

# Workshop 8.3 3D Pipe Junction O-grid

14.0 Release

The background features a series of overlapping gears in shades of purple and blue. To the left, there are wavy, translucent blue lines representing fluid flow. In the center, a green circular graphic with concentric rings is visible. To the right, there are several 3D rectangular blocks in shades of teal and black, arranged in a stepped fashion.

Fluid Dynamics

Structural Mechanics

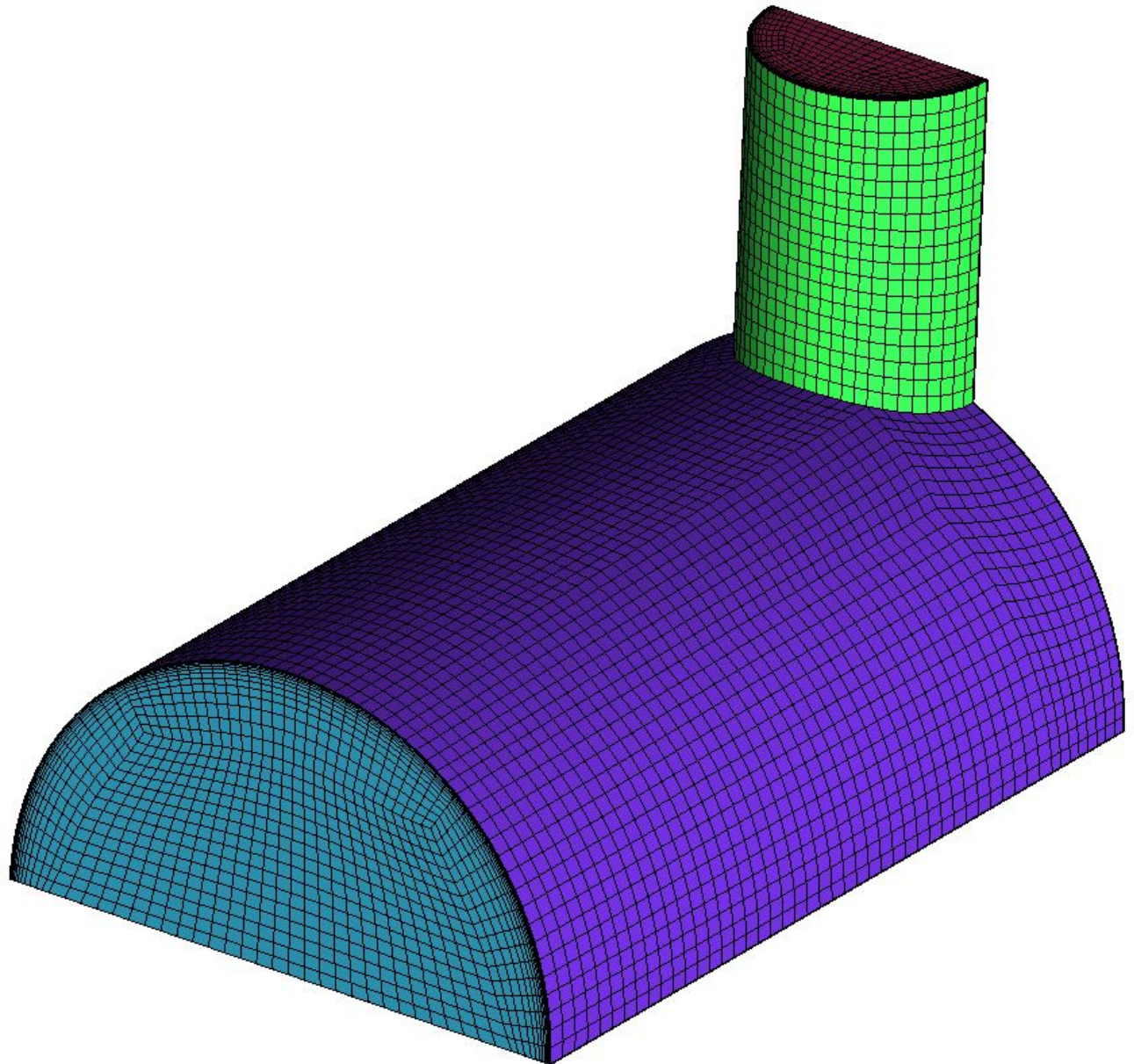
Electromagnetics

Systems and Multiphysics

## Introduction to ANSYS ICEM CFD

# 3D Pipe Junction - Ogrid

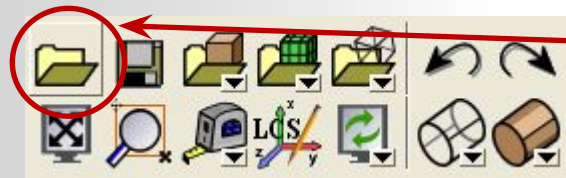
- 3D Pipe Junction -
  - In this workshop we will cover the steps for creating the mesh and blocking for the 3D Pipe Junction
  - Check quality to ensure a good mesh
  - Create an Ogrid for the pipe junction
  - Rescale Ogrid to match the mesh quality
  - Convert mesh to ANSYS format
  - Write mesh to CDB file



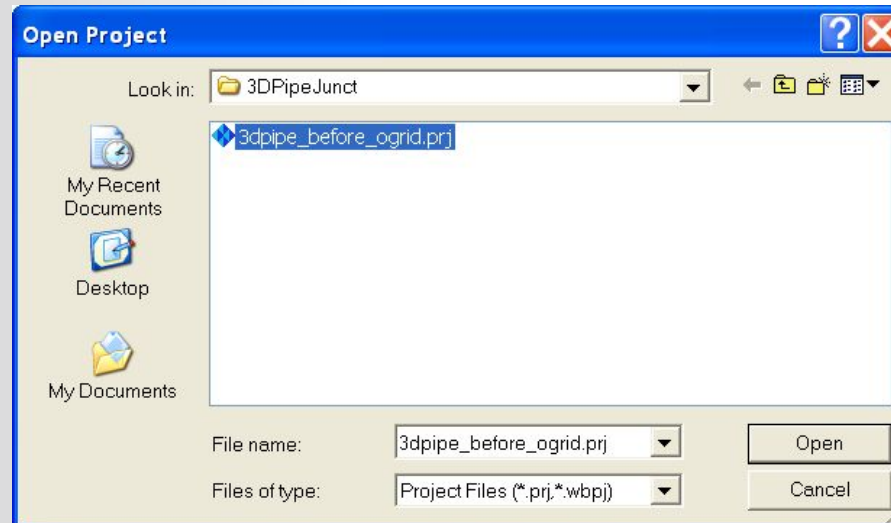
# Open Project

- **File > Open Project...**

- If you feel confident you made the blocking correctly with the previous project, **7.3, 3D Pipe Junction**, then open that project
- Otherwise, open the project provided, **3dpipe\_before\_ogrid.prj** in **3DPipeJunct** folder
- This will open the geometry and blocking

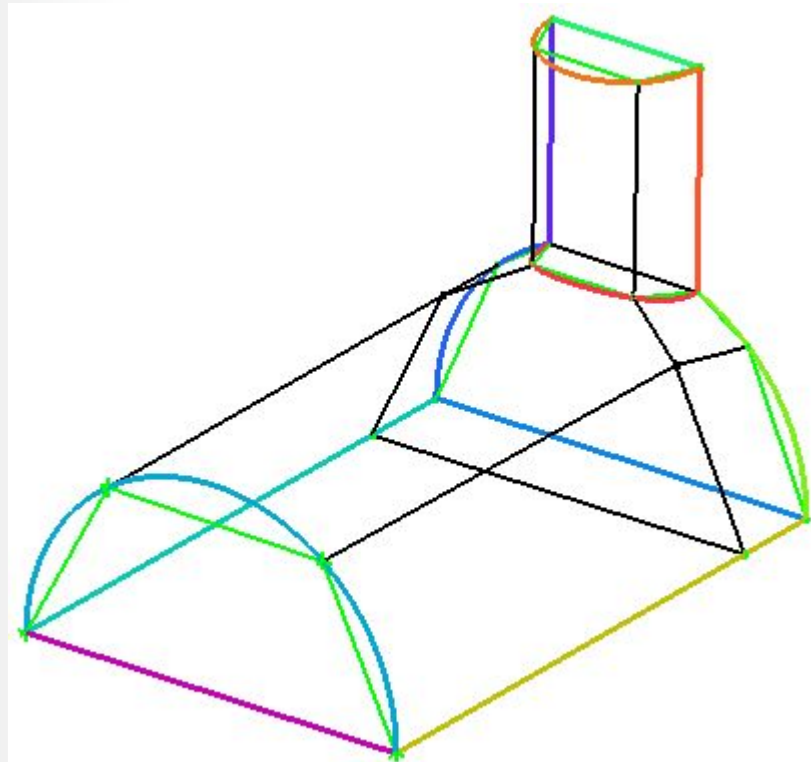
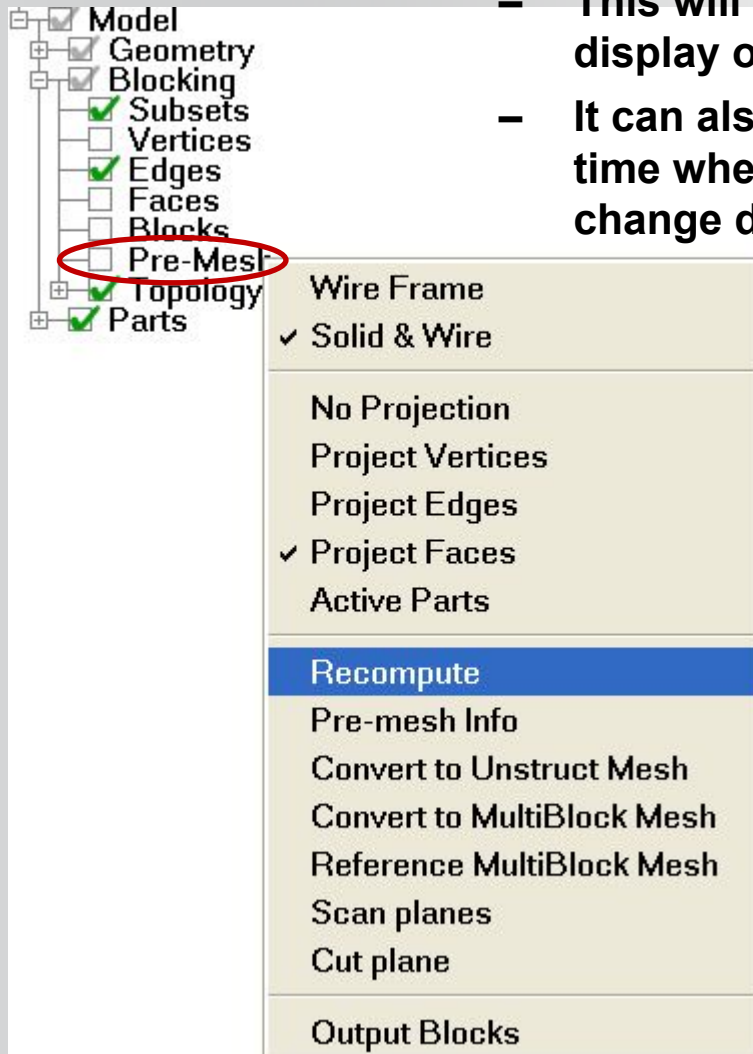


Or use the utility icon to open project



# Compute Pre-Mesh

- Right click on **Blocking > Pre-Mesh > Recompute**
  - This will compute the pre-mesh without turning the mesh display on
  - It can also be used to force a compute of the mesh at any time when turning on **Pre-Mesh** does not recognize a change done in any particular blocking operation

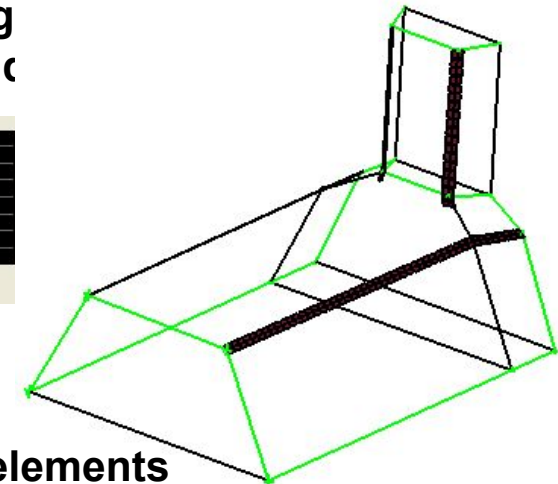
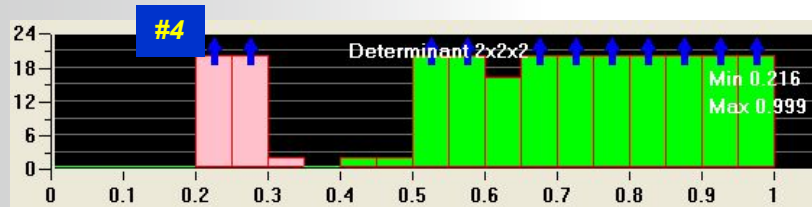


# Quality Check



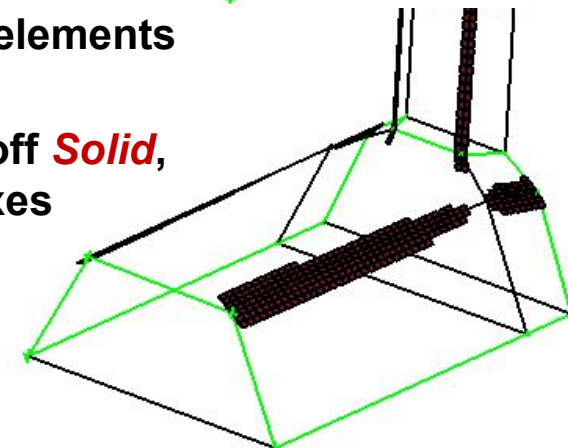
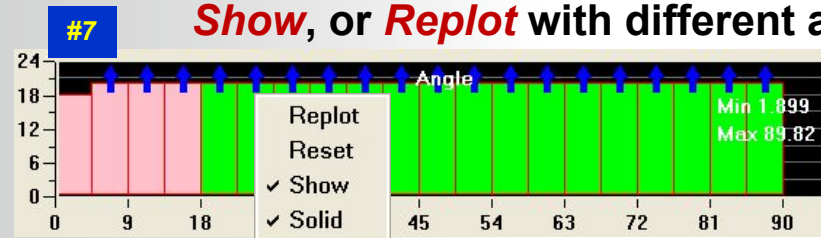
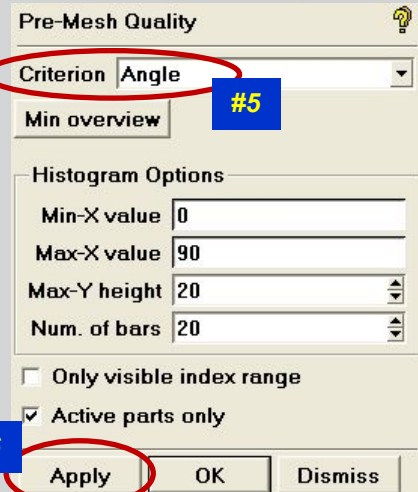
## Blocking > Pre-Mesh Quality Histograms

- Set the Criterion to **Determinant 2x2x2**
- **Apply**
- A good mesh should have little or no elements below the 0.2 – 0.3 range elements below this and



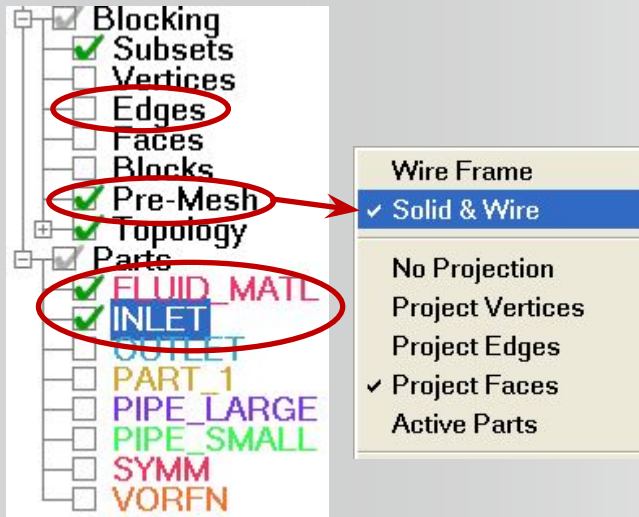
## Change criterion to **Angle**

- **Apply**
- A good mesh should have no elements below 18 degrees, so select the elements below this and inspect
- Right click in histogram to turn off **Solid**, **Show**, or **Replot** with different axes

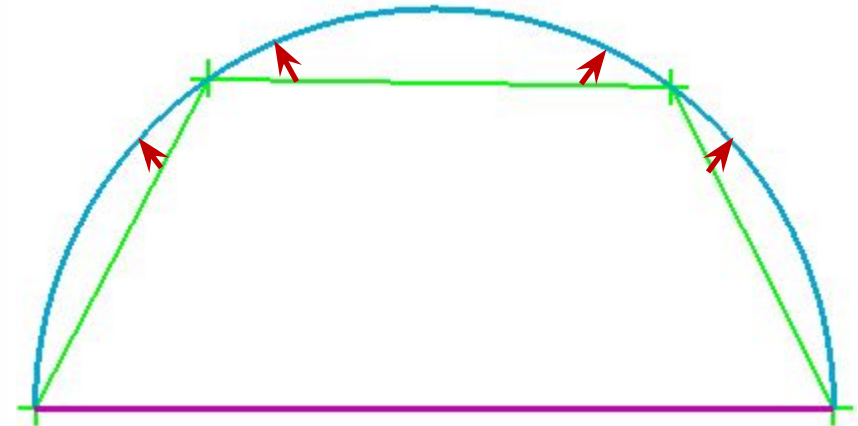
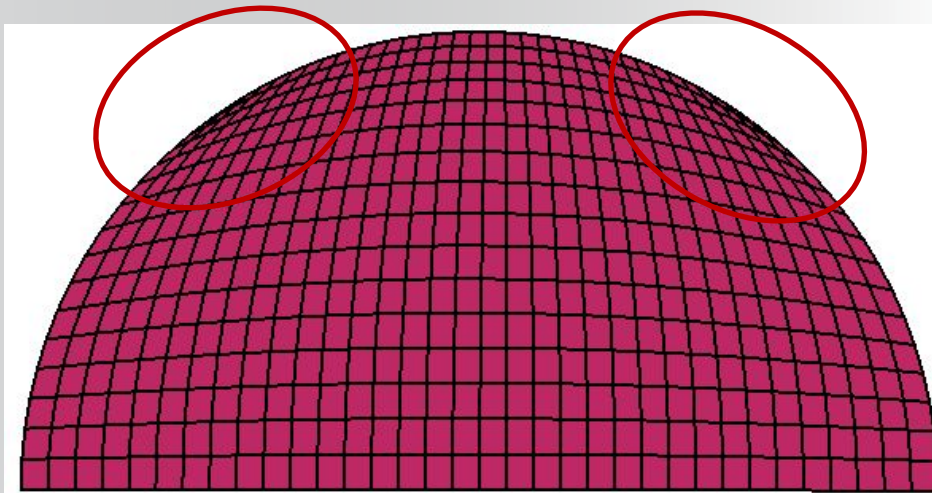


**Done** will close histogram

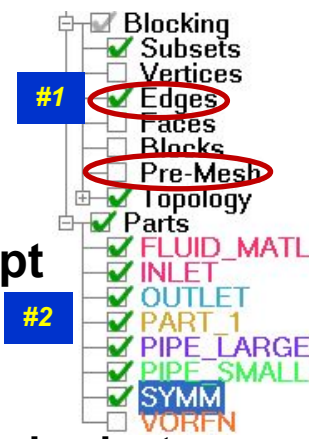
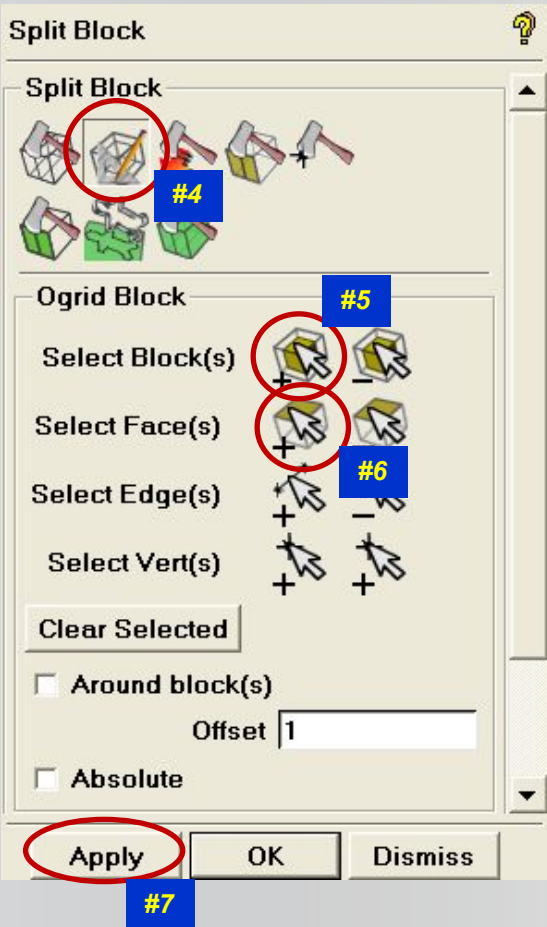
# Surface Mesh Display



- The problem mesh is easily seen by looking at just the surface mesh on the **INLET** surface
  - Turn off all parts except **FLUID\_MATL** (contains the blocks) and **INLET**
  - Turn on **Pre-mesh**
  - Turn off **Edges**
  - Right click to turn on **Pre-Mesh > Solid & Wire**
- This occurs in the block corners because the edges project to the curves, and the curve meets tangentially at these block corners



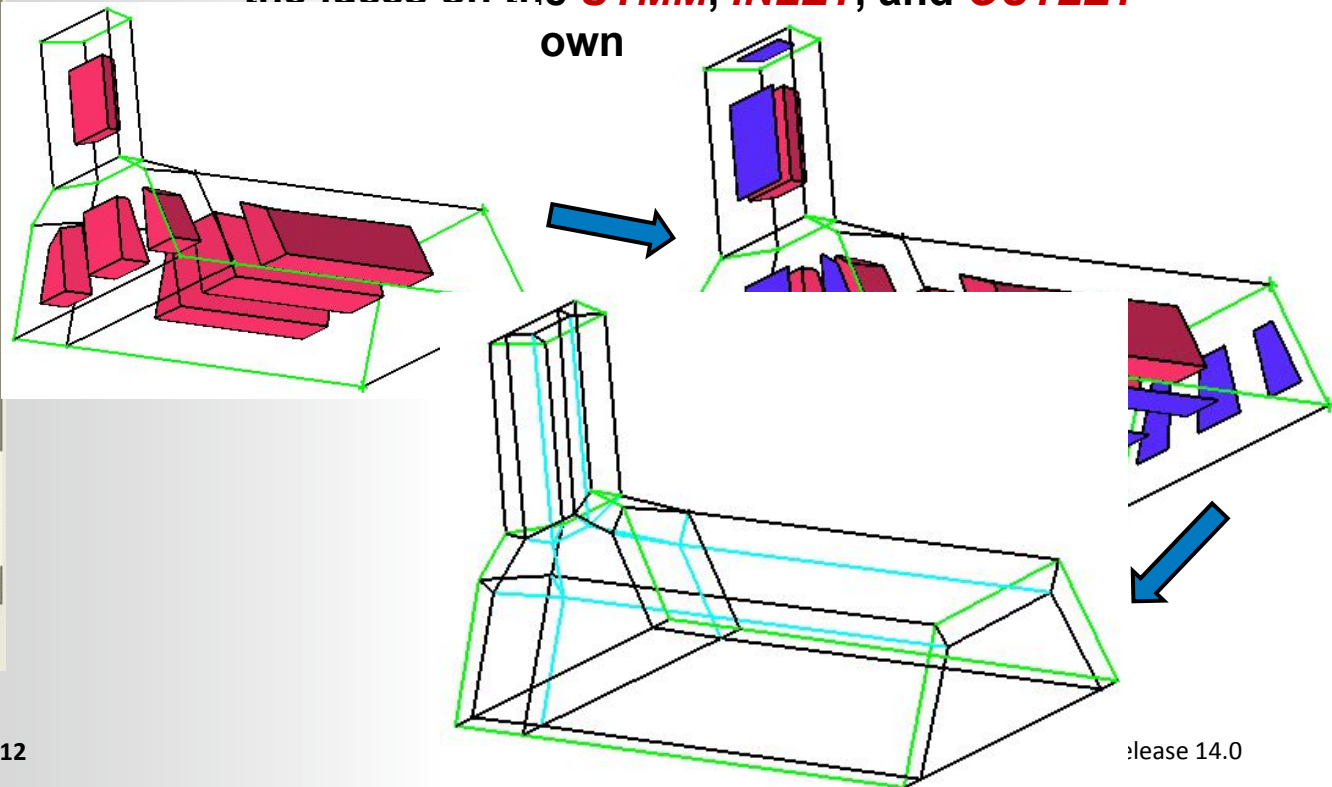
# Create Ogrid



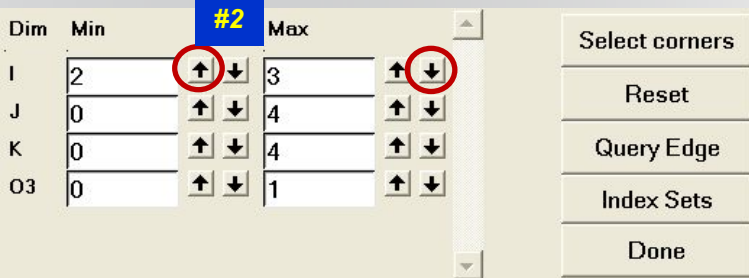
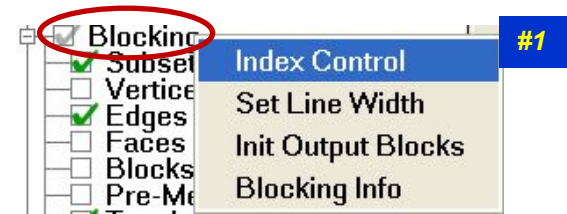
- First, turn ON **Edges** and all parts (except **VORFN**), and turn OFF **Pre-Mesh**

- **Blocking > Split Block > Ogrid Block**

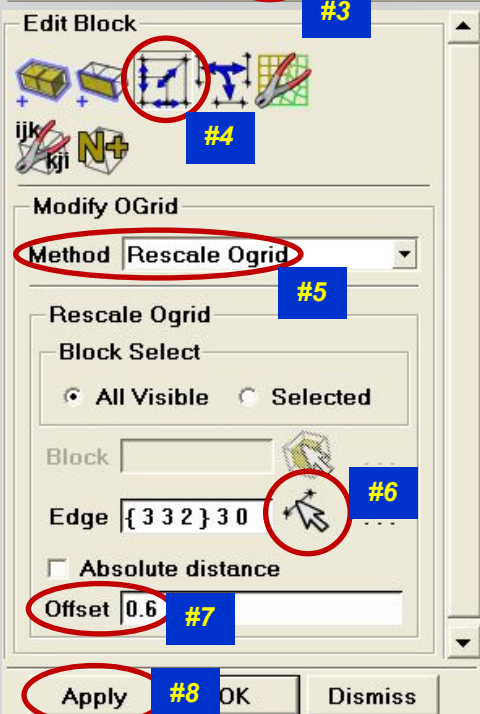
- Press the **Select block(s)** button, and select all the blocks (can box select or press 'v')
- Press the **Select face(s)** button, and select the faces on the **SYMM**, **INLET**, and **OUTLET**



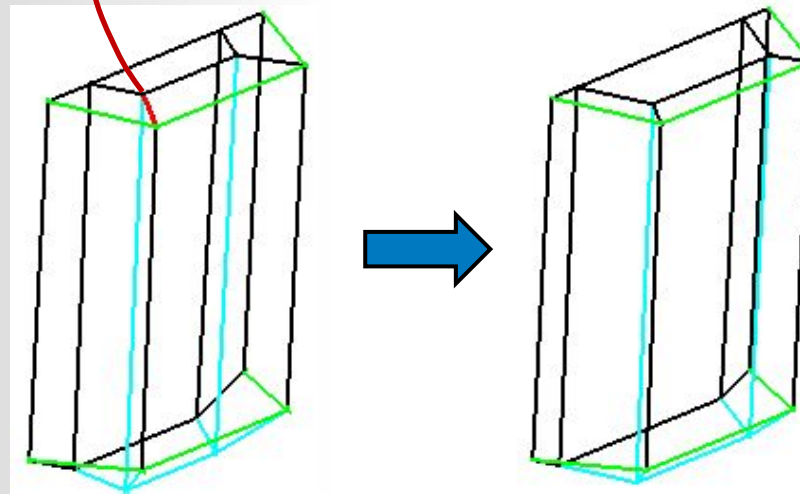
# Rescale Ogrid



- Rescaling Ogrid is one of the commands that can work on only visible vertices
- Right click on **Blocking > Index Control** in model tree



- Set *I* from 2 to 3 by using the arrows
- **Blocking > Edit Block > Modify Ogrid**
  - **Method** set to **Rescale Ogrid**
  - Press **Select edge(s)**
  - Select any of the small “radial” edges
  - Enter **0.6** for the **Offset**
  - **Apply**, then press **Reset** in the **Index Control**



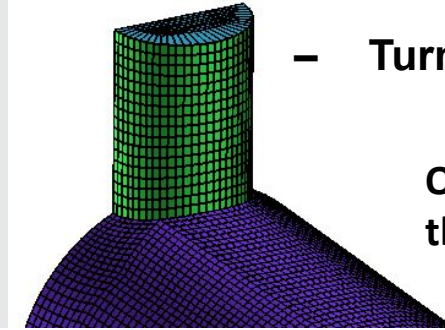
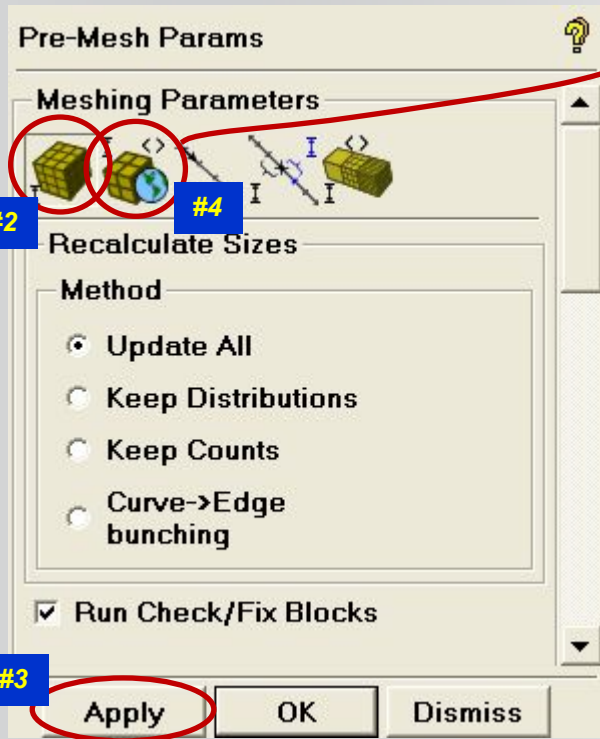
Every visible edge of the Ogrid will be 0.6 of its original length



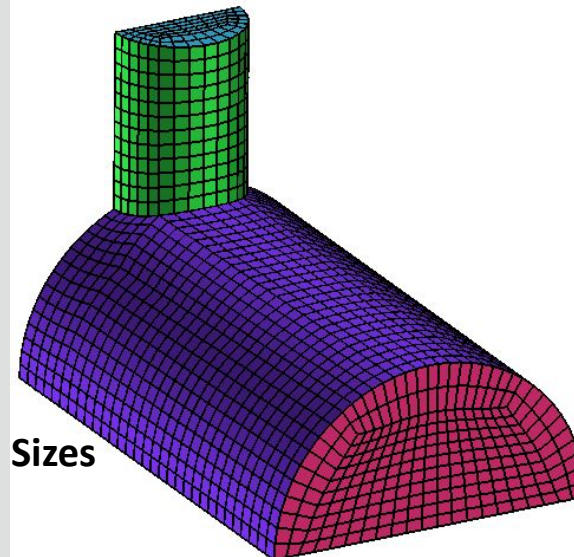
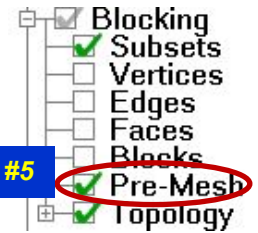
# Update Sizes



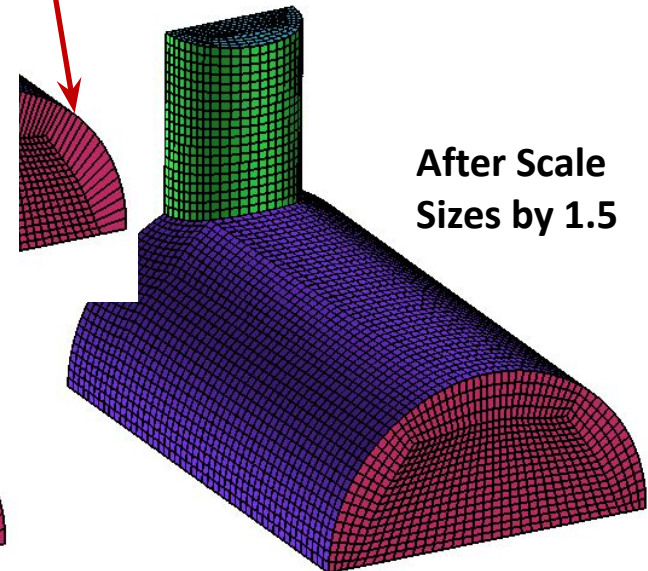
- Any new edges have 2 nodes by default
- Blocking > Pre-Mesh Params > Update Sizes**
  - Transfers surface and curves sizes to edges
  - Apply**
- You will need to redo the **Scale Sizes** by **1.5**



Turn on **Pre-Mesh**

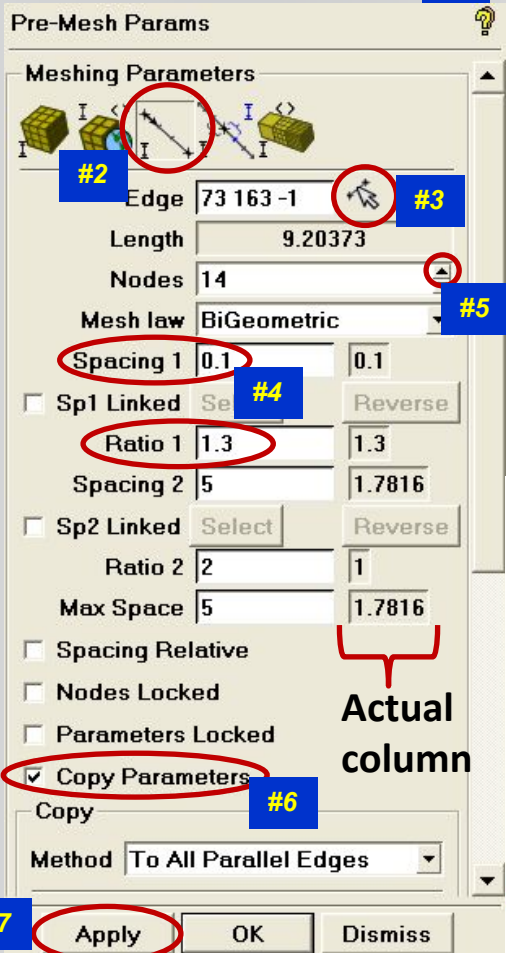


After Update Sizes




After Scale Sizes by 1.5

# Set Edge Parameters for Boundary Layer

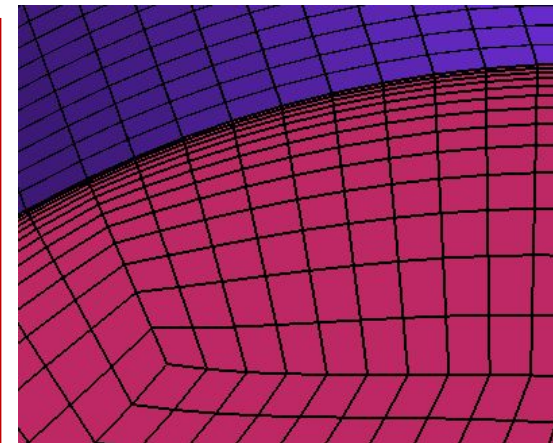
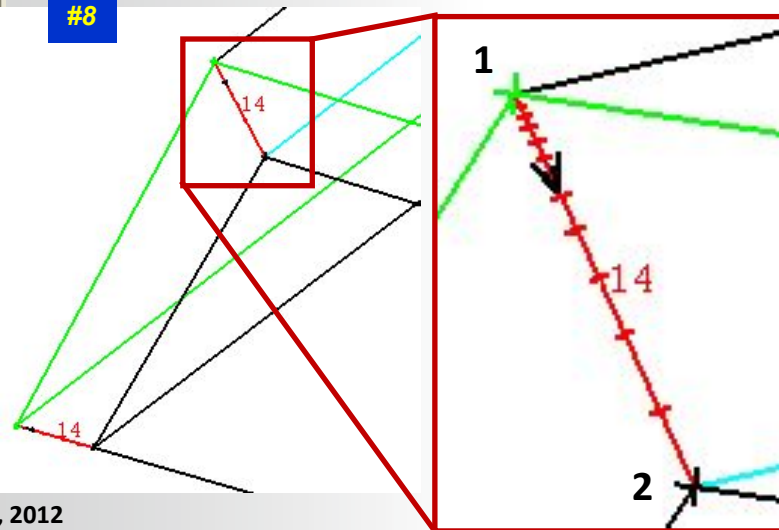
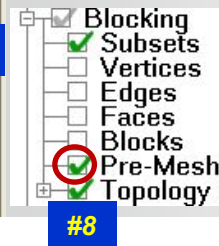


## Blocking > Pre-Mesh Params > Edge Params

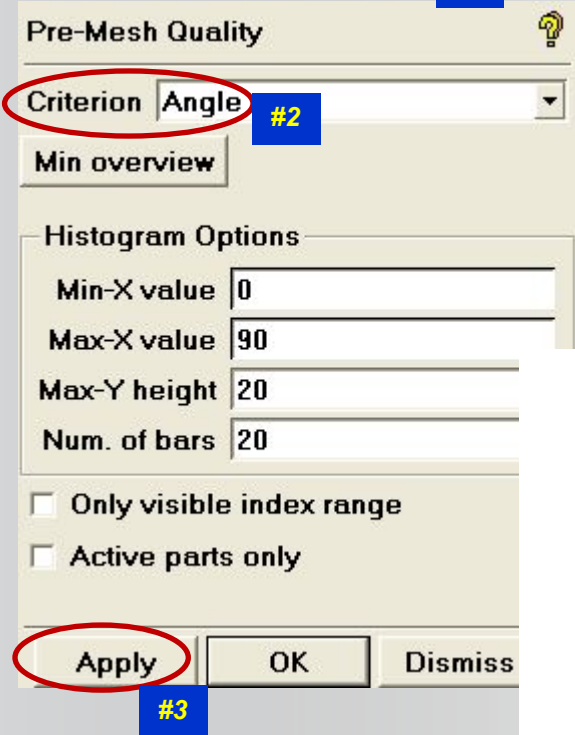
- Do NOT **Update Sizes** after setting **Edge Parameters**, or it will destroy any work you did setting sizes and distributions on edges
- Select  any of the radial edges of the Ogrid
- Set **Spacing 1** to **0.1** and **Ratio 1** to **1.3**
- Increase the nodes until **Ratio 1** in the actual column gets near the requested **Ratio 1**, about 14 nodes

Enable **Copy Parameters** and **Apply**

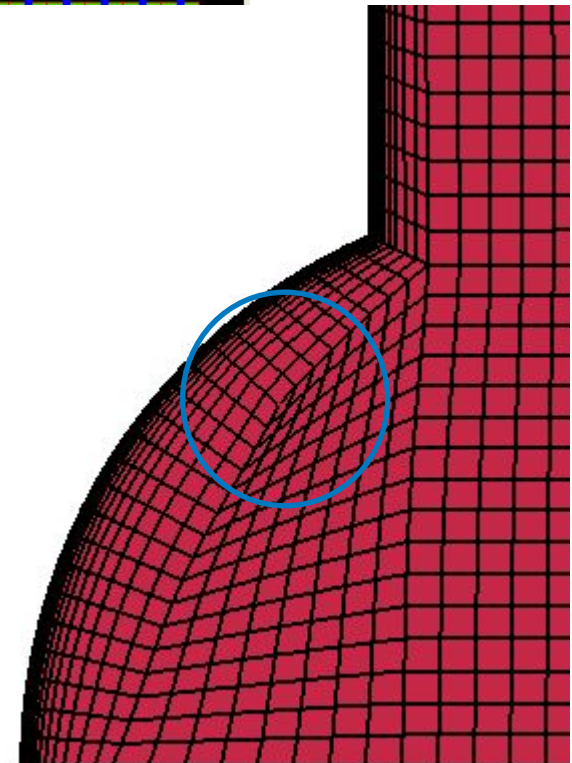
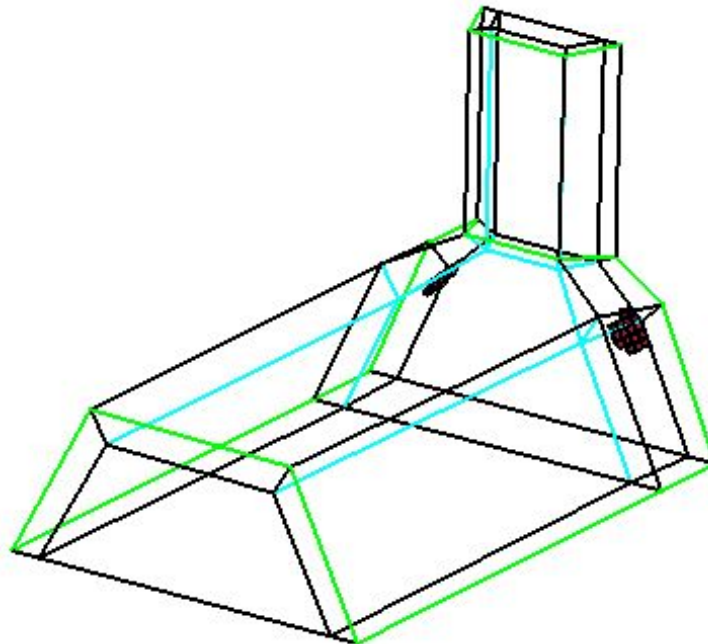
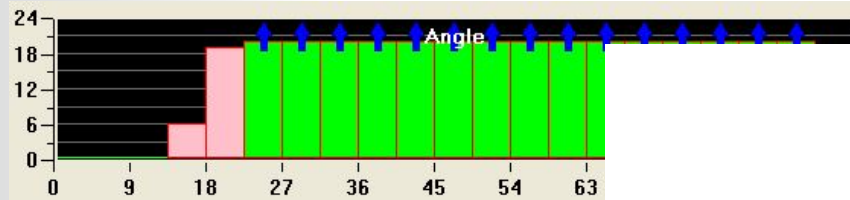
Recompute pre-mesh (turn **Pre-Mesh** OFF then ON)



# Recheck Quality



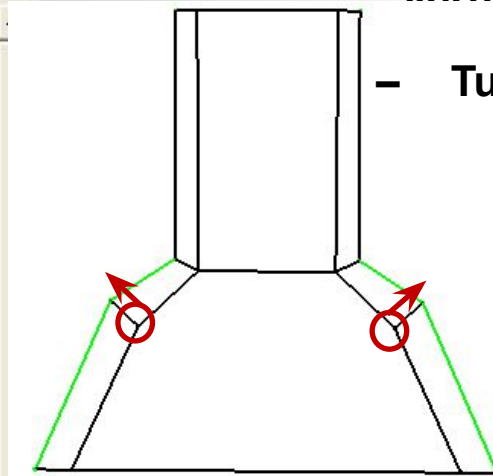
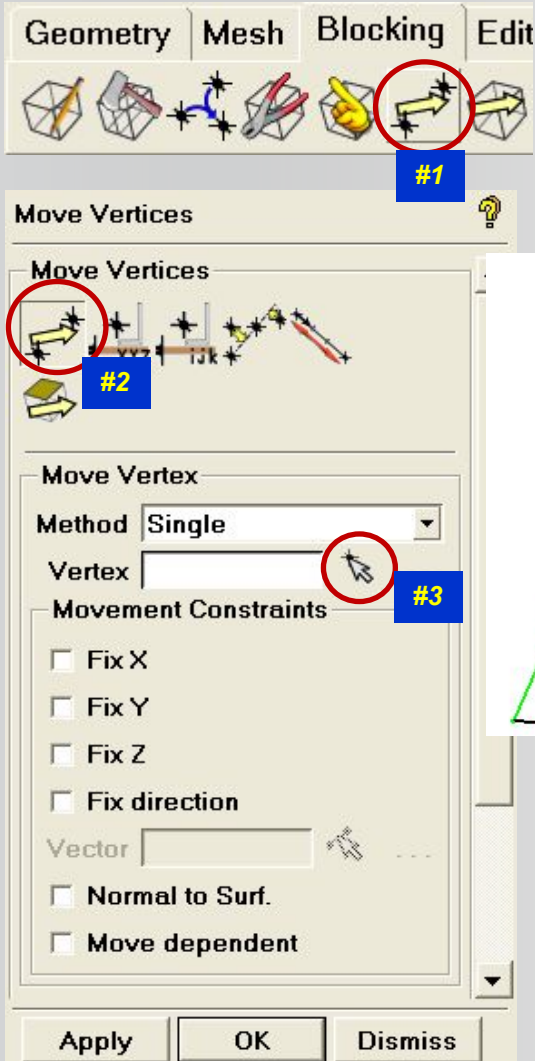
- Check determinant and angle again
- **Blocking > Pre-mesh Quality Histograms**
  - Check **Determinant 2x2x2** then **Angle**
  - Highlight **Angles** above and below 18 degrees
  - Turn off **Pre-Mesh** to see where they are at



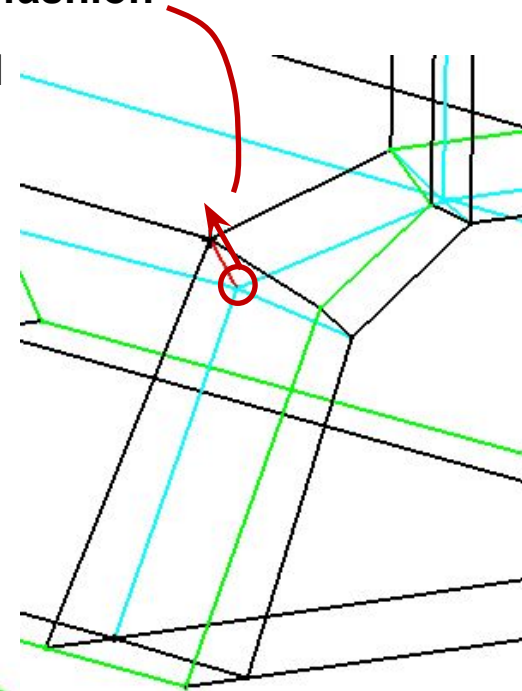
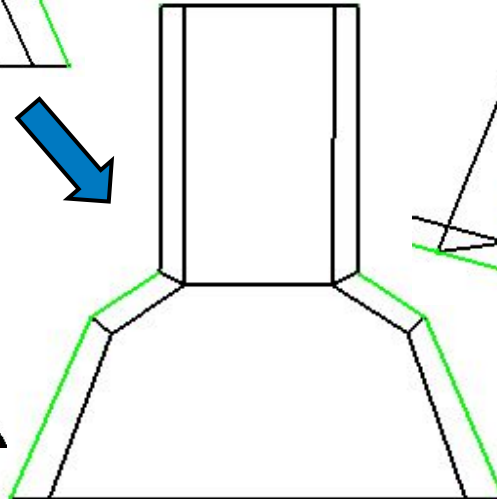
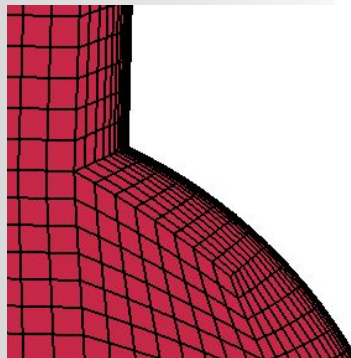
# Move Vertex

## Blocking > Move Vertex > Move Vertex

- With the **Method** set to **Single**, press the **Select vert(s)** button and drag the 2 vertices as shown
- You'll also want to drag the nearby 2 internal vertices in the same fashion



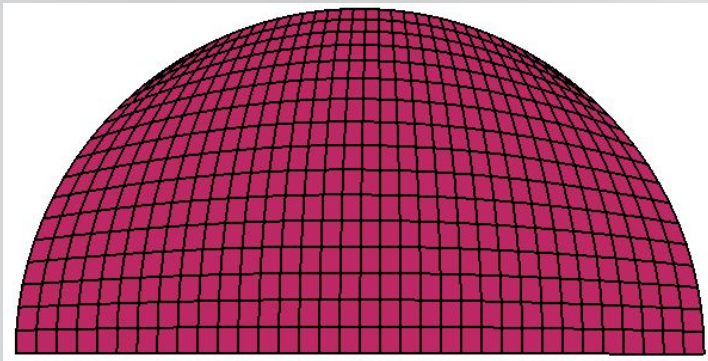
- Turn **Pre-mesh** ON



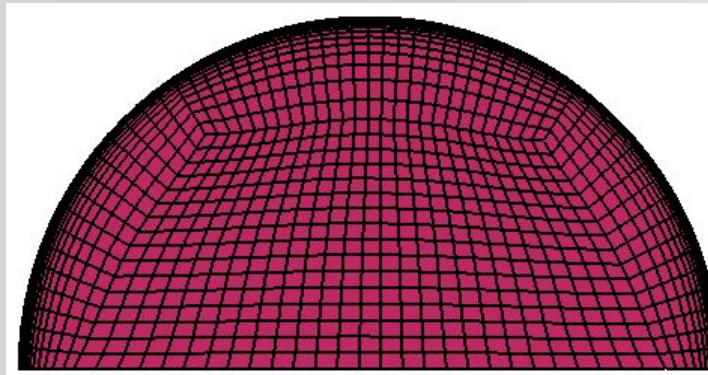
# Quality compare

- RE-check Determinant and Angle

- Mesh quality is greatly improved after Ogrid

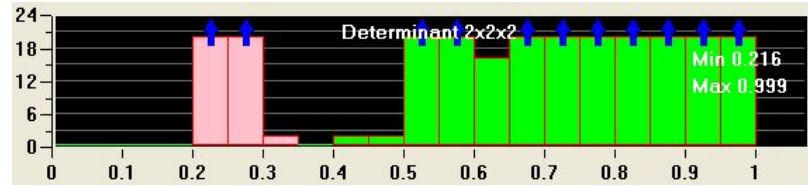


Before Ogrid Creation

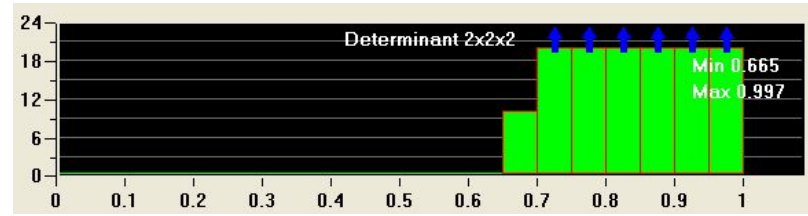


After Ogrid Creation

Initial Determinant

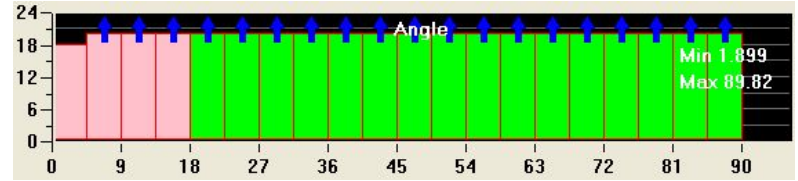


Improved Determinant

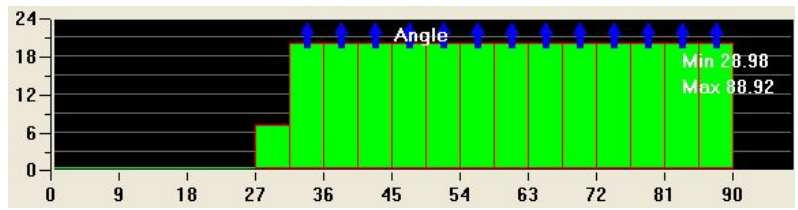


Minimum determinant should be around 0.2 or 0.3. It is well over this.

Initial Angle



Improved Angle



Minimum angle should be 18 degrees. It is decently over this too.

# Write Mesh to Solver

- First convert pre-mesh to unstructured or multiblock mesh depending on what type the solver uses (CFX uses unstructured)
  - Right click on *Pre-mesh* > *Convert to Unstruct Mesh* in the model tree
    - This writes *hex.uns* to the working directory and immediately loads it
  
- Select *Output* > *Select Solver*
  - Pick *ANSYS CFX* from the *Output Solver* list
  - *Apply*
  - *Select Output* > *Write Input*
    - Select *Yes* when prompted to save project or boundary condition file
    - Specify the *Output CFX5 file* name or use default
    - *Done* (use defaults)

