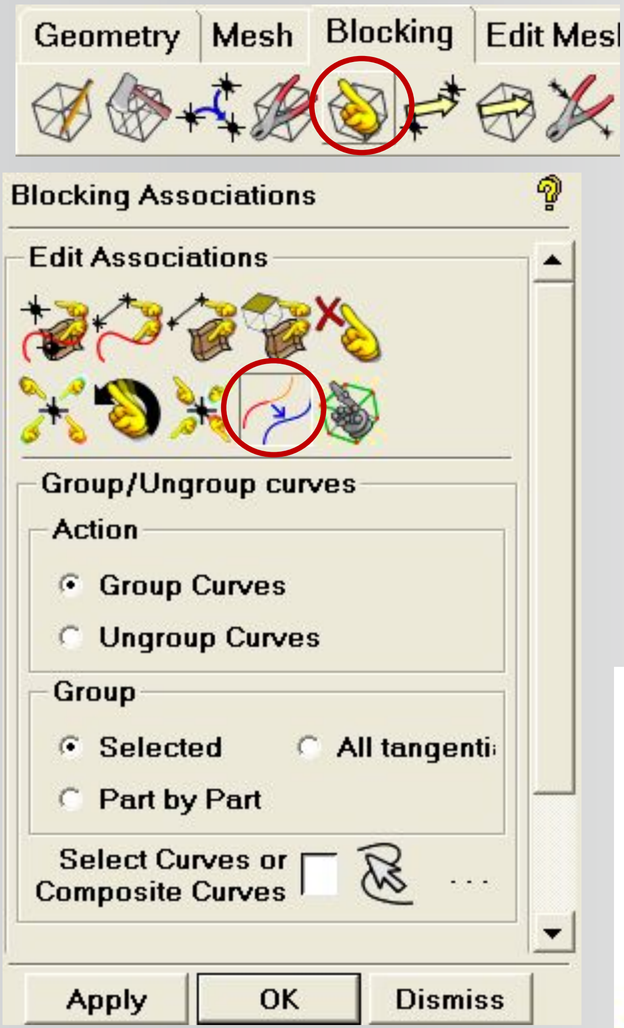
- Display association
  - Right click on ***Edges > Show Association*** in the model tree
  - Use to visually verify proper association
  - First tool in diagnosing projection problems



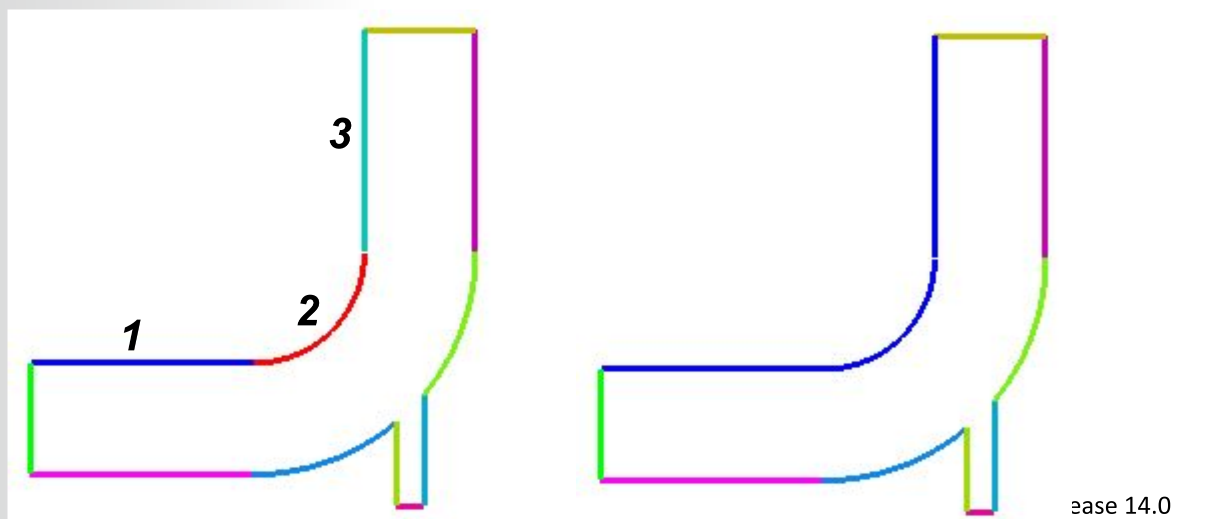
- Arrow display characteristics
  - One arrow starting from the center of every non-blue edge, and one arrow starting from every non-blue vertex pointing to the nearest location on associated entity
  - Invisible (zero in length) when edge or vertex is on top of the geometric entity it is associated to
  - If edge/vertex is mis-projected there's no need to use undo. Just redo the edge-to-curve association selecting the correct edges and curves.



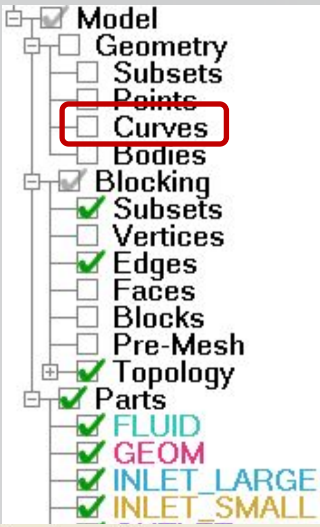
Grouping Curves



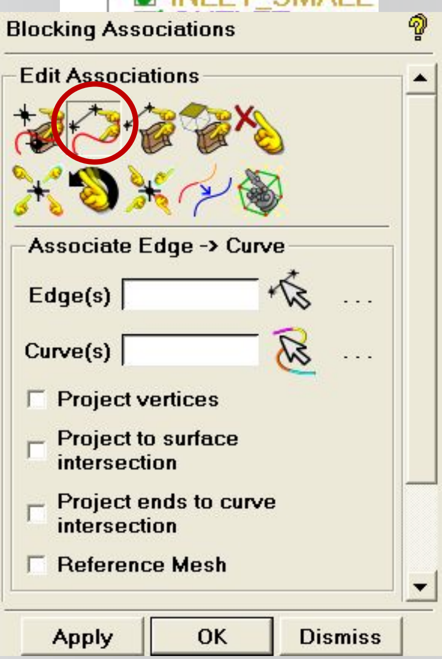
- **Alternatively: Group curves before associating edges**
  - **(not necessary for this exercise)**
  - **Select *Blocking > Associate > Group Curves***
  - **Select curves and middle mouse or *Apply***
  - **Color of first selected curve is taken**
  - **This doesn't concatenate the curves. It is only a grouping which is saved to the block file. Once the block file is closed, this grouping is gone.**
  - **Try group: *all tangential* to auto group all tangent curves in a model (this is most often the curves you will want grouped)**

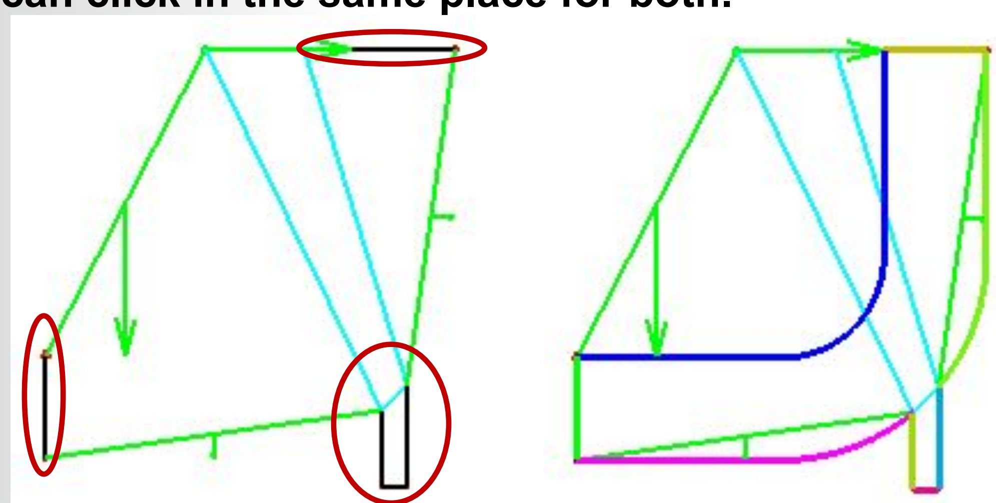


Finishing Edge to Curve Associations

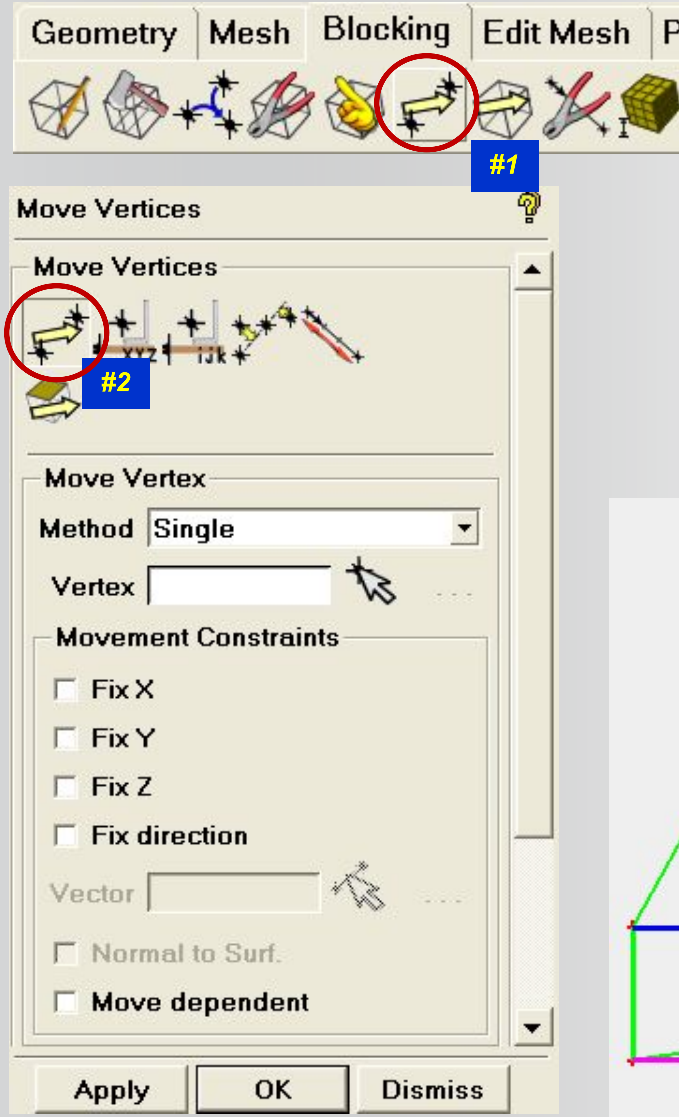


- **Associate remaining edges**
  - **Optional**
  - **Turn off *Curves* temporarily to view only *Edges***
  - **Straight edges that lie on top of straight curves do not need to be associated for mesh to project properly**
  - **However, since bar elements are only created on curve-associated edges, you may want to associate these edges in order to have elements to assign a boundary condition to the perimeter. Some solvers, such as Fluent, need the boundary elements.**
  - **For edges that lie on top of the curves, the selection highlight (red for edges, white for curves) is impossible to distinguish. Remember that the first selection will only select edges, then after middle mouse clicking, the second selection will only select curves, so you can click in the same place for both.**

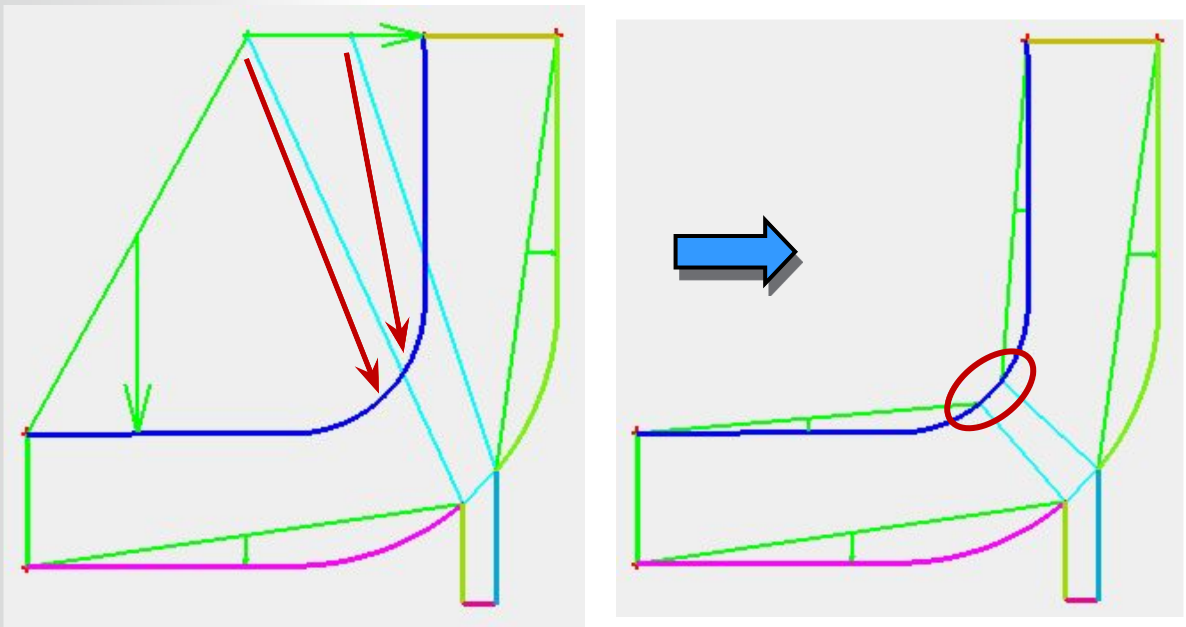




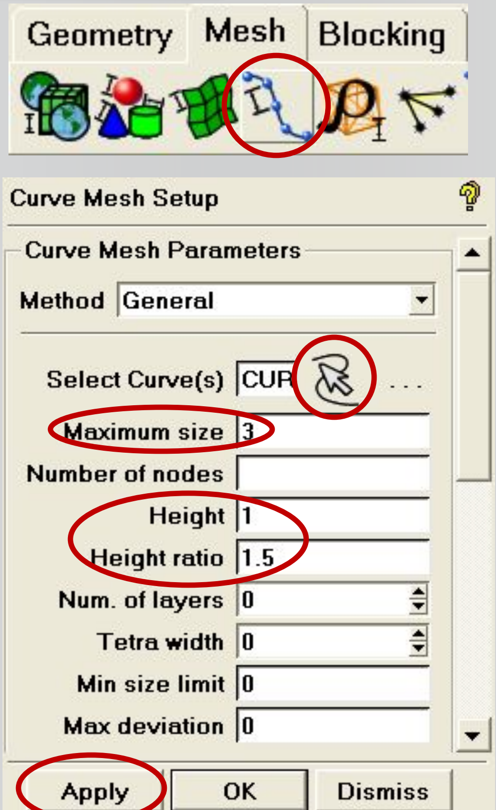
Move Vertices onto Geometry



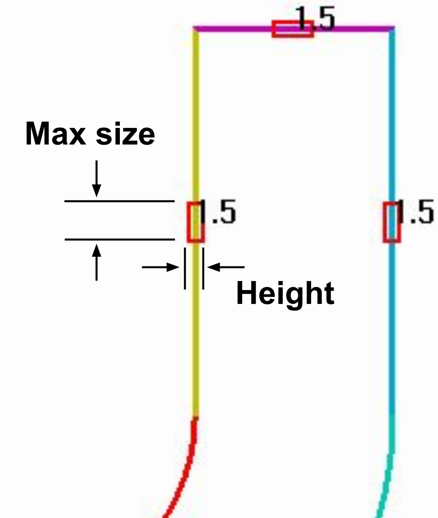
- Move remaining vertices
  - Select *Move Vertex > Move Vertex*
  - Left click, hold, and drag the vertex to desired location
  - Middle click when finished moving all vertices
  - Right mouse key will undo previous movement
  - Move vertices so blue (internal) edges are as normal to inner curve as much as possible

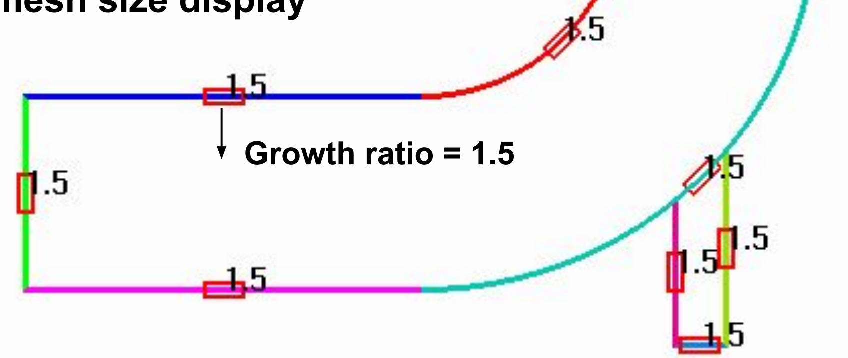


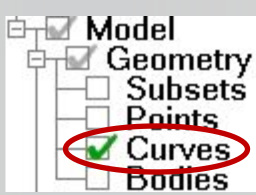
Mesh Sizes

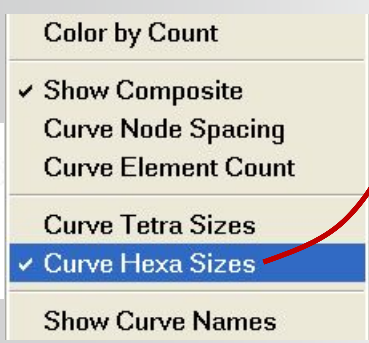


- **Set hexa sizes on curves**
  - Select ***Mesh > Curve Mesh Setup***
  - Press ***Select curve(s)*** button to enter curve selection
  - Type ***"a"*** (with cursor over main viewer) to select all curves
  - Set ***Maximum Size = 3, Height = 1,*** and ***Height ratio = 1.5***
  - ***Apply***
    - **Maimum Size** = max size of any edge of the element on that curve
    - **Height** is the first layer height (dimension normal to curve)
    - **Height Ratio** = multiplication factor to determine height of subsequent layers (in direction normal to curve)
  - Turn on ***Curves > Curve Hexa Sizes*** in model tree to view mesh size display

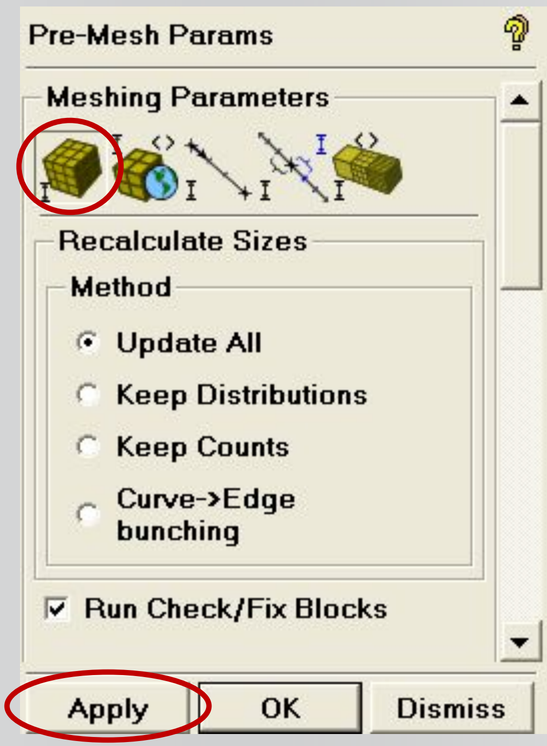




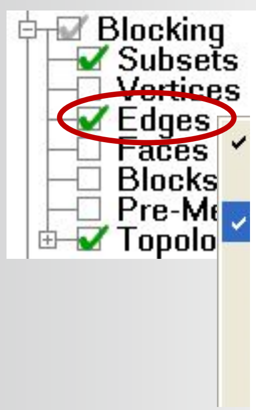


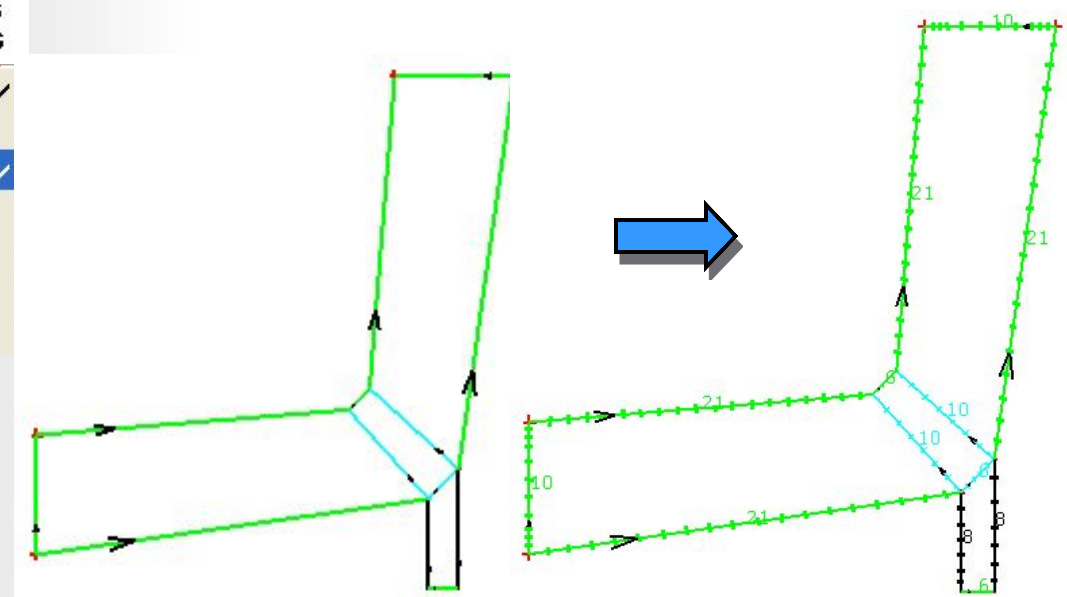


Update Sizes



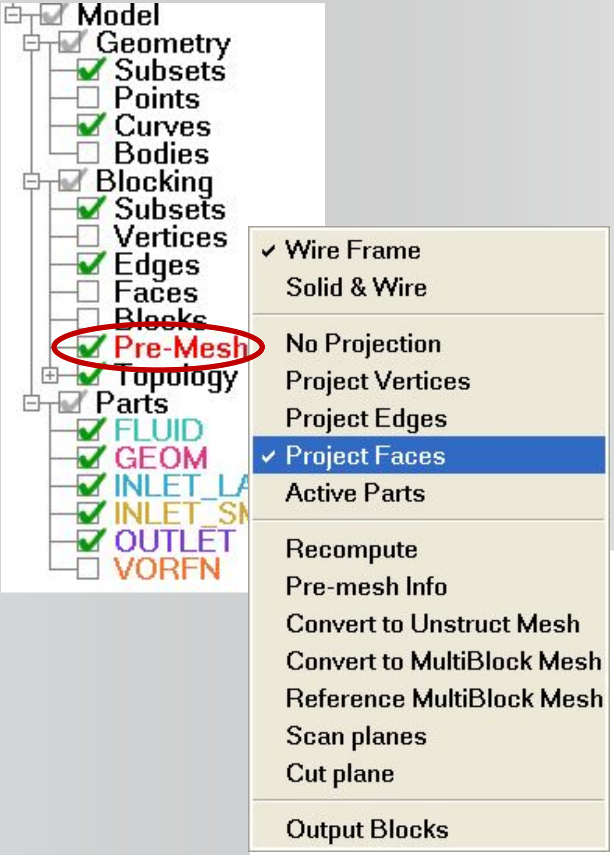
- The mesh sizes on the geometry need to be transferred to the blocking edges
  - Select ***Blocking > Pre-Mesh Params > Update Sizes***
  - Keep the default of ***Update All***
    - This will update distributions and node counts
  - Right click on ***Edges > Bunching*** in model tree to show node locations on all edges
  - ***Apply***
    - Notice the tick marks that appear on the edges



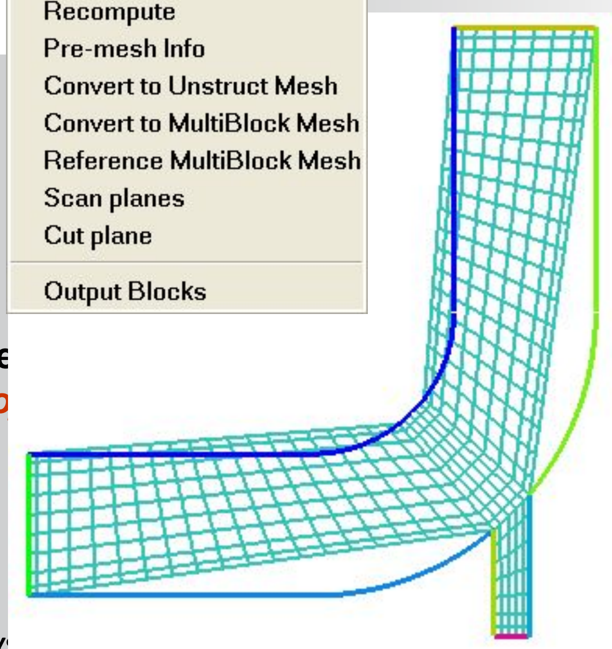


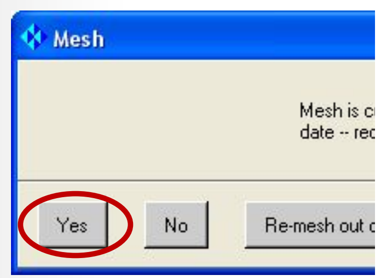
Compute Pre-Mesh

- Select ***Pre-Mesh*** in the model tree
- Select ***Yes*** to “recompute” mesh
- Right clicking on ***Pre-mesh*** will show 4 projection methods
- The selected projection method will also do all methods listed above it (except ***no projection***)
  - i.e. ***Project faces*** will project faces, edges, and vertices
- If no surfaces are present then ***Project faces*** will only project edges and vertices
  - Same result as for ***Project edges*** for 2D models

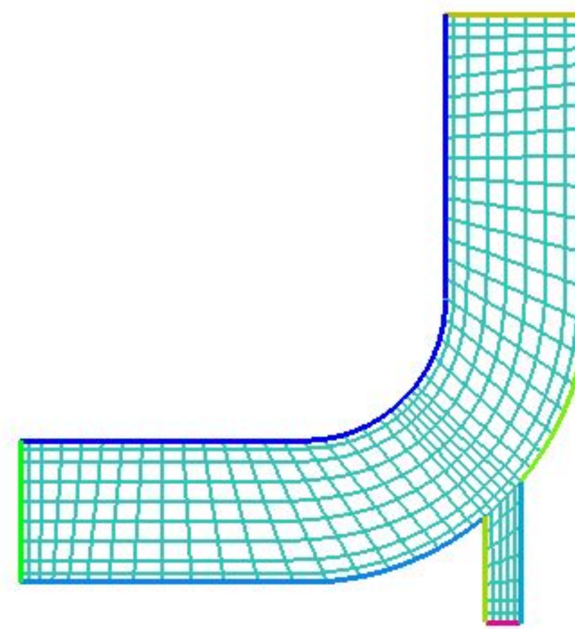


Pre me
No pro



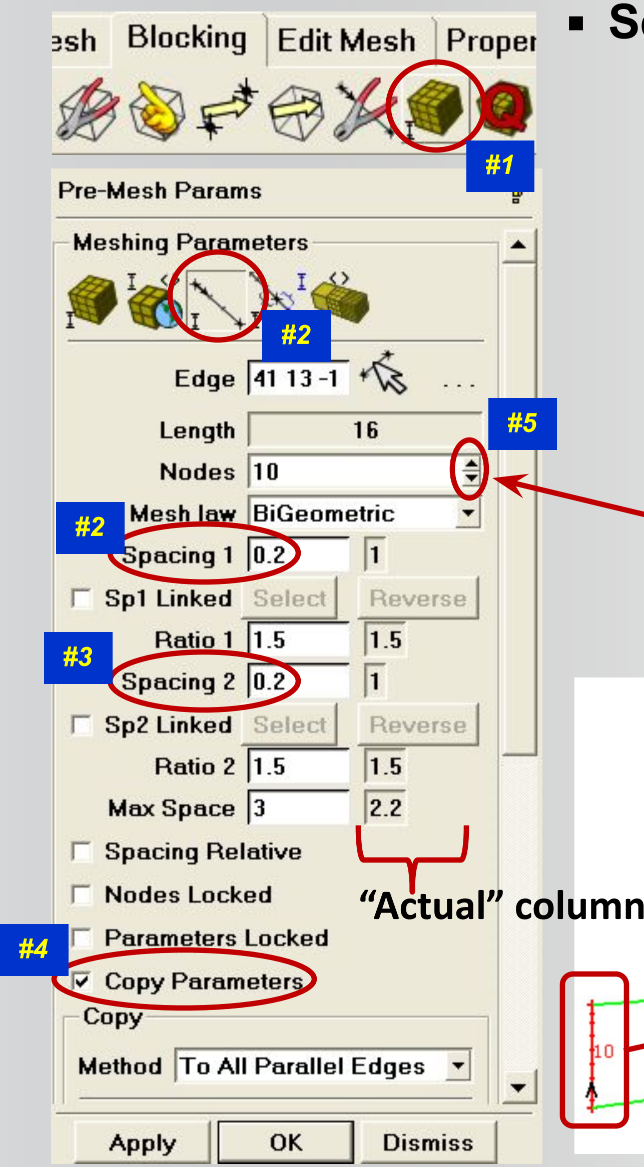


Pre mesh with
Project edges **or**
Project faces

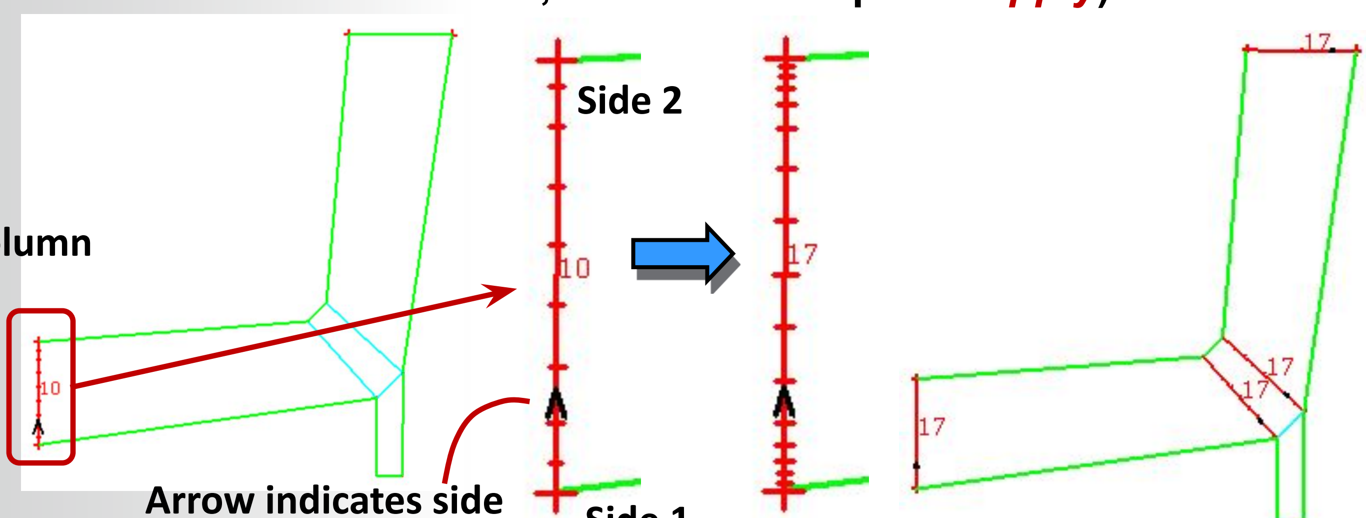


ANSYS®

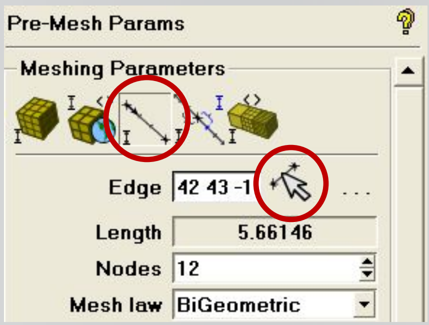
Edge Parameters



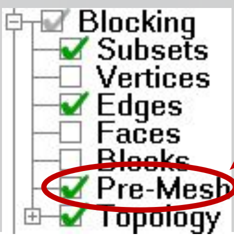
- Select ***Blocking > Pre-Mesh Params > Edge Params***
  - Press the ***Select edge(s)*** button and select the edge at the far –X side
  - Set ***Spacing 1*** = ***0.2***, ***Spacing 2*** = ***0.2***
    - The arrow indicates which side has spacing/ratio 1 and which side has spacing/ratio 2 as marked below
  - Turn on ***Copy Parameters***, with method set ***To All Parallel Edges***
  - Use the arrows to increase the nodes until the “actual” column of ***spacing 1*** and ***spacing 2*** meet the requested value of 1.5 (17 nodes) (The arrows will ***Apply*** the function each time, so no need to press ***Apply***)



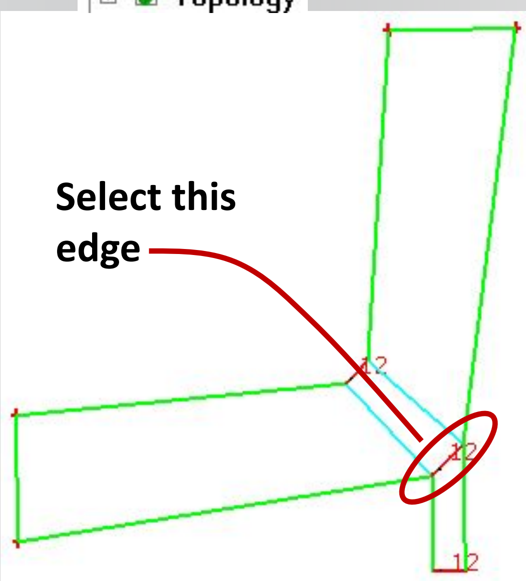
Edge Parameters (Continued)

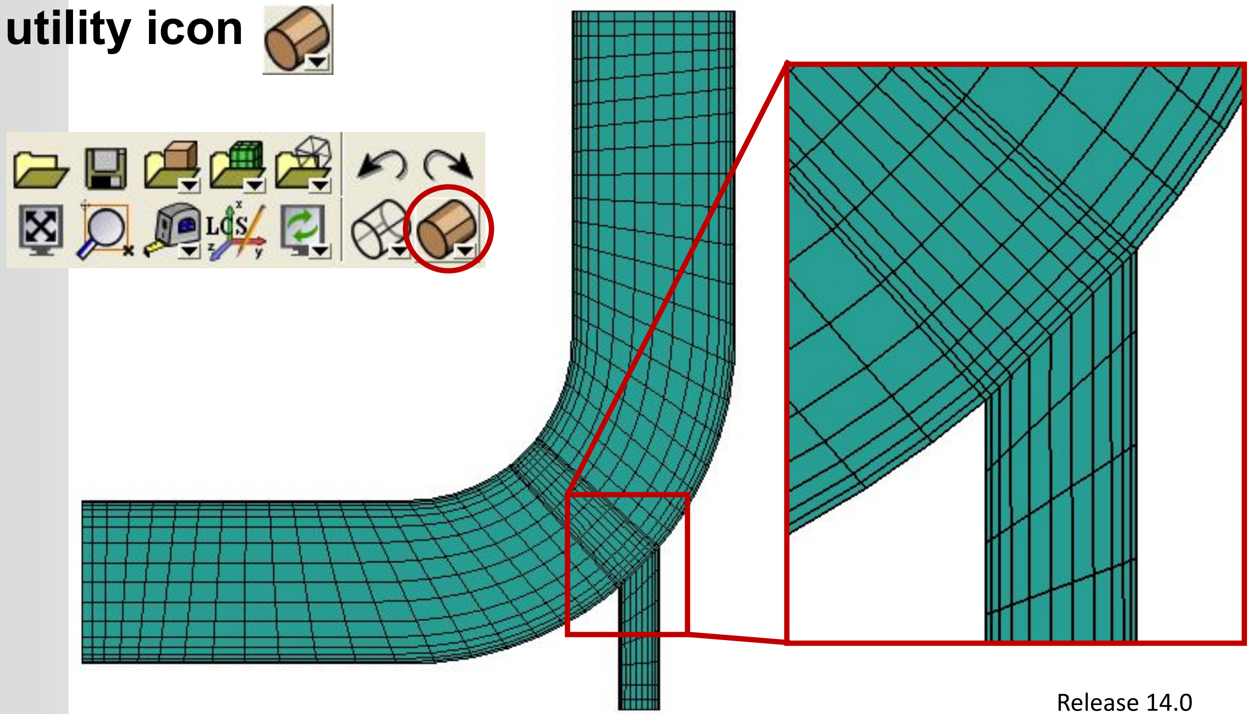


- Press the ***Select edge(s)*** button and select the edge shown
  - Set ***Spacing 1*** = ***0.2***, ***Spacing 2*** = ***0.2***
  - With ***Copy Parameters*** ON, use the arrows to increase the nodes until the "actual" column of ***spacing 1*** and ***spacing 2*** meet the requested value of 1.5 (12 nodes)

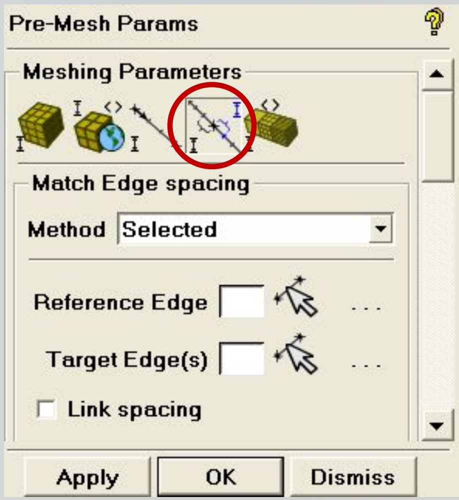


- Turn ***Pre-mesh*** off then on again to recompute
- Right mouse click on ***Pre-Mesh > Solid & Wire*** or press the utility icon

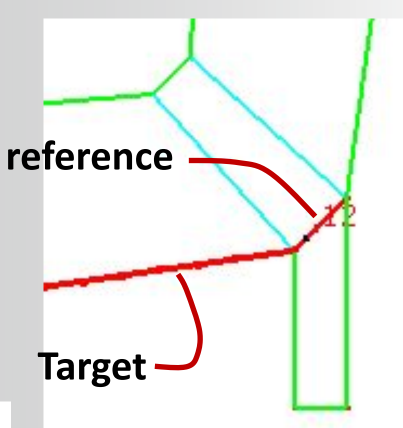


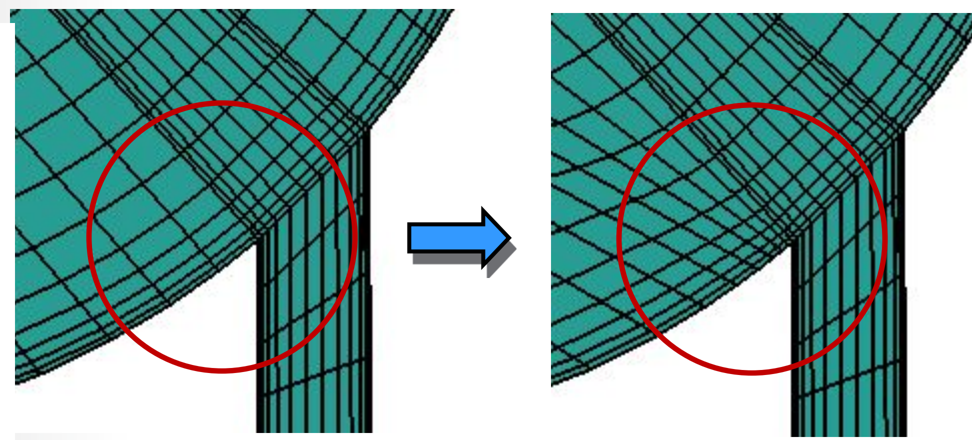


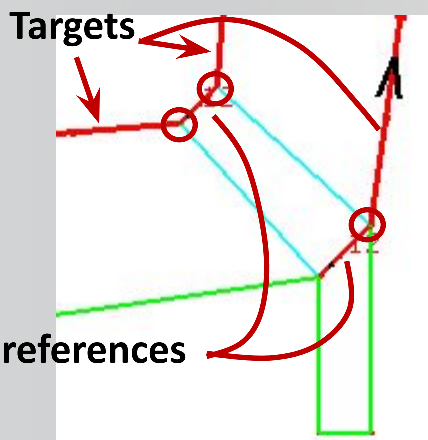
Matching Edges



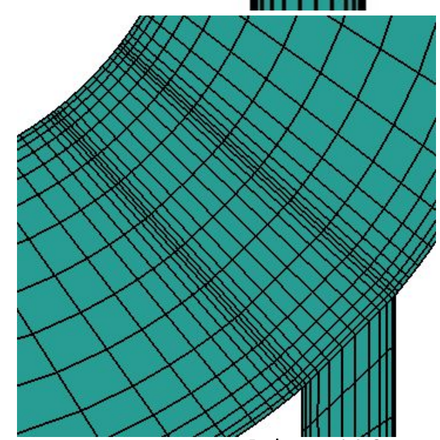
- Select ***Blocking > Pre-Mesh Params > Match Edges***
  - Select the ***Reference Edge*** as shown
  - Then select the ***Target Edge*** as shown, and middle mouse click twice to complete and exit selection
  - Turn ***Pre-mesh*** on (or off and on again if already on)





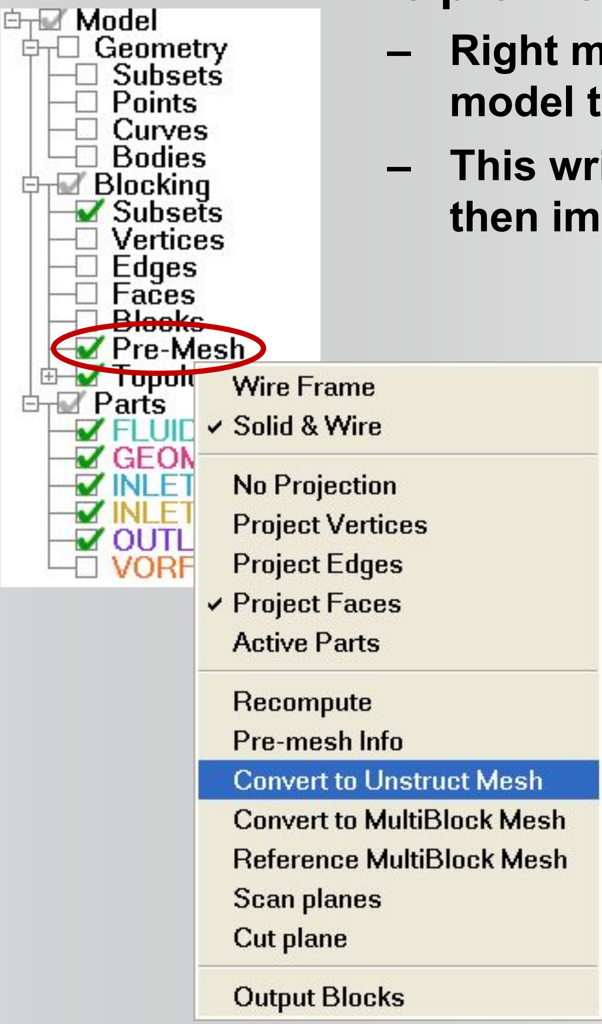


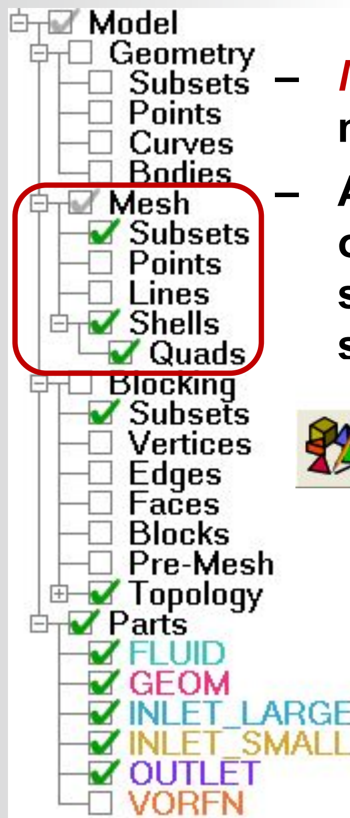
  - Repeat across the 3 other vertices
  - This function matches edge end spacings (spacing 1 and spacing 2) across a vertex
  - The reference edge should always have the smaller end spacing
  - Toggle ***Pre-mesh*** off then on to recompute



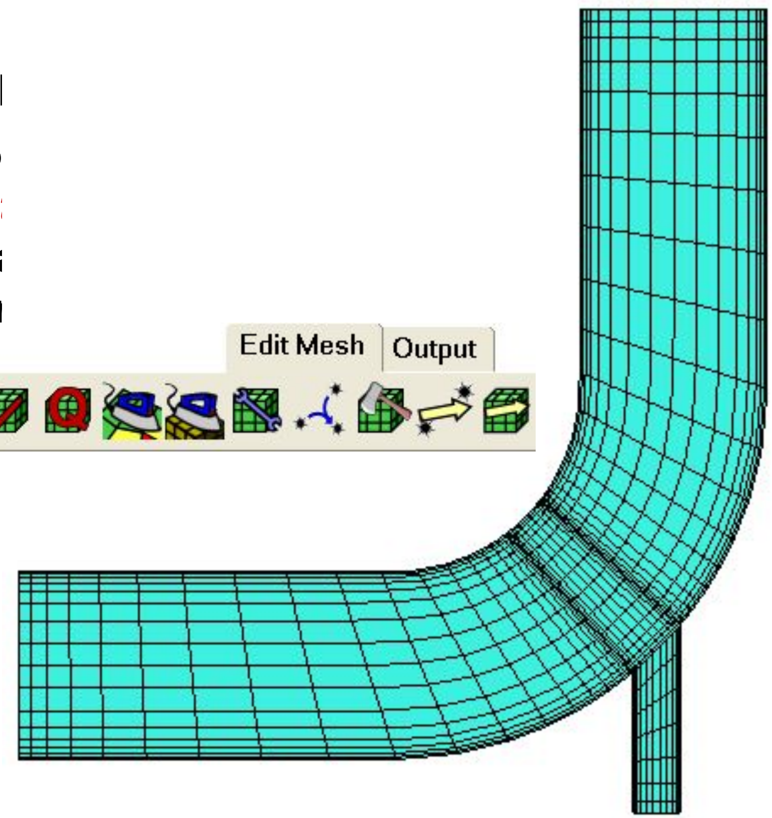
Convert to Unstructured Mesh

- **The pre-mesh cannot be written to the solver yet**
  - **Right mouse click on *Pre-Mesh > Convert to Unstruct Mesh* in the model tree**
  - **This writes the default name *hex.uns* to the working directory, then immediately loads the mesh**

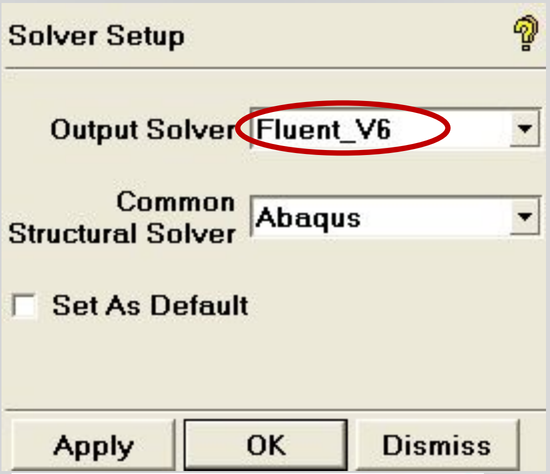


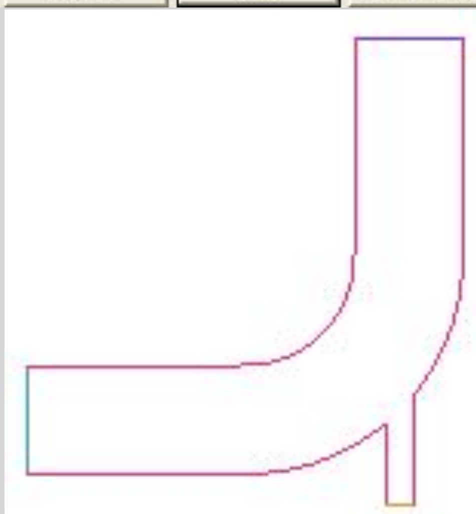


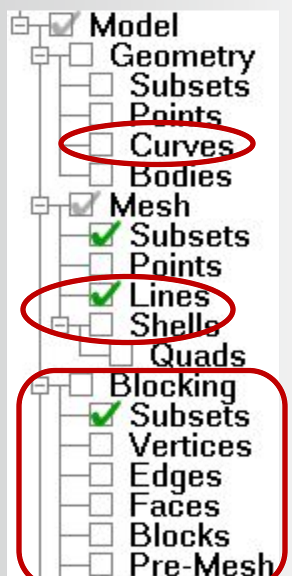
- ***Mesh***
  model
- **Any o**
  or *Ou*
  such
  solve



Select Solver

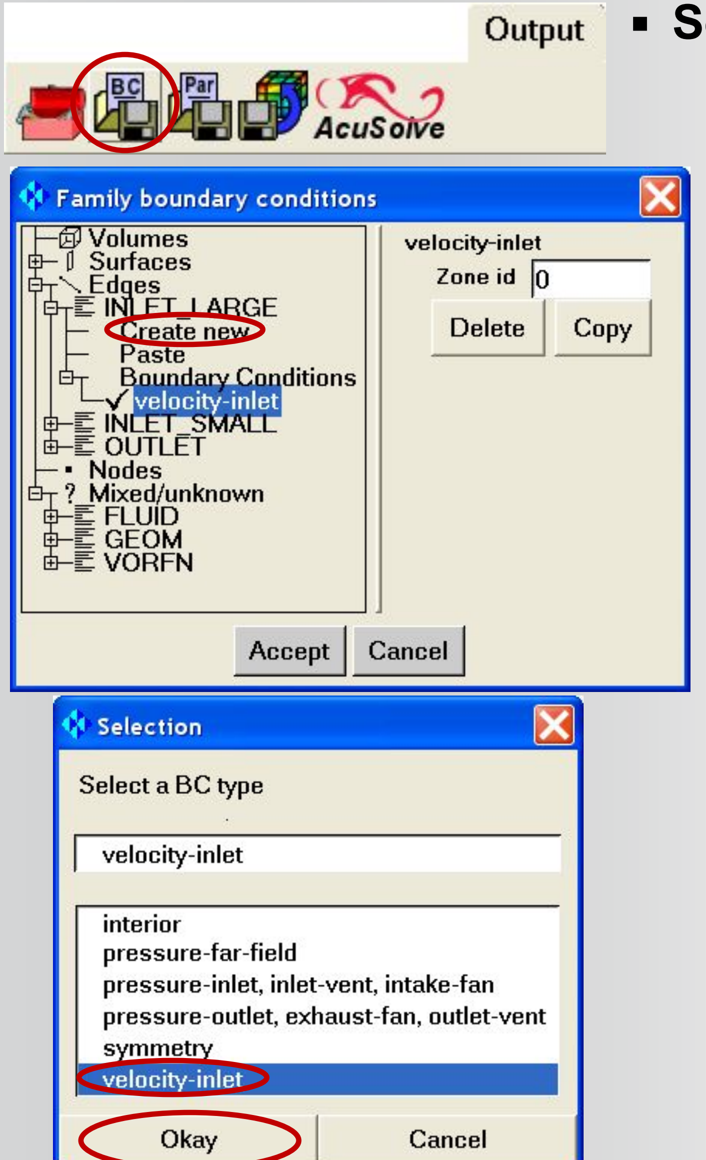






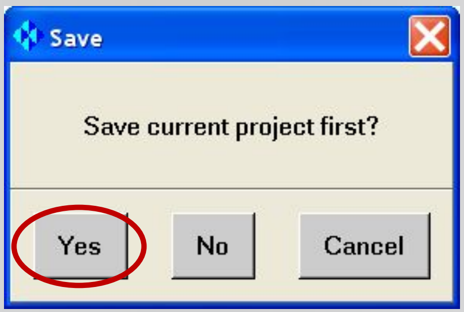
- Select ***Output > Select Solver***
  - Press the pulldown arrow next to the ***Output Solver*** and choose ***Fluent_V6***
    - The ***Common Structural Solver*** is only for the solvers Nastran, Ansys, LS-Dyna, Abaqus, and Autodyn, and allows additional pre-processing handling that is available in the ***Properties***, ***Constraints***, ***Loads***, and ***Solve Options*** tabs
  - ***Apply***
  - Turn OFF ***Mesh > Shells*** and ***Geometry > Curves***, and turn ON ***Mesh > Lines*** in the model tree
  - Turn OFF all ***Blocking*** entities, or just save and close the blocking since we are done with it
  - The ***Line*** elements are the boundary elements resulting from edge to curve associations
  - We will set boundary conditions on these

Boundary Conditions



- Select ***Output > Boundary Conditions***
  - The tree structure is organized according to the dimensions of geometry and mesh entities in parts
    - ***Volumes*** = bodies and 3D elements
    - ***Surfaces*** = surfaces and shell (2D) elements
    - ***Edges*** = Curves and Line (1D) elements
    - ***Nodes*** = Points and Node (0D) elements
    - Multiple dimension parts are in ***Mixed/unknown***
  - Expand under ***Edges > INLET_LARGE***, and press ***Create new***
  - Select ***velocity-inlet***, then ***Okay***
  - Repeat for ***INLET_SMALL***
  - Expand under ***Edges > OUTLET***, and press ***Create new***
  - Select ***pressure-outlet, exhaust-fan, outlet-vent***, then ***Okay***
  - Expand under ***Surfaces > Mixed/unknown > FLUID***, and press ***Create new***
  - Select ***fluid*** for the BC, then ***Okay***
  - ***Accept***

Write Input File



- Select *Output > Write/View Input*
  - It will first ask you to save the attribute file (*.atr)
    - This is the structural name for the boundary condition file (*.fbc), and both will be saved. Save this file. It is OK to rename it.
  - Secondly, it will ask you to save the project. Choose *Yes* to this also.
  - Third, it will ask you to select the mesh to write to the solver. Choose the project name with .uns appended. It is OK to choose *hex.uns* if you made no changes to the mesh after blocking
  - The last menu has a few options which will be different for every solver. Press *Done*.